



**Creo Elements/Direct
Drafting User's Guide:
Fluent User Interface**
Creo Elements/Direct Drafting 20.1

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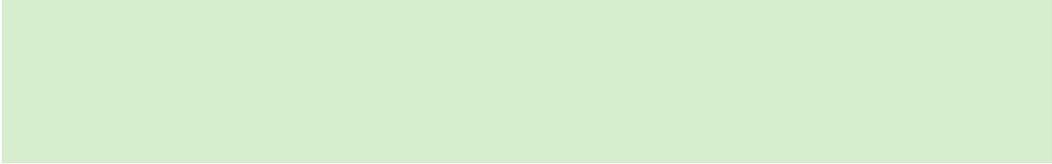
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Preface

Creo Elements/Direct Drafting is a versatile 2D design and drafting system for optimizing each stage of the design process. Using Creo Elements/Direct Drafting, you can quickly and easily create and modify 2D drawings.

Creo Elements/Direct Drafting has a full set of commands for constructing, modifying, viewing, measuring, and plotting a 2D drawing.

Using a mouse or other supported digitizer, you have the facility to:

- Perform real time zoom-in and zoom-out.
- Pan in real time, across the entire drawing or a zoomed-in portion.

Who Should Use This Guide

Read this guide if you are a:

- Design engineer
- Industrial designer

The Creo Elements/Direct Drafting documentation set is designed to help you learn to use Creo Elements/Direct Drafting quickly and easily. It consists of:

- Creo Elements/Direct Drafting User's Guide
- Installing and Configuring Creo Elements/Direct Drafting
- Writing Macros with Creo Elements/Direct Drafting
- Creo Elements/Direct Drafting Interfacing

This guide is written for users who have limited experience with CAD software.

Purpose of This Guide

This guide provides information about:

- Setting up Creo Elements/Direct Drafting for your particular environment.
- The Creo Elements/Direct Drafting user interface.
- Using the Creo Elements/Direct Drafting commands and functions.
- Using the dynamic graphic facilities.
- Working with parts and assemblies.
- How to generate hidden lines.

How to Use This Guide

Use this guide for both day-to-day use and reference. Each section contains:

- A description of the command or function.
- The procedure used to perform the given task.
- A tutorial example (when required).
- Any reference material required.

How This Guide is Organized

This guide is organized in the same order as the way you are likely to use Creo Elements/Direct Drafting:

Chapter 1	The Drafting User Interface — provides a brief overview of the system and describes how to use fly by highlighting and handles.
Chapter 2	Customizing the Creo Elements/Direct Drafting Ribbon — how to customize the ribbon to quickly access frequently-used commands.
Chapter 3	Getting Started with Creo Elements/Direct Drafting — supplies you with hints to help in getting started.
Chapter 4	Preparing the Drawing Environment — how to set up the environment and store it as a file.
Chapter 5	Commonly Used Commands — how to escape from a current activity.
Chapter 6	Changing the Environment and Making Measurements — how to change the environment in which the drawing is produced and how to measure distances on your drawing.
Chapter 7	Using the Creo Elements/Direct Drafting's Copilot Functionality — how to create a design drawing with the minimum of construction geometry.
Chapter 8	Drawing Methods — how to create a drawing with or without the aid of construction geometry.
Chapter 9	Modifying a Drawing — how to modify your drawing.
Chapter 10	Dimensioning a Drawing — how to automatically dimension a drawing.
Chapter 11	Hatching a Drawing — how to hatch drawing sections.
Chapter 12	Adding Text to a Drawing — how to add text to a drawing.
Chapter 13	Adding Symbols to a Drawing — how to add geometric tolerance symbols and other special annotation to your drawing.
Chapter 14	Handling Parts and Assemblies — how to use parts and assemblies when creating drawings.
Chapter 15	Generating Hidden Lines — how to generate hidden lines in assembly drawings.
Chapter 16	Using Layers and Infos — how to use layers and how to add textual information (infos) to elements in your drawing.
Chapter 17	Storing, Loading, and Filing — how to load and save your Creo Elements/Direct Drafting drawing files.
Chapter 18	Revision Checking — how to compare revised drawings.
Chapter 19	Associative Documentation Update — how to update 3D design

	modifications in the 2D layout.
Chapter 20	Editing Imported Pictures — how to handle imported bitmap files in Creo Elements/Direct Drafting.
Chapter 21	Linking and Embedding OLE Objects — how to insert OLE objects into a Creo Elements/Direct Drafting drawing.
Chapter 22	Annotation — how to visualize, annotate and edit MI format 2D drawings generated by Creo Elements/Direct Annotation.
Chapter 23	Printing and Plotting — how to plot your finished drawing and how to dump screens.
Chapter 24	Parametric Design with Creo Elements/Direct Drafting Fluent User Interface — how to use the Parametric Design module in Creo Elements/Direct Drafting
Chapter 25	Toolbar and Classical Menu Support in Creo Elements/Direct Drafting Fluent User Interface — how to load toolbars and classical menus in Creo Elements/Direct Drafting.
Chapter 26	Managing drawings with Windchill — how to activate Windchill PDM actions and the various Windchill PDM commands.

1

The Drafting User Interface

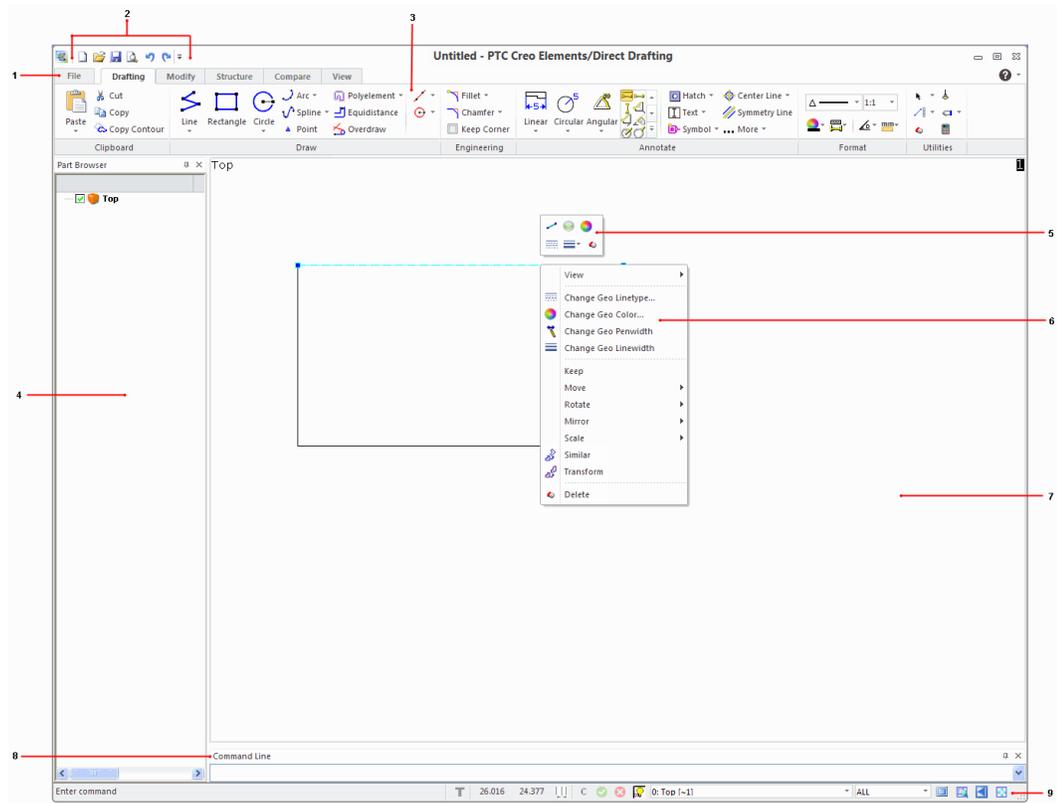
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With Creo Elements/Direct 18.0, a Fluent user interface has been introduced for Drafting. The Fluent user interface facilitates faster access to commands by grouping the commands according to their functionality.

The Creo Elements/Direct Drafting consists of the following elements:

- [File Tab on page 19](#)
- [Quick Access Toolbar on page 21](#)
- [Ribbon on page 21](#)
- [Status Bar on page 23](#)
- [Drawing Area on page 24](#)
- [Part Browser on page 24](#)
- [Command Mini Toolbar on page 24](#)
- [Context Menu on page 24](#)
- [Command Line on page 24](#)

Figure 1: The Drafting User Interface



- 1 File tab
- 2 Quick Access Toolbar
- 3 Ribbon
- 4 Part Browser
- 5 Command Mini Toolbar
- 6 Context menu
- 7 Drawing area
- 8 Command line
- 9 Status bar

You can middle-click to perform the following operations:

- Middle-click to end a command.
- Middle-click and rotate to zoom.
- Middle-click and drag to pan.

Refer to *Creo Elements/Direct Drafting Quick Reference Card* for more information.

File Tab

The **File** button in the upper-left corner of the Creo Elements/Direct Drafting window opens the **File** menu. The **File** menu contains the following commands:

-
- **New Session:** Opens a new drafting window.
 - **Open:** Opens a drawing from the file system.
 - **Open From Database:** Opens a drawing from the database which is connected to the Drafting current session. If no database is activated, this command is not available.
 - **Save** or **Save As:** Saves a drawing to the file system.
 - **Save To Database:** Saves a drawing to the database which is connected to the Drafting current session. If no database is activated, this command is not available.
 - **Database:** Offers commands that are specific to the database which is connected to the current Drafting session. The name of this command changes depending on the database. For example, if you activate Drawing Manager, Database changes to Drawing Manager. If no database is activated, this command is not available.
 - **Print:** Prints or plots a drawing. For more information, see [Printing and Plotting on page 606](#).
 - **Import/Export:** Imports or exports files. For more information, see [Storing, Loading, and Filing on page 528](#).
 - **Modules:** Activates or deactivates one of the following database modules:
 - **Design Data Management**
 - **Drawing Manager**
 - **Windchill Workgroup Manager**
 - **Integration for SAP ERP**
- You can also activate the **Hidden** and **Parametric** modules.
- **Settings:** Changes the draw and system settings. For more information, see [Preparing the Drawing Environment on page 61](#).
 - **Keys:** Customizes the keys. For more information, see [Defining Key Commands on page 57](#).
 - **Options:** Opens the **Quick Access Toolbar** settings for adding or removing commands in the Quick Access toolbar.
 - **Exit:** Exits Creo Element/Direct Drafting.
 - **Search for Command:** Type the name of the command and then select the relevant command from the list to start the command.

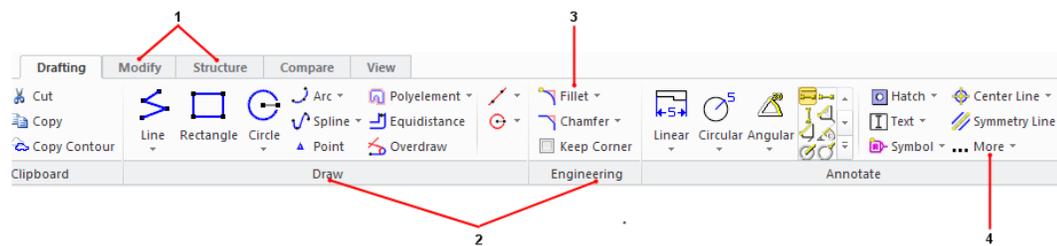
Quick Access Toolbar

The Quick Access toolbar is by default at the top of the Creo Elements/Direct Drafting window. It provides quick access to frequently-used commands. You can customize the Quick Access toolbar by adding commands (buttons and menus) to it.

Ribbon

The ribbon contains commands organized as a set of tabs. On each tab, related commands are grouped. The following figure shows the different elements of a ribbon.

Figure 1A: Ribbon



- 1 Tab
- 2 Group
- 3 Button
- 4 More button

1. Tab: The tabs available on the ribbon are:

- **Drafting**
- **Modify**
- **Structure**
- **Compare**
- **View**
- **Hidden**
- **Parametric**
- **Annotation**
- **ADU**
- **Miscellaneous**

To activate the **Hidden** module, the **Parametric** module in Creo Elements/Direct Drafting:

- a. Click **File ► Module**.
- b. Click one of the following:

-  **Hidden** to activate the **Hidden** module.
-  **Parametric** to activate the **Parametric** module.

You can customize the ribbon to display the following additional tabs:

- **Annotation**
- **ADU**
- **Miscellaneous**

For more information about customizing the Creo Elements/Direct Drafting ribbon, see [Customizing the Creo Elements/Direct Drafting Ribbon](#) on page 32.

2. Group: Commands are placed within groups according to their functionality.

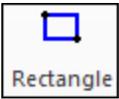
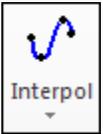
 **Note**

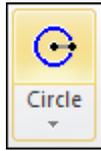
The **Utilities** group is available on all the tabs except the following tabs:

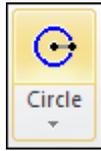
- **View**
 - **Annotation**
 - **ADU**
 - **Miscellaneous**
-

3. Button: Each button on the ribbon represents a command and consists of an icon and a label. Some buttons also contain an arrow along with an icon and label.

The ribbon has the following types of buttons:

-  or  (Large or small button with an icon and a label): Click this button to execute an operation.
-  or  (Large or small button with an icon and an arrow not separated by a line): An arrow indicates the existence of a list. Click this button to open a list of related commands. When no line separates a button, the icon and arrow perform the same action.



-  or  (Large or small button with an icon and an arrow separated by a line): A line, or separation, between the icon and arrow indicates that the icon and arrow perform different actions. Click the icon on this button to execute an operation. Click the arrow to open a list of related commands. The icon of the last-selected command in the current session is displayed on the button.

 **Note**

- The label remains unchanged irrespective of the last selection.
- The tooltip displays the name of the command that the icon represents.
- The procedural topics assume that the icon of the default command in a list is displayed on a button though you may see the icon of your last-selected command.

4. More: The rarely-used commands are listed at the end of each group under the **More** button to optimize the user interface.

Status Bar

The status bar is at the bottom of the Drafting window and has the following buttons:

- : Confirms a command or selection.
- : Ends the current command.
- : Cancels the current command.
- : Highlights the active part in the graphics area.
- : Switches the full screen mode on and off (alternatively, you can press F11). For more information, see [Viewing a Drawing in the Full Screen Mode on page 122](#).
- : Defines a new viewing window.
- : Displays the last view.
- : Fits the content into the viewport.

The status bar also displays the coordinates at which the cursor is placed in the drawing area, the name of the current part, and the catch range. For more information on catch, see [Changing the Drawing Environment and Making Measurements on page 90](#).

Drawing Area

The drawing is displayed in the drawing area. The drawing area is in between the ribbon and the status bar.

Part Browser

The part browser displays the assembly structure with the parts and subparts, the unique ID, part-sharing information, scale, and symbol for the part or the subpart.

Click **Structure** and then, in the **Parts** group, click  **Browser** (alternatively, press F12) to show or hide the part browser.

Command Mini Toolbar

Creo Elements/Direct Drafting opens the Command Mini Toolbar (CMT) when you select any element in the drawing area in the preselection mode. Depending on the current context and the selected object, the elements on the CMT change. The CMT helps you to quickly execute a command from the viewport. The first command on the CMT is the command executed upon Drag and Drop of the element. For more information, see [Command Mini Toolbar on page 52](#).

Context Menu

The context menu opens when you right-click an element in the drawing area in the preselection mode. Depending on the current context and the selected object, the commands on the context menu change. The first command on the context menu is the **View** command. If you right-click the drawing area without selecting any element, the context menu contains only the **View** buttons. For more information, see [Context Menu Commands on page 55](#).

Prompt Line

The **Prompt Line** tells you how to proceed to complete the current operation. In the absence of the **Prompt Line**, Creo Elements/Direct Drafting displays the prompt in the status bar.

You can enable or disable **Prompt Line** by clicking or clearing the **Prompt Line** check box under  **Toolbars** in the **Viewing** group on the **View** tab, respectively.

Command Line

In the **Command Line** (user input line) you can type the required values, when prompted.

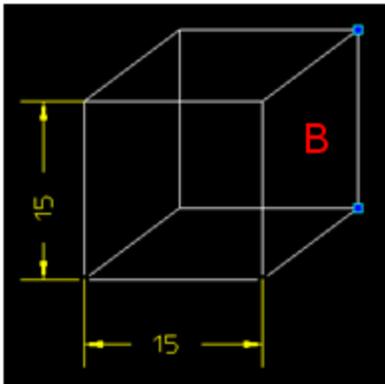
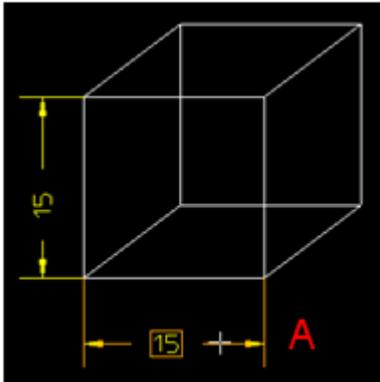
You can enable or disable **Command Line** by clicking or clearing the **Command Line** check box under  **Toolbars** in the **Viewing** group on the **View** tab, respectively.

Fly By Highlighting

When you pass over an element with your cursor, fly by highlighting indicates which elements are available for work. Fly By highlighting is available in three modes:

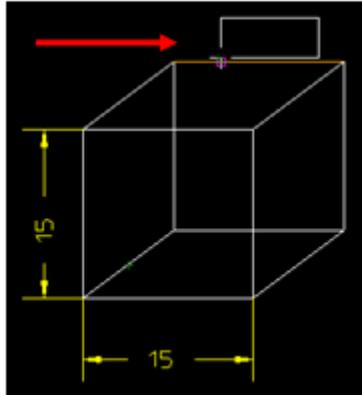
- No commands are active.

When you pass over an element with your cursor, Creo Elements/Direct Drafting highlights the elements that you can drag (A) or that have handles (when you click the element) (B).



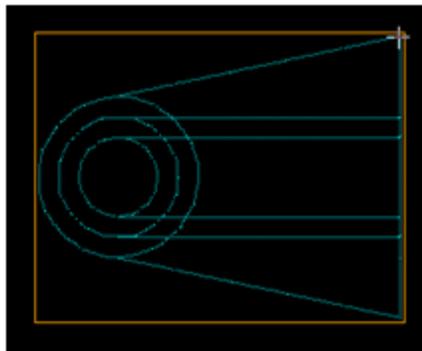
- A command is active.

When you pass over an element with your cursor, the highlighted elements can be used for the command. In the image, the arrow points to the highlighted element where the text box will be placed.



- Selection of parts.

When you hover over an element belonging to a part or the rounding box of the part, Creo Elements/Direct Drafting highlights the part.



Fly By Highlighting Settings

Creo Elements/Direct Drafting allows you to enable or disable fly-by highlighting and change the color of the highlight using the **System** dialog box.

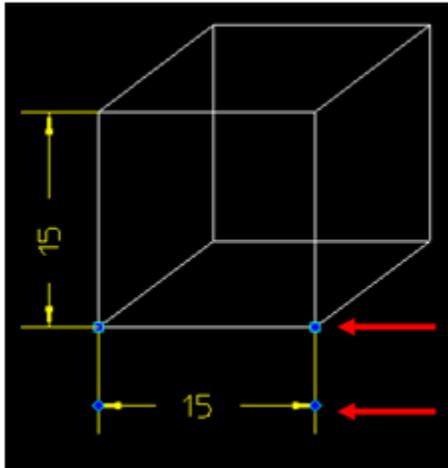
To define the settings for fly-by highlighting:

1. Click **File** ► **Settings** ►  **System**.
2. Click **FlyBy**.
3. Click or clear the **Enable Fly-By Highlighting** check box to enable or disable fly-by highlighting, respectively.
4. In the **Color** box, select the required color for highlighting.
5. Click **OK**.

Handles

Use handles to quickly and easily modify geometry elements, dimensions, text and symbols, symmetry and center lines, references lines, and parts. Handles appear as:

- Squares and diamonds for geometry elements, dimensions, symmetry and center lines, and reference lines.
- Frame around text, symbols, and parts.



Modify with handles

There are two ways to modify an element using the mouse:

- Drag: Available for dimensions, text, symbols, and parts.
 - Click the element and drag it to a new location.
 - Release.
- Click and Drag: Available for all elements.
 - Click the element to display the handles.
 - Click the element again and drag it to a new location.
 - Release.

To show the handles on more than one element at a time, press **SHIFT** and select the elements.

To remove the handles, press **ESC** or type **MH_CLEAR** in the user input line and press **ENTER**.

Dimensions: Move and reattach dimensions, change the direction of the arrows, and modify the stagger line.

- Move the dimension and text: Click anywhere on the dimension and drag.

-
- Change the direction of the arrows: Click the diamond-shaped handle. You cannot change the direction of arrows for broken dimension lines (see [broken on page 308](#)).
 - Reattach the dimension: Click the square-shaped handle where the dimension is attached to the element and drag.
 - Modify the stagger line: Click the square-shaped handle and drag.
 - Move the dimension text only: Press SHIFT and drag the text.

Text or Symbols: Move the text or symbol box.

1. Hover the cursor over the text or symbol. Creo Elements/Direct Drafting highlights a box around the text or symbol (if you have enabled fly-by highlighting).
2. Drag the text or symbol to a new location.

Symmetry or Center lines: Extend a symmetry or center line or rotate a center line.

- Extend the symmetry or center line: Click the handle and drag.
- Rotate the center line: Click the center line (not on the handles) and drag.

Reference lines: Move the reference line or stretch reference line segments.

- Move the reference line: Click the handle on the start or end of the reline and drag.
- Stretch reference line segments: Click inside the reline handles and drag.

Delete with handles

1. Click the item to be deleted.
Creo Elements/Direct Drafting highlights the item or displays handles.
2. Press DELETE.

Handles Settings

Creo Elements/Direct Drafting allows you to enable or disable the handles and change handle settings using the **System** dialog box.

To enable or disable the modification handles:

1. Click **File** ► **Settings** ►  **System**.

The **System** dialog box opens.

2. Click **Handles**.

-
3. Click or clear the **Enable Modification Handles** check box to enable or disable handles, respectively.
 4. Click **OK**.

You can also define the following settings for handles using the **Handles** pane in the **System** dialog box:

- **Handles**
- **Selection**
- **Allowed Types**

Handles

In the **Handles** area, set the following:

Fill Color	Select a color for the square and diamond-shaped handles.
Box Color	Select a color for the frame around text, symbols, and parts.
Size	Type the size of the square handle in pixels.

Selection

In the **Selection** area, click any of the following:

Local	Show the handles of the current part.
Subtree	Show the handles of the current part and its subpart. If you click an element in another part, the entire part is selected.
Global	Show the handles of all parts. Select the entire part when your cursor hovers over the rounding box of the part.

Note

You must click the element to view the handles.

Allowed Types

In the **Allowed Types** area, specify which elements will display handles when they are clicked.

Click the check box of the required element types from the following:

- **Geometry**

-
- **Dimensions**
 - **Construction Geo**
 - **Associative Geo**
 - **Texts**
 - **Parts**

To specify all element types, click the **ALL** check box.

Further Information

For further technical information about FlyBy, refer to the Creo Elements/Direct Drafting help system.

Type `help flyby` in the Creo Elements/Direct Drafting user input line and press ENTER.

Creo Elements/Direct Drafting displays further information about FlyBy.

2

Customizing the Creo Elements/ Direct Drafting Ribbon

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You can customize the Creo Elements/Direct Drafting ribbon to quickly access frequently-used commands.

You can customize the ribbon by:

- [Creating a User-Defined Command on page 34](#)
- [Modifying a User-Defined Command on page 35](#)
- [Adding a Button \(Command\) to the Ribbon on page 36](#)
- [Removing a Tab, Group, or Button from the Ribbon on page 36](#)
- [Adding a Tab on page 37](#)
- [Adding a Group on page 38](#)

-
- [Adding an Element to the Ribbon on page 39](#)
 - [Renaming a Tab, Group, or Button on page 40](#)
 - [Hiding or Showing a Tab or a Group on page 40](#)
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 - [Exporting and Re-using a User-Defined Tab on page 43](#)
 - [Exporting Ribbon Customization Settings on page 45](#)
 - [Importing Ribbon Customization Settings on page 46](#)

You can customize the ribbon using the **Options** dialog box.

To open the **Options** dialog box:

- Click **File** ► **Options**.

Alternatively, right-click the ribbon and select **Customize Quick Access Toolbar**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

To apply and view the customized settings, click **OK** in the **Options** dialog box.

To cancel the modified settings, click **Cancel** in the **Options** dialog box.

Creating a User-Defined Command

You can create your commands and add them to the Creo Elements/Direct Drafting ribbon.

To create a command:

1. Click **File ► Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. Click **New**.

The **Create New Command** dialog box opens.

3. In the **Command ID** area, type a name for the command in the **Name** box.

Note

Creo Elements/Direct Drafting displays the specified name as the label for the button (command) on the ribbon.

4. Optionally, type a tooltip for the command (button) in the **Tool Tip** box.

5. Optionally, type a category for the command in the **Category** box.

The default category is **User-Defined**.

If you type a new category name, a category is created with the specified name, and the command is added to the category.

6. Optionally, in the **Icon** area, click **Browse** to select an image for the button.

Note

- Creo Elements/Direct Drafting supports PNG and BMP images for buttons.
 - Creo Elements/Direct Drafting partially supports JPG and JPEG images for buttons. Creo Elements/Direct Drafting converts these images to BMP images internally.
-

7. In the **Command Action** box, provide the action (function) details.

See the *Creo Elements/Direct Drafting / Creo Elements/Direct 2D Access Programming Reference Guide* for details about functions and commands.

 **Note**

Creo Elements/Direct Drafting executes the specified function when you click the button.

8. Click **Add**.

The command is now available in the **User-Defined** list of the **Choose commands from** box.

 **Note**

You must provide at least a name and an action to create a command.

Modifying a User-Defined Command

You can modify the user-defined commands in Creo Elements/Direct Drafting.

To modify a user-defined command:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Choose commands from** box, select **User-Defined**.

The user-defined commands are listed.

3. Select the command.

4. Click **Modify**.

The **Create New Command** dialog box opens.

 **Note**

Modify is activated only after you select a user-defined command.

5. Edit the settings.

6. Click **Modify** to save the edited settings.

If you want to delete the user-defined command, click **Delete** in the **Create New Command** dialog box.

Adding a Button (Command) to the Ribbon

You can add the default Creo Elements/Direct Drafting buttons as well as user-defined buttons to the Creo Elements/Direct Drafting ribbon.

To add a button to the ribbon:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, select the group to which you want to add a button.
3. In the **Choose commands from** box, select the tab name.

The commands on the selected tab are listed.

Tip

To view the list of user-defined commands, select **User-Defined**.

4. Select a command and click **Add**.

The button for the selected command is added to the group on the ribbon.

Note

- You can add buttons to user-defined groups only.
 - You cannot add buttons directly to a tab.
-

Removing a Tab, Group, or Button from the Ribbon

To remove a tab, group, or a button from the ribbon:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, select the tab, group, or button (command).
3. Click **Remove**.

The selected tab, group, or button is removed from the ribbon.

 **Note**

- You can remove the user-defined tabs and groups only.
 - You cannot remove buttons of the default groups.
-

 **Tip**

You can also use the context menu to remove tabs, groups, or buttons.

Adding a Tab

You can add a tab to the Creo Elements/Direct Drafting ribbon.

To add a tab:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, select a tab below which your tab should be added.

The selected tab is highlighted.

 **Note**

- If you do not select a tab, your tab is added at the end of all tabs.
 - If an item is already selected, your tab is added below the parent tab of the selected item.
-

3. Click **New Tab**.

A tab is added below the selected tab. In addition, a group is automatically added to the tab.

4. Select the tab, click **Rename**, and specify a name for the tab.

 **Tip**

You can also use the context menu to add a tab.

Adding a Group

You can add a group to the Creo Elements/Direct Drafting ribbon.

To add a group:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, select a tab or group.

Ensure that you select a tab or group below which you need to add the group.

The selected tab or group is highlighted.

 **Note**

If you do not select a tab or group, your group is added at the end of all groups of the first tab.

3. Click **New Group**.

A group is added below the selected tab or group.

4. Click **Rename** and specify a name for the group.

 **Note**

You can add groups to user-defined tabs only.

 **Tip**

You can also use the context menu to add a new group.

Adding an Element to the Ribbon

You can add the following elements to the user-defined groups in the Creo Elements/Direct Drafting ribbon:

- A button with an arrow not separated by a line (**Menu**)
- A button with an icon and an arrow separated by a line (**Split Menu**)
For details about different types of buttons, see [Ribbon on page 21](#).
- A separator

Note

You can also add the default buttons similar to the buttons (created by using **Menu** or **Split Menu**) in the user-defined groups.

To add a new element:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

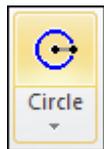
2. In the **Customize the Ribbon** list, select a user-defined group to which you want to add a new element.
3. Click **New Element**.
4. Select one of the following:

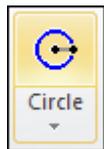
- **Menu** to add a button with an arrow not separated by a line.

For example, .

The button is added at the end of the selected group.

- **Split Menu** to add a button with an icon and an arrow separated by a line.



For example, .

For details about different types of buttons, see [Ribbon on page 21](#).

The button is added at the end of the selected group.

- **Seperator** to add a separator.

A separator is added below the selected group.

 **Note**

You can also add a separator between two buttons.

5. Click **Rename** and specify a name for the button (created by using **Menu** or **Split Menu**).
6. To add a command (button) to the button (created by using **Menu** or **Split Menu**), see [Adding a Button \(Command\) to the Ribbon on page 36](#).

Renaming a Tab, Group, or Button

To rename a tab, group, or button:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, click the tab, group, or button.

The selected item is highlighted.

3. Click **Rename** and specify a name for the tab, group, or button.

The new name is displayed in the **Customize the Ribbon** list and on the ribbon after you click **OK**.

 **Note**

You can rename user-defined tabs and groups only.

 **Tip**

You can also use the context menu for renaming elements on the ribbon.

Hiding or Showing a Tab or a Group

You can select the tabs or groups that you want to hide or view on the ribbon.

To hide or show a tab or a group:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, clear or click the check box of the tab or group to hide or show the tab or group, respectively.

Modifying the Button Style

You can modify the appearance of a button on the ribbon.

To modify the appearance of a button:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, select a button.

The **Modify** button is activated.

Tip

To select multiple buttons in the **Customize the Ribbon** list:

- Press SHIFT+[button] for range selection of buttons. Buttons that are within a group are selected only if the group buttons are expanded.
- Press CTRL+[button] for individual selection of buttons.

3. Click **Modify**.

The **Modify Button** dialog box opens.

4. In the **Icon** area, click **Browse** to replace the current icon with a different icon.

Note

- Creo Elements/Direct Drafting supports PNG and BMP images for icons.
- Creo Elements/Direct Drafting partially supports JPG and JPEG images for icons. Creo Elements/Direct Drafting converts these images to BMP images internally.

To reset the icon, click **Reset**.

Note

You can change or reset the icon of the following:

- A user-defined button created by using **Menu**.
 - A default button with an icon and an arrow, and not separated by a line.
-

5. In the **Button Layout** area, select one of the following:
 - **Small** to display the button as a small button.
 - **Small - No Icon** to display the button as a small button without the icon.
 - **Large** to display the button as a large button.
6. Optionally, click or clear the **Hide** check box to hide or display the label of the button, respectively, on the ribbon.

Changing the Order of a Tab, Group, or Button

You can change the order of a tab, group, or button on the ribbon in Creo Elements/Direct Drafting.

To change the order of a tab, group, or button:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, select the button, group, or tab.

3. Click  to move up or  to move down.

Tip

You can also use the context menu to change the order of a tab, group, or button on the ribbon.

Exporting and Re-using a User-Defined Tab

You can export a user-defined tab and import it to a module. You can export a single user-defined tab at a time. You can also import a single user-defined tab or multiple user-defined tabs to a module at a time.

To export a user-defined tab:

1. Click **File ► Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, select the user-defined tab.
3. Right-click and select **Export Tab**.

The user-defined tab is exported as an XML file to the `Modules` subdirectory of the installation directory. The XML file carries the name of the user-defined tab and contains information about the commands or buttons on the user-defined tab.

Note

The `Modules` subdirectory of the installation directory is the default directory in which Creo Elements/Direct Drafting saves the exported XML file. To change the default directory, use the `ME_SET_MODULE_SEARCH_PATH` command in the user input line. See `ME_SET_MODULE_SEARCH_PATH` in the *Creo Elements/Direct Drafting / Creo Elements/Direct 2D Access Programming Reference Guide* for details.

To import a single user-defined tab or multiple user-defined tabs to a module, use the `REGISTER_MODULE` command in the user input line to import the XML file in the Creo Elements/Direct Drafting user interface. See `REGISTER_MODULE` in the *Creo Elements/Direct Drafting / Creo Elements/Direct 2D Access Programming Reference Guide* for details.

To start or stop the module:

1. Click **File ► Modules**.
2. Select the module.

 **Note**

The modules (consisting of user-defined tabs) are removed when you exit Creo Elements/Direct Drafting.

 **Tip**

You can create macro files to perform activities like licensing-related tasks, if required, when you start or stop the module.

- You can create a macro file in the `Modules` subdirectory (default) or in the path specified using the `ME_SET_MODULE_SEARCH_PATH` command and the file name must be identical to the name of the user-defined module. For example, you can create a macro file `IBS.m` for the user-defined module `IBS` and include the load-specific macros in the file. Creo Elements/Direct Drafting uses this macro file during the start of the module. Ensure that you have included the commands required to activate the buttons on the imported tabs in the macro file. You can also add licensing-related commands to the macro file.
 - When you stop a user-defined module, Creo Elements/Direct Drafting removes the tab of the corresponding module. You can create a macro file in the `Modules` subdirectory (default) or in the path specified using the `ME_SET_MODULE_SEARCH_PATH` command and the file name must be identical to the name of the user-defined module appended by `_unload`. For example, you can create a macro file `IBS_unload.m` for the user-defined module `IBS` and include the unload-specific macros in the file. You can add commands to release any licenses acquired for the module or perform the desired cleanup actions using this macro file.
-

To import a user-defined tab to the Creo Elements/Direct Drafting ribbon, use the `IMPORT_TAB` command in the user input line and import the XML file. See `IMPORT_TAB` and `ME_SET_MODULE_SEARCH_PATH` in the *Creo Elements/Direct Drafting / Creo Elements/Direct 2D Access Programming Reference Guide* for details.

Note

- You can specify the name of an existing tab for an imported tab. Importing a tab with a name similar to an existing tab does not affect the existing tab. Consequently, Creo Elements/Direct Drafting adds another tab with a similar name in the Creo Elements/Direct Drafting ribbon.
 - The import of user-defined tabs is independent to the registration of user-defined modules.
 - The imported tabs are removed when you exit Creo Elements/Direct Drafting. Consequently, they are unavailable when you start Creo Elements/Direct Drafting next time.
-

You can modify (add elements to, move the elements, and so on) the imported user-defined tab or module by starting Creo Elements/Direct Drafting with the `moduleadmin` option:

1. Click **Start** ► **All Programs** ► **PTC** ► **Creo Elements Direct Drafting [version]**.
2. Right-click **Creo Elements Direct Drafting [version]**.
3. Select **Properties**.
4. In the **Target** box, append `-moduleadmin` at the end of the existing string.
5. Click **OK**.

Note

- If you start Creo Elements/Direct Drafting without using `-moduleadmin`, the user-defined tabs are considered as system tabs and you cannot modify them.
 - The imported tabs are available in the Creo Elements/Direct Drafting ribbon even after you use **Reset** in the **Options** dialog box to restore default ribbon settings.
-

Exporting Ribbon Customization Settings

You can export the existing customization settings of the Creo Elements/Direct Drafting ribbon to a XML file. The same customization settings can be used on other computers by importing the XML file in Creo Elements/Direct Drafting.

To export the ribbon customization settings:

-
1. Click **File ► Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. Click **Import/Export**.
3. Click **Export Customization**.

The **Choose exported Creo UI file** dialog box opens.

4. Specify the path and the file name for the XML file.

Importing Ribbon Customization Settings

You can import the XML file generated by **Export Customization** to replace the existing ribbon customization settings.

To import the ribbon customization settings:

1. Click **File ► Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. Click **Import/Export**.
3. Click **Import Customization**.

The **Choose exported Creo UI file** dialog box opens.

4. Select the XML file.

Note

The imported ribbon customization settings override the existing ribbon customization settings. It is recommended to export the existing ribbon customization settings before using **Import Customization**. You can then use the exported file to revert to the existing ribbon customization settings, if required.

Restoring the Default Ribbon Settings

You can reject the changes that you have made for customizing the ribbon. The default ribbon settings are restored after rejecting the ribbon customization changes.

To restore the default ribbon settings:

1. Click **File ► Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. Click **Reset**.

3

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Starting Creo Elements/Direct Drafting

To start Creo Elements/Direct Drafting with the standard Fluent UI:

- Click **Start** ▶ **All Programs** ▶ **PTC** ▶ **Creo Elements Direct Drafting [version]** ▶ **Creo Elements Direct Drafting [version]**.

To start Creo Elements/Direct Drafting with alternative UI:

- Classic UI
 - Click **Start** ▶ **All Programs** ▶ **PTC** ▶ **Creo Elements Direct Drafting [version]** ▶ **Alternative UI** ▶ **Creo Elements Direct Drafting [version] (Classic UI)**.
- Mixed UI
 - Click **Start** ▶ **All Programs** ▶ **PTC** ▶ **Creo Elements Direct Drafting [version]** ▶ **Alternative UI** ▶ **Creo Elements Direct Drafting [version] (Mixed UI)**.
- Windows UI
 - Click **Start** ▶ **All Programs** ▶ **PTC** ▶ **Creo Elements Direct Drafting [version]** ▶ **Alternative UI** ▶ **Creo Elements Direct Drafting [version] (Windows 9x)**.

Basic Working Methods

This section informs you about few hints that may help you to get started with Creo Elements/Direct Drafting.

The best way to learn is to practice, and we recommend you try the examples given in this guide. If you need further explanation of any of the 2D functions or buttons, refer to this guide. It contains a description of how to use each button or function.

An index is included at the back of the guide. It contains not only the command and function names but also a task reference.

Cut, Copy, and Paste

Creo Elements/Direct Drafting provides  **Cut**,  **Copy** and  **Paste** which operate via the Clipboard.

To use them, click **Drafting** and then, in the **Clipboard** group, click the following buttons:

- Click  **Cut** to cut elements.
- Click  **Copy** to copy elements.
- Click  **Paste** to paste elements.

The  **Select** buttons (including  **Polygon**; see [Selecting Drawing Elements on page 77](#)) can be used to facilitate the selection of elements to be cut or copied.

The handling of elements via the Clipboard can be manipulated via the **Clipboard** settings.

To view the **Clipboard** settings:

1. Click **File** ► **Settings** ►  **System**.

The **System** dialog box opens.

2. Click **Clipboard**.

You can change the following settings:

Inverse Black and White	Click the Inverse Black and White check box. White changes to black, while black changes to white.
Global Selection	Click the Global Selection check box. Like the Global selection function (see Selecting Drawing Elements on page 77), this setting selects the entire parts tree rather than just the current part. If you paste to Creo Elements/Direct Drafting, this parts tree is retained. If you copy to another application, the parts tree is smashed.

Paste As Subpart

When you have cut or copied an item, you can paste this item such that it is owned by the current part using the **As Subpart** button. Refer to [Hierarchical Parts Structure on page 431](#) for details on the Creo Elements/Direct Drafting parts structure concept.

To use **As Subpart**:

1. Click **Drafting** and then, in the **Clipboard** group, click the arrow next to  **Paste**.
2. Click **As Subpart**.

Undo and Redo

In Creo Elements/Direct Drafting, you can undo mistakes with  **Undo** or  **Redo**:

- On the Quick Access Toolbar, click  to undo an operation or press CTRL +Z.

-
- On the Quick Access Toolbar, click  to redo an operation (after you have clicked  **Undo**) or press CTRL+Y.

Each time you click  **Undo** or  **Redo**, you will undo or redo all the changes that were created by the last button you clicked.

You can set a limit on the number of steps to keep in the  **Undo** history in the **System** dialog box.

1. Click **File** ► **Settings** ►  **System**.

The **System** dialog box opens.

2. Click **Environment**.

3. In the **Undo** area, click one of the following:

- Click **Unlimited** to allow unlimited number of steps for  **Undo**.
- Click **Max steps** and type the maximum number of steps allowed in the  **Undo** history.

Limitations:

- You can undo or redo complete macros, but you cannot use  **Undo** or  **Redo** inside macros.
-  **Undo** or  **Redo** cannot undo or redo:
 - Commands related to parametric design or hidden lines.
Once you have executed one of these commands, you can no longer undo or redo any of your previous commands.
 - The RESET_SYSTEM command.
 - Commands for default settings (although these commands do not affect the  **Undo** or  **Redo** history).

Undo and Redo with state history

Each time you enter a command and change your drawing, Creo Elements/Direct

Drafting adds a new state to the  **Undo** or  **Redo** history. Creo Elements/Direct Drafting marks the new state with a unique ID. You can use several

commands to work with these unique IDs. For further information, see the *Creo Elements/Direct Drafting / Creo Elements/Direct 2D Access Programming Reference Guide*.

Select, Click, Specify, and Enter

Many operations require similar user interaction. For example, you are often prompted to select an element, enter a value, enter a filename, or specify a point.

Some of the most commonly used terms are listed below:

- **Select** — Use the left mouse button (or a supported digitizer) to select or draw a selection box around items in the drawing.
- **Click** — Briefly press the left mouse button (or a supported digitizer) to select items such as a menu option or an element in the drawing.
- **Enter** — Send information to the computer. You send information to the computer from the user input line. Normally, you enter information in the user input line and then press ENTER. For certain commands and functions you must enter a numerical value, and for others a string enclosed in quotes (' ' or " "). Additionally, values to be entered can be first calculated by typing the appropriate mathematical expression in parentheses:
 $(15 * \tan 30)$
- **Specify** — Press the cursor at the appropriate position or enter the coordinates separated by a comma.

Command Mini Toolbar

The Command Mini Toolbar (CMT) helps you to quickly execute a command from the viewport. The CMT appears when you preselect elements. The following table lists the buttons in the CMT for different element types:

Element Type	CMT Buttons
Geometry	<ul style="list-style-type: none"> •  Move •  Copy •  Change Geometry Color •  Change Line Type •  Element Line Width •  Element Pen Width •  Delete
Dimension	<ul style="list-style-type: none"> •  Modify Move Text & Line •  Properties •  Break •  Slant •  Stagger •  Delete

Element Type	CMT Buttons
Text	<ul style="list-style-type: none"> •  Move •  Copy •  Text Modify •  Text In Place •  Delete
Part	<ul style="list-style-type: none"> •  Move •  Edit Part •  Edit Top •  View Part •  View Top •  New Subpart (&Edit) •  Delete

The CMT includes element-specific commands according to your selection. For example, if you select a dimension, geometry, and text as the element-types, the CMT displays the following buttons:

-  **Move**
-  **Copy**
-  **Modify Move Text & Line**
-  **Change Geometry Color**
-  **Change Line Type**
-  **Change Line Width**
-  **Properties**
-  **Text Modify**
-  **Delete**

 **Note**

If you preselect parts with other element-types, only the parts-specific buttons are displayed on the CMT.

Context Menu Commands

The context menu appears when you right-click elements in the drawing area in the preselection mode. Depending on the current context and the selected object, the commands on the context menu change.

If you do not preselect any elements, the context menu displays the following commands (also present in the **Viewing** group on the **View** tab):

-  **Redraw**
-  **Fit**
-  **New**
- **Last**
-  **Zoom**
-  **Pan**
-  **Center**
- **Store**
- **Recall**

The context menu displays element-specific commands if you preselect any elements. The following table lists the commands on the context menu for different element-types and preselection modes:

Element Type	Context Menu	Preselection Mode
Geometry	<ul style="list-style-type: none">  Change Geo Linetype  Change Geo Color  Change Geo Penwidth  Change Geo Linewidth 	Single or Multiple
Part	<ul style="list-style-type: none">  Edit Part  Initialize New Subpart  Create New Subpart Gather Into Part  View Part Rename Part Smash Part  Share/Unshare Part  Symbol Part 	Single
Dimension	<ul style="list-style-type: none">  Change Dim Modify Dim 	Single
	<ul style="list-style-type: none">  Change Dim 	Multiple
Text	<ul style="list-style-type: none">  Change Text  Edit Text 	Single
	<ul style="list-style-type: none">  Change Text 	Multiple

Additionally, the context menu also contains the following for element transformations (except when dimensions are preselected):

- **Move**
- **Rotate**
- **Mirror**
- **Scale**

-
-  **Similar**
 -  **Transform**

The context menu also contains the **Keep** command, which you can use to retain the original element after the element is transformed. The last command on the context menu is  **Delete**.

 **Note**

- The context menu includes element-specific commands according to your selection. For example, if you preselect a single text and a single part, the context menu displays the part-specific commands, text-specific commands, the element transformation commands, and  **Delete**.
 - For other element types (for example, pictures and OLE objects), the context menu only displays the element transformation commands and  **Delete**.
-

Viewports

A viewport is a rectangular area on the screen whose position and size you can define. Specific parts of your drawing can be displayed in the viewports. This enables you to study the drawing or add information with greater ease and accuracy. You can create up to 16 viewports but usually between 2 and 6 are sufficient.

The viewports are identified by a number in the top right hand corner. You will notice one is in inverse video – this is the current viewport. All window functions operate on the current viewport. For example, if you click **View** and then in the

Window group, click  **New** and make a box in any viewport, the new view is drawn in the current viewport. Any viewport can be made current by clicking its identification number on the screen.

Defining Key Commands

To define a custom key command:

1. Click **File** ► **Keys**.
The **Key Definitions** dialog box opens.
2. Click the required key (for example, 1).

- Specify any or all of the following commands:

Plain	In the Plain box, type the command to be executed when the key is pressed.
Shift	In the Shift box, type the command to be executed when the key is pressed in combination with SHIFT.
Control	In the Control box, type the command to be executed when the key is pressed in combination with CTRL.
Alt	In the Alt box, type the command to be executed when the key is pressed in combination with ALT.

- Click the **Execute** check box if you want the command to execute when the associated key (or key combination) is pressed.

If you clear the **Execute** check box, the command is written to the user input line in Creo Elements/Direct Drafting when you press the key (or key combination). To execute the command, press ENTER.

 **Note**

Some keys (such as F1) are disabled because they serve a predefined function in the Windows environment.

How to Delete Elements

As you learn to work with the system you may make mistakes. To delete an element or piece of text:

- Select the element or text you want to delete.
- In the **Utilities** group, click  **Delete**.

Getting Help

The *Creo Elements/Direct Drafting / Creo Elements/Direct 2D Access Programming Reference Guide* contains technical reference information about all the commands and functions that you can use within Creo Elements/Direct Drafting.

To use the help system:

- On the top-right of Creo Elements/Direct Drafting window, click .
- Enter the command or function on which you require help by either:

-
- Clicking on a command or function on the screen menu.
 - Typing the command or function name in Creo Elements/Direct Drafting user input line.

Creo Elements/Direct Drafting responds by displaying the required help information.

3. Close the Help window to return to the normal Creo Elements/Direct Drafting user interface.

Switching Languages

By default, the Creo Elements/Direct Drafting user interface is displayed in English language. To switch to the language of your choice, type the appropriate command in the user input line and press ENTER:

Language	Command
English	<code>change_locale 'en'</code>
French	<code>change_locale 'fr'</code>
German	<code>change_locale 'de'</code>
Italian	<code>change_locale 'it'</code>
Spanish	<code>change_locale 'es'</code>
Japanese	<code>change_locale 'ja'</code>

Alternatively, specify this command above the `Startup_menus` string in the `startup.m` file located in the Creo Elements/Direct Drafting installation directory.

Note

If you installed Creo Elements/Direct Drafting in Custom mode, any languages that you did not select during installation are not available at Creo Elements/Direct Drafting runtime.

Switching the Beeper On or Off

Optionally, you can enable or disable the "beeper" in Creo Elements/Direct Drafting. Simply type `BEEPER ON` or `BEEPER OFF` in the user input line and press ENTER.

Exiting Creo Elements/Direct Drafting

To exit Creo Elements/Direct Drafting:

- Click **File** and then click **Exit**.

Creo Elements/Direct Drafting prompts you to save changes, if any, before exiting.

Further Reading

You can now use Creo Elements/Direct Drafting to produce high quality 2D drawings.

When you are ready to learn more about the system and its command structure refer to *Writing Macros with Creo Elements/Direct Drafting*, which contains information on macro writing and command syntax.

4

Preparing the Drawing Environment

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Editing the Creo Elements/Direct Drafting Environment

The Creo Elements/Direct Drafting environment specifies various settings, which determine, for example:

- Line color.
- Dimension appearance.
- Text appearance.

Click **File ► Settings** to access various dialog boxes for defining settings related to your design and drafting environment. For example, you can use  **Construction** to select the color and linetype, which is used to display construction geometry in your drawing.

Many of the dimension settings can be set using the commands contained in the Creo Elements/Direct Drafting environment.

Setting the Environment

You can define the following settings for the drawing environment using **File ► Settings**.

 Construction	Edit the construction geometry settings, for example, line color.
 Geometry	Edit the geometry settings, for example, line color.
 Line Patterns	Add or edit user-defined line patterns.
 Dimension	Edit the dimension settings, for example, dimension offset.
 Hatch	Edit the hatch settings, for example, hatch distance.
 Text	Edit the text settings, for example, text size.
 Projected Reference Points	Edit the settings (such as color or linetype) for projected reference points and their associated lines and circle.
 Reflines	Edit the reference line settings, for example, line color.
 Symmetry Line	Edit the symmetry line settings, for example, line color.
 Center Line	Edit the center line settings, for example, line color.



Parts

Edit the part representation settings, for example, spotlight color.



System

Edit general settings for the following:

- Parametric Design
- Copilot functions
- Clipboard
- Handles
- Flyby highlighting
- Auto save mechanism
- Menu cascade timeout
- Redraw

Viewport

Edit the viewport settings, for example, viewport background color.

Grid

Edit the grid settings.

Show

Edit the general show settings and the show settings for infos, layers, and urls.

Color

Define the color scheme for Creo Elements/Direct Drafting from the following:

- **Creo Colors**
- **CoCreate Colors**



Hidden

Edit the settings for hidden lines, for example, line type.



**Parametric
Import DXF/DWG**

Edit the settings for parametric display and icons.

Analyze the data, and after reading the drawing file, set its properties.

Saving the Environment Settings

Creo Elements/Direct Drafting saves your environment settings when you exit.

The settings are saved to <application data path>\PTC\Creo Elements Direct Drafting [version]\[version]\.

Setting the Default Color Scheme

You can set a default color scheme for Creo Elements/Direct Drafting.

When you set the default color scheme, the default colors of the following are defined:

- Geometry
- Text
- Hatches
- Dimensions
- Highlight color
- Constraint color
- Centerline color
- Symmetry line color
- Reference line color
- Viewport colors

To set the default color scheme:

1. Click **File** ► **Settings** ► **Color**.
2. Click a color scheme:
 - **Creo Colors**
 - **CoCreate Colors**

The **Creo Colors** scheme is the default color scheme for Creo Elements/Direct Drafting. The default colors in the **Creo Colors** scheme are:

- Element Types

Element Type	Default Color	RGB Value
Geometry	Black	(0 0 0)
Dimensions	Blue	(0 0 1)
Text	Black	(0 0 0)
Hatches	Black	(0 0 0)

- Viewport Color

Viewport	Default Color	RGB Value
Background	White (Almost)	(251 251 252)
Cursor	Black	(0 0 0)
Text	Black	(0 0 0)
Border	Custom	(238 240 241)

The **Viewport Settings** dialog box (see [Viewport Settings on page 129](#)) displays the default colors for the selected color scheme.

Setting Angular and Linear Units

You can select angular and linear units using  **Angle Units** and  **Length Units**, respectively.

To set the angular unit:

1. Click **Drafting** and then, in the **Format** group, click  **Angle Units**.
2. Click the required angular unit from the following:

-  **Degree**
-  **Gradian**
-  **Radian**
-  **User**

To set the linear unit:

1. Click **Drafting** and then, in the **Format** group, click  **Length Units**.
2. Click the required linear unit from the following:

-  **cm**
-  **meters**
-  **km**
-  **mm**
-  **µm**
-  **inches**
-  **feet**
-  **yards**
-  **miles**
-  **mils/thou**
-  **pinches**
-  **user**

Normally, you set the current units before initiating drawing. However, if you reset the current units after you have started to draw, Creo Elements/Direct Drafting automatically converts all existing elements without changing the lengths.

For example, if you set the current units to meters and draw a line one unit long, and when you change the current units to millimeters, the line measures 1000 units.

Defining Your Own Units

The  **User** and  **user** buttons in the **Format** group on the **Drafting** tab allow you to define your own units in Creo Elements/Direct Drafting. For example, an input of 5 mm allows you to work in 5 mm units, and an input of 5 degrees allows you to work in 5 degree units.

Loading a Drawing Frame

A number of standard drawing frames are supplied with Creo Elements/Direct Drafting. To load one of these frames:

1. Click **File** ►  **Open** to display the Creo Elements/Direct Drafting File Browser.

The **Open** dialog box opens.

2. Click **Navigator**.

The Creo Elements/Direct Drafting Navigator opens.

3. In the left pane of the **Navigator**, navigate to the Creo Elements/Direct Drafting installation directory.

The thumbnails of drawing frames are displayed for preview.

4. Double-click the thumbnail of the required drawing frame.

The frame is a symbol part which is always loaded below the Top part (refer to [Handling Parts and Assemblies on page 429](#)). Each frame consists of two concentric rectangles: the outer one the size of the respective paper size, the inner one between 10 to 50 mm smaller per side.

The frames are drawn in magenta with the dotted linetype and therefore usually are not plotted.

Drawing frame part names have the following format:

`format_A.mi`, `format_A0.mi`, `format_B.mi` and so on.

The ten frame options are:

-
- A
 - A0
 - A1
 - A2
 - A3
 - A4
 - B
 - C
 - D
 - E

Setting the Drawing Scale

The drawing scale affects only the physical size of construction and drawing lines during plotting. Dimensions, text, and symbols are not affected by this setting. The current drawing scale is displayed in the **Part Browser** (see [Part Browser on page 459](#)). The default drawing scale is 1. The maximum drawing scale is 99999.

For example, when the drawing scale is set to 2, a line drawn 100 mm long is plotted 200 mm long.

To set the drawing scale:

1. Click **File** ► **Settings** ►  **System**.

The **Environment** pane of the **System** dialog box opens by default.

2. In the **Drawing** area, type the drawing scale in the **Scale** box.

You can check the result by viewing the **Part Browser**.

Using the Import Configurator

The Import Configurator controls the translation process and mapping of configurable entities in the DWG/DXF Translator. You can change the default settings using the Import Configurator.

To use the Import Configurator:

1. Click **File** ► **Settings** ► **Import DXF/DWG**.

The **Import Configurator** dialog box opens. This dialog box has two tabs; **Advanced** and **Configure**.

2. Click the **Advanced** tab in the **Import Configurator** dialog box.

3. Under **Model & Layouts Selection**, click the **Read all layout data** check box to enable DXF/DWG >MI translation.
 - a. Click the **Model Space** or **Translate paper space** check boxes for Model space or Paper space translation.
 - b. Click layouts under the **Model & Layouts** section, to create files with more than one Paper space.
 - c. Click the **Side by side layout view** check box for better visualization. It controls whether to show overlapping or side-by-side layouts.
4. Under **Units**, the **DXF/DWG** box shows the units of the selected DXF/DWG file. The following table lists the scenarios for the **Custom Units** check box.

Custom Units	Multiplier	Result
Cleared—The Multiplier box is not visible.	Not applicable.	The units of the imported DXF/DWG file are translated to the MI file as follows: <ul style="list-style-type: none"> • Imperial units are translated using UnitsScale as 25.4 and units as inches. • Metric units are translated using UnitsScale as 1 and units as mm.
Selected—The Multiplier box is visible.	No—The Scale Factor box is not visible.	The DXF/DWG file is imported with the same geometrical units. For example, a DXF/DWG file that contains a line measuring 5m is imported as 5000 mm.

Custom Units	Multiplier	Result
	Yes—The Scale Factor box is visible.	<p>The DXF/DWG file is imported with the geometrical units multiplied by a user-defined scale factor.</p> <p>For example:</p> <ul style="list-style-type: none"> • If a DXF/DWG file contains a line measuring 5mm, and the scale factor is 10, the line is imported as 50mm. • If a DXF/DWG file contains a line measuring 5ft, and the scale factor is 10, the line is imported as 50ft. <p>The general rule for importing a file is: source dimension x scale factor = measured dimension</p> <p>Where, source dimension is the dimension of the geometry in the source DXF/DWG file, and measured dimension is the final dimension after importing the file.</p>

5. Under **Mapping**, map the elements.
 - a. Click the **Font** tab in the **Mapping** section to view or edit the font settings such as source font name, destination font name, width ratio, line spacing, and vertical line spacing.
 - b. Click the **Arrow Head** tab to view and edit the arrow settings such as names of the arrows, arrow scale, and arrow fill.
 - c. Click the **Line Type** tab to define the mapping of AutoCAD linetypes to MI linetypes. The first parameter is the AutoCAD linetype name and the second parameter is the corresponding Creo Elements/Direct Drafting linetype name.
6. Click the **Configure** tab in the **Import Configurator** dialog box.
7. Under **General Settings**, do the following:
 - a. In the **Start Section**, DXF to MI Translator is displayed by default.
 - b. Click the **Kanji translate** check box to translate kanji characters.
 - c. Click the **Keep attributes with part** check box to create additional subparts for each block containing attributes. The attributes are kept with the part to which they belong.
 - d. Click the **Translate DXF invisible entities** check box to translate invisible entities to DXF visible entities.

-
- e. Click the **Record by layer info** check box to add special informational text to every entity for color, line types, each layer, layer's color, and status (visible, frozen and locked).
 - f. Click the **Translate invisible layers** check box to translate AutoCAD layers that are "Frozen" and "Off" to an MI file.
8. Under **Hatch**, do the following:
- a. Type a value in the **Minimum hatch spacing** box. Typical values should be greater than 0.001mm or 0.0001 inches. The default value is 0.0.
 - b. Type a value in the **Contour snap gap** box, which specifies the maximum distance between two entities in a hatch contour. The default value is 1E-12.
 - c. Click the **Hatch close contours** check box to close the hatch contour. Clear the check box to close the hatch contour depending on the **Contour snap gap** value.
 - d. Click the **Hatch to contours** check box to convert DXF/DWG hatches to contours. Clear the check box to translate DXF/DWG hatches to MI hatches.
 - e. Click the **Create manual hatch** check box to create manual hatches. Clear the check box for automatic hatches.
9. Under **Dimensions**, do the following:
- a. Click the **Recompute dim block** check box to re-compute the dimension data structure and update it to latest dimension data structure for correct translation.
 - b. Click the **Explode DXF dimensions** check box to translate dimensions to geometric elements, Clear the check box to translate dimensions to dimension elements.
10. Under **Lines and Curves**, do the following:
- a. In the **Translate DXF pen line size** box, select a value to define the DXF/DWG linesize conversions.
 - b. In the **Polymarker type** box, select a value to define the default polymarker for DXF/DWG/SVG points. The default value is 2.
 - c. Click the **Spline control point translation** check box to convert a spline using its control points and knots. Clear the check box to convert a spline using its fit points.
11. Click **Reset** to revert to the default settings.
12. Click **OK** to save and apply your changes.

-
13. Click **Cancel** to close the import configurator dialog box without applying your changes.

Further Information

For further technical information about the Creo Elements/Direct Drafting environment, refer to the Creo Elements/Direct Drafting help system.

Type `help save_environment` in the Creo Elements/Direct Drafting user input line and press ENTER.

Creo Elements/Direct Drafting displays further information about the environment.

5

Commonly Used Commands

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Most of these commands have a similar purpose — they allow you to escape from whatever activity is currently being performed. The knowledge that you can easily delete or cancel any mistakes should help you gain confidence in using Creo Elements/Direct Drafting.

Deleting Elements

To delete elements, press DELETE or click  **Delete**.

Using Delete Key

1. Click an element in the drawing area.

The handles appear.

2. Press DELETE.

To delete multiple elements:

1. Hold SHIFT and click all the elements in the drawing area.

The handles appear.

2. Press DELETE.

Using Delete

Click  **Delete** to delete drawing elements or pieces of text.

For example, to delete everything in the active part:

1. In the **Utilities** group, click  **Delete**.
2. In the **Utilities** group, click  **Select** ►  **All**.
3. Click  **Confirm**.

To delete an element or text:

1. Click  **Delete**.
2. Select the element or text to be deleted.

You can also enclose the elements to be deleted in a box

Alternatively, you can also use preselection to delete elements.

To delete an element or text:

1. Click the text or element.

The Command Mini Toolbar appears.

2. Click  **Delete** on the Command Mini Toolbar.

Alternatively, press DELETE.

The selected text or element is deleted.

Deleting Hatching

To delete a hatch:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Delete**.
3. Select the hatching to be deleted.

See [Hatching a Drawing on page 366](#) for more information.

Highlighting Elements Before Deletion

The  **Select** buttons enable you to select the elements that you want to delete (or modify).

1. Click  **Delete**.
2. In the **Utilities** group, click  **Select** and then select the required button.

Depending upon the selected button, the elements selected for delete are highlighted. For example, click  **Select** and then click  **All**. All the elements in the drawing get highlighted.

Alternatively, press F2 to open the **Select** window. Then click **Select** and **All**. All the elements in the drawing get highlighted

3. When you are satisfied that the elements to be deleted have been correctly selected, click  **Confirm**.

Alternatively, enclose the elements to be deleted in a box. The elements are preselected and the Command Mini Toolbar appears. Click  **Delete** on the Command Mini Toolbar to delete the elements.

Global Selection

Each time you select components, you can decide whether you want to select from the current part or from the whole parts tree. Global selection allows you to select components from the whole parts tree:

- In the **Select** window, click **Select**, **Global**, and **All**.

If you do not click **Global**, Creo Elements/Direct Drafting defaults to the current part.

Example

1. In the **Utilities** group, click  **Delete**.
2. In the **Utilities** group, click  **Select** >  **Advanced**.

The **Select** window opens.

Alternatively, press F2 to open the **Select** window.

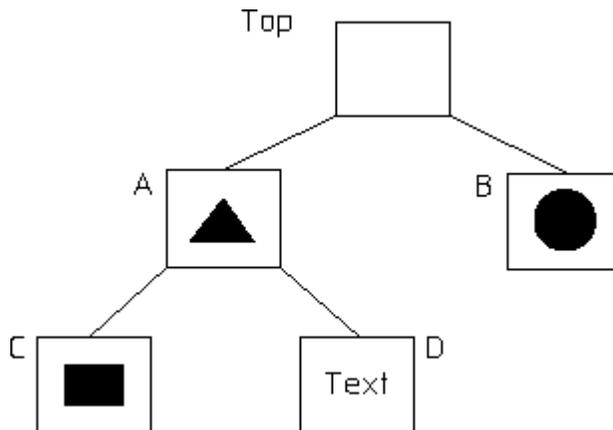
3. Click **Select, Global, All, Confirm** and **OK**.

This command string deletes all components of the parts tree even if the active part is not the top part.

It should, however, be noted that even though all components (geometry, construction geometry, and text) contained within the parts tree will be deleted, it is not possible to delete the complete parts tree. The current part is never deleted and a path linkage to the top part is always maintained.

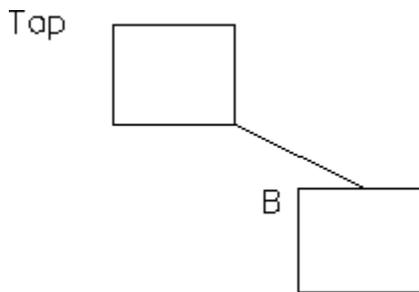
Several examples using the parts structure in [Figure 2 on page 75](#) show the power of **Select**.

Figure 2. Example of a Parts Structure



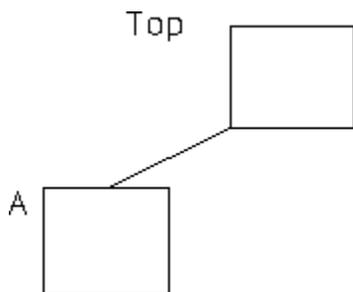
If you delete globally with 'B' as the active part, the Top part and 'B' will remain. See [Figure 3 on page 76](#).

Figure 3. Top Part and 'B'



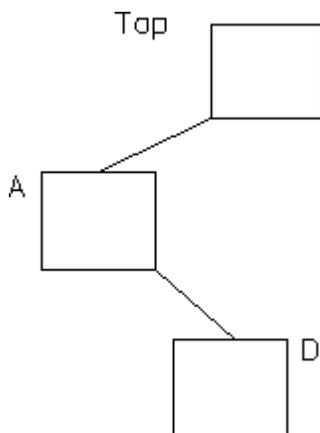
If you delete globally with 'A' as the active part, the Top part and 'A' will remain. See [Figure 4 on page 76](#).

Figure 4. Top Part and 'A'



If you delete globally with 'D' as the active part, the Top part, 'D', and its parent, 'A', will remain. See [Figure 5 on page 76](#).

Figure 5. Top Part + D + A



Selecting Drawing Elements

Use  **Select** to let Creo Elements/Direct Drafting know you are about to make a selection

Click  **Select** (for example, to move or to delete elements). Then click any option or combination of options from the selection block.

To make a selection (for example, to move or delete elements):

- In the **Utilities** group, click  **Select** and then, select the required button:

Select Button	Purpose
 All	Selects all elements in the active part.
 Box	Selects all elements you enclose in a box.
 Polygon	Selects all elements you enclose in a polygon.
 Global	The selection applies to the entire parts tree.
 Global Perm	The selection, and all subsequent selections in this command, applies to the entire parts tree.
 Subtree	Selects a subtree of a part or an assembly.
 PreSelected	Selects the current preselection. See Preselecting Elements on page 87 .
 Advanced	Displays a separate Select window, which provides functions and options for complex selections.

Advanced Selection Functions

To use **Advanced** selection functions:

- In the **Utilities** group, click  **Select** >  **Advanced**.

The **Select** window opens.

Alternatively, press F2 to open the **Select** window.

The functions and options included in the **Select** window enable you to define complex selections. These include the  **All**,  **Box**,  **Polygon**,  **Global**,  **Global Perm**,  **Subtree**, and  **PreSelected** selection buttons described above.

Function	Description
Select	Initiates a selection.
Confirm	Confirms a selection.
Add	Adds the next elements you select to the current result.
Subtract	Subtracts the next elements you select from the current result.
And	Results in a logical AND (set intersection) of the current result and the elements selected next.
Or	Results in a logical OR (set union) of the current result and the elements selected next.
Exor	Results in a logical EXOR (exclusive OR) of the current result and the elements selected next.
Not	Unselects the current result. All other elements become the current result.
Vertex	Selects vertices.
Color	Selects colors.
Line Type	Selects linetypes.
Line Size	Selects linesizes.
Pen Size	Selects pensizes.
Font	Selects fonts.
Groups	Selects groups.
Contours	Selects contours (see Selecting Contours on page 84).
Geo	Selects geometry.
Point	Selects points (singularities) but not vertices.
Line	Selects geometry lines.
Circle	Selects geometry circles.
Arc	Selects arcs.
Fillet	Selects fillets.
Spline	Selects splines.
Poly El	Selects polyelements.
PRPs	Selects projected reference points.
Text	Selects text.
Hatch	Selects hatches.
Dim	Selects dimensions.
Edit Dim	Selects edited dimensions.
Sym Line	Selects symmetry lines.
Center Line	Selects center lines.
Ref Line	Selects reference lines.
Leader Line	Selects leader lines.
CGeo	Selects construction geometry.

Function	Description
CLine	Selects construction lines.
CCircle	Selects construction circles.
Part	Selects parts.
Namespace	Select namespaces.
Info	Selects infos.
URL	Selects URLs.
Layer	Selects layers.
Back	Removes the last change to the selection.
Load	Enables you to load a (previously stored) selection file.
Store	Enables you to store the selection definition to a selection file.
Execute	Immediately applies the current selection.
OK	Immediately applies the current selection and closes the Select dialog box. This button has the combined effect of the Execute and Close .
Close	Closes the Select window.

Note

All the **Select** options can be chosen directly without clicking **Select**. However, if you do not click **Select** first:

- The elements you choose will not be highlighted on the screen for you to view before the operation of your choice. For example, deleting will be performed immediately.
- You will not be able to use the options **Add**, **Subtract**, **Not**, and so forth.

Example: Deleting All Green Elements

If you want to select elements drawn in a particular color or linetype you must first click **Select** and then specify the respective color or linetype.

1. In the **Utilities** group, click  **Delete**.
2. In the **Utilities** group, click  **Select** >  **Advanced**.

The **Select** window opens.

Alternatively, press F2 to open the **Select** window.

3. Click **Select** and then click **Color**.

The **Color** dialog box opens.

4. In the **Color** box, select **Green** and click **OK**.
5. Click **Confirm** and then click **OK**.

This deletes all elements drawn in green in the current part.

Further examples are provided in the following sections.

Adding Elements to Your Selection

This option allows you to add elements, either selected individually or enclosed in a box, to your current selection. For example:

1. Click  **Delete**.
2. Press F2 to open the **Select** window.
3. Click **Select**.
4. Select several elements to be deleted.
5. Click **Add**.
6. Click **Box**.
7. Enclose further elements to be deleted in a box.
8. Click **Confirm**.
9. Click **OK**.

Subtracting Elements from Your Selection

To subtract elements from your current selection, click **Subtract** in the **Select** window.

Removing Elements from Your Selection

Click **Not** in the **Select** window to make the previous selection result unselected, and all previously unselected elements to be treated as selected.

Note

Not always works globally on all parts.

Enclosing Elements in a Box

To select a group of drawing elements by enclosing them in a box:

1. Click **Box** in the **Select** window.
2. Enclose the required elements in a box.

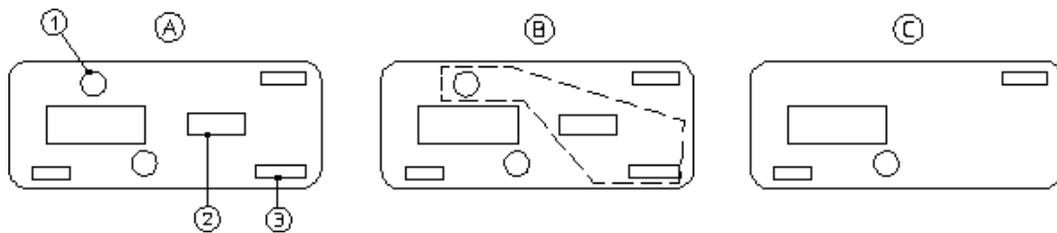
The elements are then shown in a different linetype and Creo Elements/Direct Drafting prompts you to **Confirm** that you want to delete them. Before you click **Confirm**, you may want to alter your selection by, for example, subtracting or adding some elements.

3. Click **Confirm**.
4. Click **OK**.

Enclosing Elements in a Polygon

If you need greater flexibility when selecting elements, use **Poly** to enclose elements within a polyline. This is particularly useful when elements are not symmetrically grouped horizontally and vertically.

Figure 6. Selecting Elements with a Polygon



Suppose you want to delete the elements 1, 2, and 3:

1. Click  **Delete**.
2. In the **Utilities** group, click  **Select** ►  **Polygon**.
Alternatively, press F2 to open the **Select** window and then click **Poly**.
3. Click a succession of points to define the enclosing polygon around elements 1, 2, and 3.
4. Close the polygon by clicking the start point again or type **CLOSE** in the user input line and press **ENTER**.

Selecting All Elements

To select everything contained in the current part (and subparts):

1. Click  **Delete**.
2. Press F2 to open the **Select** window.
3. Click **All**.

-
4. Click **OK**.

Creo Elements/Direct Drafting prompts you to click **Confirm** if you really want to delete all elements selected. If not, click  **Cancel** and select again.

5. Click  **Confirm**

6. Click  **OK**.

Selecting a Vertex

Click **Vertex** in the **Select** window if you want to delete all elements that share a specified point.

Selecting Elements by Type

The **Select** window facilitates the selection of specific element types in your Creo Elements/Direct Drafting drawing.

Selecting Elements Using Information Strings

Click **Info** in the **Select** window to select all elements that have a particular information string associated with them. You will be prompted to type in the information concerned.

Selecting Layers

Click **Layer** in the **Select** window to select all elements assigned to a specified layer. When you choose this option, the system prompts you for the layer information.

Selecting Text

Click **Text** in the **Select** window to select the required text. Creo Elements/Direct Drafting prompts you to identify the text to be selected.

Selecting Reference Lines

Click **Ref Line** in the **Select** window to select reference lines.

Selecting Construction Geometry

Click **CGeo** in the **Select** window to select and highlight construction geometry. Alternatively, click **CCircle** to select construction circles only or click **CLine** to select construction lines only.

Selecting Drawing Geometry

Click **Geo** in the **Select** window to select and highlight drawing geometry. Alternatively, you can:

- Click **Line** to select lines.
- Click **Circle** to select circles.
- Click **Arc** to select arcs.
- Click **Spline** to select splines.
- Click **Fillet** to select fillets.
- Click **Point** to select points.
- Click **Poly EI** to select polyline elements (see [Drawing Polylines on page 172](#)).

To make a selection on pensize or linesize:

1. Click **Line Size** or **Pen Size** in the **Select** window.
2. Type a value (for example, 2).
3. Click **Confirm**.
4. Click **OK**.

Selecting Parts

Click **Part** in the **Select** window to select elements in a particular part. Also, you can click **Subtree** to select a subtree of a part or assembly.

Example

To delete a selected part from anywhere in the parts tree:

1. Click  **Delete**.
2. Press F2 to open the **Select** window.
3. Click **Select**, **Global**, **Part**, and **OK**.
4. Type the part name in the user input line and press ENTER.
5. Click  **Confirm**.
6. Click  **OK**.

Note

When deleting a part, you automatically delete all associated subparts.

Selecting Dimensions

Click **Dim** in the **Select** window to select and highlight dimensions. Alternatively, click **Edit Dim** in the **Select** window to select edited dimensions only.

Selecting Contours

Click **Contours** in the **Select** window to select contours (closed loops of connected elements).

Note

To account for the fact that contours may be nested (closed loops of elements within other closed loops of elements), Creo Elements/Direct Drafting allows you to specify a contour level. See [Specifying the Contour Level on page 85](#) for details.

The general procedure is as follows:

1. Select a given subset (such as an area enclosed in a selection polygon) of your drawing.
2. Apply further selection criteria to refine your selection.
3. Click **Contours** to scan this refined selection for contours.

For example:

- To move all contours in your drawing by a given distance:

1. Click **Modify** and then, in the **General** group, click the arrow next to  **Move**.

2. Click  **2 Pnts** and define the distance and direction of the move.

3. In the **Utilities** group, click  **Select** >  **Advanced**.

Alternatively, press F2 to open the **Select** window.

4. Click **Select, All, Contours, OK**.

5. Click  **Confirm** to move the selected contours.

- To delete all elements other than contours:

1. Click  **Delete**.

2. In the **Utilities** group, click  **Select** >  **Advanced**.

Alternatively, press F2 to open the **Select** window.

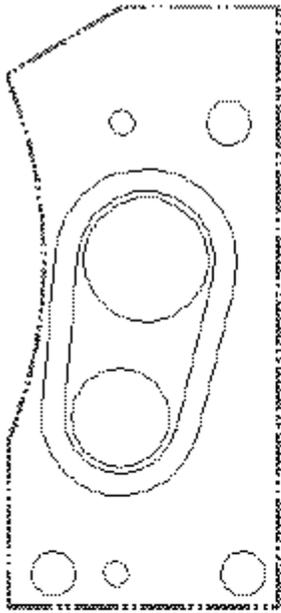
3. Click **Select, All, Contours, Not, OK**.
 4. Click  **Confirm** to delete all non-contour elements.
- To change the color of all contours to green:
 1. Click **Drafting** and then, in the **Format** group, click  **Colors**.
 2. Select the **Green** color.
 3. In the **Utilities** group, click  **Select** ►  **Advanced**.
Alternatively, press F2 to open the **Select** window.
 4. Click **Select, All, Contours, OK**.
 5. Click  **Confirm** to change the color of all contours to green.

Specifying the Contour Level

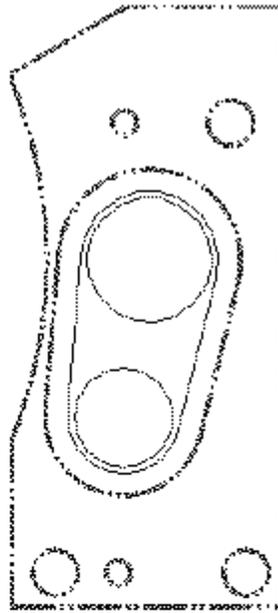
To facilitate the selection of nested contours, Creo Elements/Direct Drafting enables you to specify a selection level by typing `contour_level` followed by an integer.

<code>contour_level 0</code>	This is the default. Only the "outer" contours in the selection polygon are selected. See the part labeled 0 in Figure 7 on page 86 .
<code>contour_level 1</code>	In addition to the outer contours, the first contours encountered within the outer contours are selected. See the part labeled 1 in Figure 7 on page 86 .
<code>contour_level 2</code>	In addition to the contours in contour level 1 (including all level 0 contours), the first contours encountered within level 1 contours are selected. See the part labeled 2 in Figure 7 on page 86 .
<code>contour_level 3</code>	In addition to the contours in contour level 2 (including all level 1 and level 0 contours), the first contours encountered within level 2 contours are selected. See the part labeled 3 in Figure 7 on page 86 .

Figure 7. Contour Levels



0



1



2



3

Copying Contours to the Clipboard

The **Copy Contour** button in the **Clipboard** group on the **Drafting** tab enables you to select contours (closed loops of connected elements) to the Clipboard. **Copy Contour** does not modify the drawing. From the Clipboard, the copied contours can be pasted to another place in the same Creo Elements/Direct Drafting drawing, to another Creo Elements/Direct Drafting drawing, or to Creo Elements/Direct Modeling.

To copy a set of contours:

1. Specify the contour level (see [Specifying the Contour Level on page 85](#)) for the selection you are about to make.
2. Click **Drafting** and then, in the **Clipboard** group, click **Copy Contour**.
3. Click points in your drawing to define a selection polygon around the area from which you want to select contours.

Note

Contours fully enclosed in this polygon are highlighted.

4. Type `contours` in the user input line and press ENTER.
5. Click  **OK**.

Selecting URLs

Click **URL** in the **Select** window to select and highlight URLs.

Preselecting Elements

Automatic Preselection

With automatic preselection enabled, you can add elements to the preselect list and use them for **Modify** operations without using the  **Select** buttons.

To enable automatic preselection:

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. Click **Preselect**.
3. Click the **Automatically use preselection** check box.

Note

Automatic preselection is enabled by default in Creo Elements/Direct Drafting Fluent UI.

The active preselection ends after you start a new command or middle-click to end the current command. The active preselection also ends when you click  **Cancel** or press ESC.

Confirming Commands

Click  **Confirm** to confirm that the deletion or modification you have defined should be implemented.

Undoing Commands

Click  **Undo** when you have just deleted or modified something but realize you have made a mistake. This will return you to the previous display.

Ending a Command

Click  **OK** to terminate the active command or function, whether or not you have completed it. You can also terminate any command or function by selecting another command.

In the case of functions, clicking  **OK** will terminate the currently active function, but the previously selected command will remain active.

Canceling Commands

It may happen that you select a series of commands and functions but do not complete any of them. Although selecting a new command will terminate whatever command or function is active and allow you to continue working, it is advisable to click  **Cancel** and then start again. Otherwise the response time may be slightly slower than normal.

Further Information

For further technical information about the commonly-used commands in Creo Elements/Direct Drafting, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- `help delete`
- `help wui_select`
- `help select`
- `help undo_redo`

Creo Elements/Direct Drafting displays further information about the commonly-used commands.

6

Changing the Drawing Environment and Making Measurements

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Use the functions shown in this chapter as an aid to make designing and drafting easier. These functions are arranged in groups of functions and are available for use at all times.

Functions are essentially different from commands in that a function can interrupt a command (or another function) without terminating it.

For example, while using the  **Fillet** button to draw a fillet radius, you may want to zoom in on the corner or change the current linetype. Both **Linetype** and **View** are interrupt function groups.

Once a function has been executed, control is automatically returned to the previously unfinished command in which you were working.

Setting Linetype, Line Color, and Line Width or Line Size

To set the linetype, line color and line width (pen size) or line size for geometry, construction geometry, or hatching:

1. Click **File** ► **Settings**.
2. Click the settings for which you want to set the linetype, line color and line width or line size:

-  **Construction**
-  **Geometry**
-  **Hatch**
-  **Reflines**
-  **Symmetry Line**
-  **Center Line**

3. In the **Type** box or **Linetype** box, select the linetype.

[Figure 8 on page 92](#) shows the built-in linetypes available in Creo Elements/Direct Drafting and their corresponding symbolic representation. If you want to define your own linetypes, follow the instructions in [Creating User-Defined Linetypes on page 93](#).

Figure 8. Linetypes and Corresponding Symbolic Representation



Item	Description
1	Solid
2	Dashed
3	Long Dashed
4	Dot Center
5	Dash Center
6	Phantom

Item	Description
7	Center Dash Dash
8	Dotted

- In the **Color** box, select the line color. See [Defining Custom Colors on page 100](#) for instructions on defining custom colors.
- Specify the line width (pen size) or line size using **Width** and **Size**.

Note

Line width and line size are mutually exclusive; you can only specify one of these two settings. The difference between line width (pen size) and line size is explained in [Line Width vs. Line Size on page 95](#).

The last line setting you defined is displayed in the **Format** group on the **Drafting** tab. By default, the line settings for geometry are shown. Note that this information is context sensitive. For example, when you draw construction geometry, the line settings for construction geometry are shown.

To select a linetype, line color, line width or line size, simply select the required setting in the **Format** group on the **Drafting** tab:

- Click **Drafting** and then, in the **Format** group, click the arrow next to **Line Type** and select the required linetype.
- Click **Drafting** and then, in the **Format** group, click  **Select Color** and select the required color.

Creating User-Defined Linetypes

In addition to the standard linetypes supplied with Creo Elements/Direct Drafting (see [Figure 8 on page 92](#)), you can quickly and easily create your own linetypes through a repeated sequence of dot and dash patterns.

To create a user-defined (custom) linetype:

- Click **File** ► **Settings** ►  **Line Patterns**.

The **Line Pattern** dialog box opens.

Alternatively, click **Drafting** and then, in the **Format** group, click the arrow next to **Line Type**. Select the last option to open the **Line Pattern** dialog box.

- Click **Add**.

The **Add Line Pattern** dialog box opens.

-
3. In the **Name** box, type a name for the new line pattern.
 4. In the **Pattern** box, type one of the following:
 - A sequence of 1s and 0s, in which each 1 denotes a dot and each 0 denotes a gap. Note that the first number of this sequence must not be a 0. For example:
1111011110000
 - A sequence of comma-separated natural numbers, in which the first (third, fifth, and so on) number denotes the number of dots and the second (fourth, sixth, and so on) number denotes the number of gaps. (Creo Elements/Direct Drafting will convert any sequence of 1s and 0s to this notation.) For example:
4, 1, 4, 4

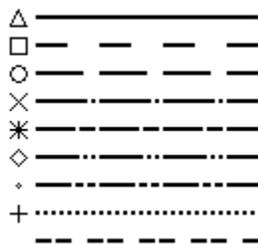
 **Note**

Both of these examples produce the same linetype.

5. Click **OK**.

The following figure gives you a visual impression of the new linetype defined in the two examples above.

Figure 9. Eight Built-In Linetypes Plus a User-Defined Linetype



 **Note**

The newly defined linetype pattern is appended to the list of built-in linetypes.

To apply your new linetype, follow the instructions in [Setting Linetype, Line Color and Line Width or Line Size on page 92](#).

Exporting User-Defined Linetypes

Any user-defined linetypes are saved to your current MI file. By sending this MI file to another Creo Elements/Direct Drafting user, you can make your user-defined linetypes available to this user.

These additional linetypes will be listed as "imported" linetypes in the linytype menu.

Editing User-Defined Linetypes

To edit a user-defined (custom) linytype:

1. Click **File** ► **Settings** ►  **Line Patterns**.

The **Line Pattern** dialog box opens.

Alternatively, click **Drafting** and then, in the **Format** group, click the arrow next to **Line Type**. Select the last option to open the **Line Pattern** dialog box.

2. In the **Name** column, select the user-defined linytype you want to edit.
3. Click **Edit** and edit the linytype as per your requirements. Refer to [Creating User-Defined Linetypes on page 93](#) for guidance.

Deleting User-Defined Linetypes

To delete a user-defined (custom) linytype:

1. Click **File** ► **Settings** ►  **Line Patterns**.

The **Line Pattern** dialog box opens.

Alternatively, click **Drafting** and then, in the **Format** group, click the arrow next to **Line Type**. Select the last option to open the **Line Pattern** dialog box.

2. In the **Name** column, select the user-defined linytype you want to delete.
3. Click **Delete**.

A message prompting for confirmation of the linytype deletion appears.

4. Click **Yes**.

Line Width vs. Line Size

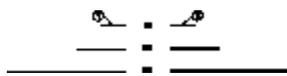
The thickness of elements can be set by setting the line width or line size properties.

Note that line width differs from line size:

- Line width (pen size) and line size are mutually exclusive: Specifying linesize resets pensize to 0, and specifying pensize resets linesize to 0.

- When you scale (increase or decrease the size of) an element drawn in a given line size, this line size will change with the size of the element.
- When you scale an element drawn in a given line width, the line width remains unchanged even if you scale the element. This behavior reflects the pen width concept of a conventional drawing board: an enlarged detail of a part is drawn with the same pen as the part.
- Line size supports HP-GL/2 and PostScript.

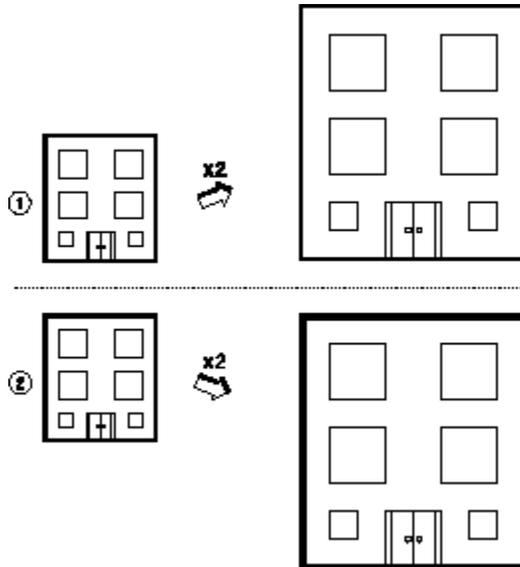
Figure 10. Scaling with Pensize and Linesize



Item	Description
1	Line Width (Pen Size): The line is non-scalable. This means that if you create a line having a pensize of 1mm, and then scale it by a factor of 2, the thickness of the line remains 1mm.
2	Line Size: The line is scalable. In this case, if you create a line having a linesize of 1mm, and then scale it by a factor of 2, the new thickness of the line is 2mm.

Figure 11 on page 97 shows one benefit of specifying pensize and linesize for specific elements.

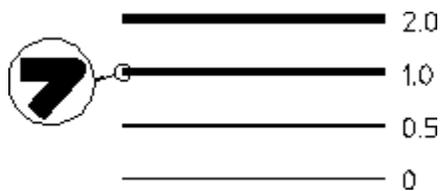
Figure 11. Difference between Pensize and Linesize



- 1 All geometry elements are drawn with the property pensize. When scaled, all lines remain the same size (thickness).
- 2 The outline of the building is drawn with the property linesize. All other elements are drawn with the property pensize. When scaled, the thickness of the outline increases but the other elements remain the same thickness.

Figure 12 on page 97 shows some sample sizes. The ends of lines are rounded so that when more than one line intersects, a rounded end is formed. Otherwise, the line end would appear stepped.

Figure 12. Lines Drawn with Different Sizes



Mapping Pen Color, Linetype and Pen Width

The **Pens** map is based on eight pens. This map enables you to map pen color, linetype, and pen width. The **Pens** map is available for:

-  **Geometry**
-  **Refline**

-
-  **Symmetry Line**

-  **Center Line**

To set up the **Pens** map:

1. Click **File** ► **Settings** and open the settings dialog box of any of the entity types listed above.

For example, click **File** ► **Settings** ►  **Geometry**.

The **Geometry Settings** dialog box opens.

2. In the **Line** area, click **Pens** to display the **Pens** map.
3. Click the row that you want to change.

The selected row gets highlighted.

4. Click **Change**.

The **Pen Settings** dialog box opens.

5. In the **Type** box, select a linetype.
6. In the **Color** box, select a color.
7. In the **Width** box, type the line width (pen size) that you want to associate with this color and linetype combination.

The difference between line width (pen size) and line size is explained in [Line Width vs. Line Size on page 95](#).

8. Click **OK** to return to the **Pens** map.
9. Repeat this procedure till the **Pens** map contains the required combinations of pen color, linetype, and pen width.

Optionally to save your current **Pens** map in a file, click **Save**. To load a saved **Pens** map for re-use or editing, click **Input**.

Changing the Linetype, Color, Line Width or Line Size of Existing Elements

To change the linetype, color, line width or pen width of existing geometrical element:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. To change the linetype:

- a. Under **Modify**, click  **Line Type**.

-
- The **Change Line Type** dialog box opens.
- b. In the **Type** box, select the required linetype.
 - c. Click **OK**.
3. To change the color:
- a. Under **Modify**, click  **Color**.
The **Select Color** dialog box opens.
 - b. In the **Color** box, select the required color.
 - c. Click **OK**.
4. To change the pen width:
- a. Under **Modify**, click  **Pen Width**.
 - b. Type the new pen size in the user input line and press ENTER.
5. To change the line width:
- a. Under **Modify**, click  **Line Width**.
 - b. Type the new line size in the user input line and press ENTER.
6. Select the elements that you want to modify.

You can change the settings before selecting the required elements.

Alternatively, you can use the CMT to change the linetype, color, pen width, or line width of existing geometrical elements.

 **Note**

- The linetypes you choose on the screen will be plotted unless plot transformation has been altered (see `PLOT_TRANSFORMATION` and `SCREEN_TRANSFORMATION` in the online Help).
 - When plotted, `PHANTOM` and `CENTER_DASH_DASH` linetypes look the same.
 - You can use the `LINEPATTERN` function to define your own linetype. Refer to Writing Macros with Creo Elements/Direct Drafting. If the linetype you have defined cannot be displayed or plotted, Creo Elements/Direct Drafting defaults to the nearest standard linetype shown in [Figure 8 on page 92](#).
 - The spacing of all broken linetypes is set using the plot default option `Ltype Len` in `PLOT`.
-

Changing Color

You can select or change colors for geometry, construction geometry, text, hatching and dimensioning.

The colors available for you to use are as follows:

- Black and White.
- The six basic colors:
Red, Yellow, Green, Cyan, Magenta, and Blue.
- Custom (user-defined) colors.

Note

The colors you choose are linked to the pen transformation so that you can allocate plotter pens to the colors displayed on the screen. (See PLOT_TRANSFORMATION in the online Help.)

Defining Custom Colors

To define custom colors for use in Creo Elements/Direct Drafting:

1. In any of the **Color** box, select **Custom**.

The **Color** dialog box opens.

The **Basic colors** area displays a number of basic color.

2. Click the basic color you want to add to your color selection list.
3. To create colors other than the basic colors, click **Define Custom Colors**.
4. Move the color selection reticule to change the associated hue, saturation, luminosity, red, green and blue values.

These values are indicated in **Hue**, **Sat**, **Lum**, **Red**, **Green**, and **Blue** box respectively. Optionally, you can replace these values to create a custom color.

5. When the required color appears in the **Color|Solid** box, click **Add to Custom Colors**.

The added custom color appears in the **Custom colors** area of the **Color** dialog box.

6. Optionally, repeat the last two steps to create additional custom colors.
7. Click **OK**.

Note

The last custom color you defined becomes the current (active) color in the **Color** box.

To access your other custom colors (if any), select **Custom** in the **Color** box.

The Dithering Effect

The ability to display the various RGB and HSL colors is dependent on your hardware. The number of colors that can be displayed typically ranges from 16 to 256 (or a much larger number) according to your graphics hardware.

However, since not all hardware has the ability to display the range of colors the RGB and HSL color system offers, a dithering effect is used to increase the number of shades available. This creates the impression of a larger range of colors than the hardware actually supports.

This dithering effect creates patterns of dots in the available colors. The eye then combines them into a perceived color different from the actual colors used to produce the patterns.

Although dithering can create a larger range of colors for surfaces, the drawing lines and the status bar can be displayed only in the range of colors supported by the hardware.

Setting the Color for Vertices

To set the color of vertices in your Creo Elements/Direct Drafting drawings:

1. Click **File** ► **Settings** ►  **Geometry**.
The **Geometry Settings** dialog box opens.
2. In the **Vertex Color** area, select the required colors for the vertex.
Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.
3. Select the required line type for the parts spotlight.

Viewing and Converting Colors

You can view information about the colors of all elements in a part and its subparts using the **Color Conversion** dialog box. You can also convert the colors of existing elements in a part and its subparts, to other colors.

To open the **Color Conversion** dialog box:

- Click **File** ► **Settings** ► **Color** ► **Convert Colors**.

You can use the **Color Conversion** dialog box for:

- Viewing color information of all elements in a part and its subparts.
- Converting colors of elements in a drawing.
- Storing the color conversion map file.
- Loading the color conversion map file.

Viewing Color Information

The **Color Conversion** dialog box provides information about :

- Color information table
- Preview viewport
- Parts list
- Preview colors
- Background colors

Color Information Table

The color information table of the **Color Conversion** dialog box consists of the following columns:

- **Seq Num**: Sequence number of a color in the table.
- **Source Color**: Color present in the drawing.
- **Target Color**: A color to map the source color to, if required.
- **Count**: Number of elements in the drawing that have the source color.

The standard colors in a drawing are represented by their names in the color information table. For other colors, Creo Elements/Direct Drafting displays the RGB value in the color information table.

Note

You can only edit the **Target** column of the color information table.

Preview Viewport

The preview viewport of the **Color Conversion** dialog box displays the preview of the drawing.

Note

For big drawings, Creo Elements/Direct Drafting displays parts that are smaller than a certain limit as magenta boxes (instead of entire parts) during the redraw process, by default. Creo Elements/Direct Drafting enables REDRAW_MODE PARTS_AS_BOX during the redraw process. As a result, many of magenta-colored elements are visible in the preview viewport (even though the magenta color may not be present in the drawing). To avoid this situation, Creo Elements/Direct Drafting disables REDRAW_MODE PARTS_AS_BOX by setting its value to zero when you open the **Color Conversion** dialog box. When you close the **Color Conversion** dialog box, the value of REDRAW_MODE PARTS_AS_BOX is restored.

Parts List

The **Part Id** box of the **Color Conversion** dialog box displays the list of parts in the drawing.

In the **Part Id** box, select a part ID to view the color map associated with the part ID.

Note

Creo Elements/Direct Drafting selects the part ID of the top-level part of a drawing by default.

Preview Colors

The **Preview** box of the **Color Conversion** dialog box displays the original colors, target colors, or a single user-defined color of drawing elements in the preview viewport.

In the **Preview** box, select one of the following:

- **Source** to view the drawing elements in their original colors.
- **Target** to view the drawing elements in the target colors that you have defined.
This is the default.
- **One Source Color** to view the drawing elements in a single color.
Creo Elements/Direct Drafting prompts you to select a source color.
Click the cell which contains the color to define the single color.

Creo Elements/Direct Drafting displays the drawing elements that contain the single source color.

 **Note**

- You can view a single source color of the part which you select in the **Part Id** box.
 - **OK** is disabled when **One Source Color** preview is active.
-

Background Colors

The **Background** box of the **Color Conversion** dialog box displays the elements of the drawing in the preview viewport according to the color scheme.

In the **Background** box, select any of the following:

- **Neutral** to view all the elements against a neutral-grey color background.
This is the default.

 **Note**

Elements having a color which is similar to the neutral background color are not visible in the preview viewport. When you open the **Color Conversion** dialog box, Creo Elements/Direct Drafting displays a warning if a drawing contains elements that have the same color as the neutral background color.

- **Creo** to view the elements against the default background color of **Creo Colors** scheme.
- **CoCreate** to view the elements against the default background color of **CoCreate** scheme.

 **Note**

For information about the color schemes, see [Setting the Default Color Scheme on page 63](#).

Converting Colors

To convert the color of existing elements in a drawing to another color:

1. Click **File** ► **Settings** ► **Color** ► **Convert Colors**.

The **Color Conversion** dialog box opens.

2. In the **Part Id** box, select the required part.

The color scheme of the part is displayed in the color information table.

3. In the color information table, click the **Target Color** cell for the **Source Color** cell which you want to change.

The **Color** dialog box opens.

4. Select the required color in the **Color** dialog box.

The selected color is displayed in the **Target Color** cell of the color information table.

Tip

- To remove a target color, right-click the required **Target Color** row in the color information table and select **Clear**.
- To remove all target colors, right-click the color information table and select **Clear All**.

Note

If there are duplicate source or target colors in the source-to-target color mapping, Creo Elements/Direct Drafting displays a warning if any of the following situations occur:

- If you define the same color as a source and a target color and the source color instance has no target color mapped to it. For example, if blue is the source color (not mapped to a target color), and the same blue color is defined as the target color for the green source color.
- If you define the same color as a target color more than once.

In addition, Creo Elements/Direct Drafting displays the sequence number of the duplicate colors in a bold orange color (RGB 255 128 0) in the color information table.

5. Click **OK** in the **Color Conversion** dialog box.

Note

- If you click **Clear All**, Creo Elements/Direct Drafting removes the target colors in the **Target Color** column of the color information table. Also, Creo Elements/Direct Drafting removes rows that contain color count as 0 (zero).
 - Creo Elements/Direct Drafting supports  **Undo** and  **Redo** for color conversion.
-

Storing the Color Conversion Map File

You can save your source-to-target color preferences in a map file after you have defined the target colors in the **Color Conversion** dialog box. The source-to-target color mapping is stored in a text file. The colors are stored as RGB values. Each source-to-target color pair is stored on a new line. For example, the source-to-target color mapping for black-to-white color conversion and red-to-yellow color conversion is saved as:

```
0, 0, 0; 255, 255, 255; (black-to-white color conversion)
255, 0, 0; 255, 255, 0; (red-to-yellow color conversion)
```

To store the source-to-target color mapping:

1. Click **File** ► **Settings** ► **Color** ► **Convert Colors**.

The **Color Conversion** dialog box opens.

2. Define the target colors in the **Target Color** column of the color information table.
3. Click **Store**.

The **Save As** dialog box opens.

4. Specify the path and the file name, and save the file.

Loading Color Conversion Map File

Creo Elements/Direct Drafting allows you to load a map file which contains predefined or user-defined source-to-target mapping of colors to convert the colors of a drawing.

To load a color map file:

1. Click **File** ► **Settings** ► **Color** ► **Convert Colors**.

The **Color Conversion** dialog box opens.

-
2. In the **Preset** box, select one of the following:
 - **None**
This is the default. No color map is loaded.
 - **CoCreate to Creo** (predefined color map)
 - **Creo to CoCreate** (predefined color map)
 3. To load a user-defined source-to-target color map file, select **User defined** in the **Preset** box and select the map file in the **Open** dialog box.

If there are duplicate source or target colors in the source-to-target color mapping, Creo Elements/Direct Drafting displays a warning. See [Converting Colors on page 105](#) for more information.
 4. Click **OK**.

 **Note**

- If a source color which is defined in the map file is present in the drawing, the corresponding target color is set in the **Target Color** column of the color information table.
 - If a source color which is defined in the map file is absent in the drawing, a row (containing the color information) is added to the color information table and the count is set to 0 (zero). You can view the information when you select **Source** in the **Preview** box.
-

Changing the Catch Mode

The Catch box enables you to select a point on the screen without putting the cursor directly on that point. The cursor will automatically "snap" to this point. The current catch mode is always displayed in the status bar. The system will catch to a given point only if the point lies within the (imaginary) circle around the edges of the cursor.

By default, the catch mode is set to **ALL**. You can change the catch mode to any of the following:

Setting

VERTEX

ELEM

INTERSECTION

CENTER

Catches to

Vertex

Element

Intersection

Center point

Setting	Catches to
GRID	Ruler/grid point
OFF	Disables Catch

Select the catch mode in the Catch box on the status bar.

When you select **ALL** in the Catch box, Creo Elements/Direct Drafting catches using the following order of priority:

1. **VERTEX**
2. **INTERSECTION**
3. **GRID**
4. **ELEM**

If the system finds a vertex within the catch range it catches to the closest such vertex; otherwise it searches for intersections and so on. If the system can't find any of these, it uses the given input point.

Note

By default, catch mode ELEM is used, when moving elements like geometry or parts via handles. To use a different catch mode than ELEM, type MH_USE_CATCH_MODE ON in the user input line. Now, the current catch mode is used to move the elements via handles.

Measuring Distances

Creo Elements/Direct Drafting allows you to obtain values for arc lengths, distances, point coordinates, angles, and areas using the  **Measure** buttons. These buttons are useful if you need to specify a value measured from existing features on the drawing. To illustrate this, consider the following example.

Example

Suppose you want to draw a vertical line starting at 100,100 and of the same length as an existing line drawn at an angle of 40 degrees:

1. In the **Utilities** group, click  **Measure**.
2. Click  **Length**.
3. Click the existing line.

The measured length of this line is displayed in the user input line.

-
4. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
 5. Click  **Vertical**.
 6. Specify the begin point of the vertical line you want to create.
 7. When prompted for the end point or the length of this vertical line, type the measured length in the user input line and press ENTER.

Creo Elements/Direct Drafting draws the vertical line as specified. The length of this line is the same as that of the line you measured.

 **Note**

- **Follow** (described later in this section) is not active while  **Measure** is active.
-

Measuring Distance

To measure distance between any two points:

1. In the **Utilities** group, click  **Measure**.
2. Click  **Distance**.
3. Click the begin point and the end point in the drawing to measure the required distance.

The distance between the two points is displayed in the user input line.

Measuring a Radius

To measure the radius of a curved element:

1. In the **Utilities** group, click  **Measure**.
2. Click  **Radius**.
3. Click the curved element whose radius you want to measure.

The radius of the curved element is displayed in the user input line.

Measuring Lengths

To measure the length of any element (line, circle, arc, fillet, chamfer or spline):

-
1. In the **Utilities** group, click  **Measure**.
 2. Click  **Length**.
 3. Select the element.

The length of the element is displayed in the user input line.

Measuring Horizontal Distance

To measure horizontal distance between two points:

1. In the **Utilities** group, click  **Measure**.
2. Click  **Horizontal**.
3. Click two points in the drawing to identify the horizontal distance you want to measure.

The horizontal distance between the two points is displayed in the user input line.

Note

This option is dependent on the current catch mode.

Measuring Vertical Distance

To measure vertical distance between two points:

1. In the **Utilities** group, click  **Measure**.
2. Click  **Vertical**.
3. Click two points in the drawing to identify the vertical distance you want to measure.

The vertical distance between the two points is displayed in the user input line.

Note

This option is dependent on the current catch mode.

Displaying Point Coordinates

To display the exact coordinates of any existing point:

1. In the **Utilities** group, click  **Measure**.
2. Click  **Point**.
3. Click the point.

The exact coordinates of this point are shown in the user input line.

Note

This option is dependent on the current catch mode.

Measuring Angles

To measure the angle between two lines:

1. In the **Utilities** group, click  **Measure**.
2. Click  **Angle**.
3. Click a point on the first line.
4. Click a point on the second line.

Similarly, to measure an arc segment angle, click the arc start and end points.

The angle between two tangents drawn at an intersection point of two circles can be measured by clicking a point on the first circle and then a point on the second circle.

Measuring Area Properties

A Single Face

To calculate the area and area properties of a selected face, the face must be completely bounded by elements.

Creo Elements/Direct Drafting calculates and displays the following area properties for a single face:

- Number of holes in the face.
- Density of the face.
- Perimeter length of the face (outer perimeter).

-
- Area.
 - Center of Area (= Center of Mass).
 - Principal Axes (of Inertia through the Center of Area).
 - Second Moments of Area about principal axes.
 - Second Moments of Area about coordinate system axes.
 - Product of Second Moment of Area about coordinate system axes.
 - Moments of Inertia about principal axes.
 - Moments of Inertia about coordinate system axes.
 - Product of Moment of Inertia about coordinate system axes.
 - Section Moduli (about principal axes).
 - Distance from first neutral axis to extreme fiber.
 - Distance from second neutral axis to extreme fiber.
 - Radii of Gyration.
 - Angle between coordinate system and principal axes.

Multiple Faces

If you identify multiple faces, Creo Elements/Direct Drafting uses the density values to calculate the Common Center of Mass. The density must be specified before a face is identified, otherwise the default value (1) or the previous density value is used.

Creo Elements/Direct Drafting calculates and displays the following total values for multiple faces:

- Total Area.
- Center of Area (common).
- Center of Mass (common).
- Principal Axes through the (common) Center of Area.
- Second Moments of Area about (common) Principal Axes.
- Second Moments of Area about Coordinate System Axes.
- Product of Second Moment of Area about Coordinate System Axes.
- Moments of Inertia about (common) Principal Axes.
- Moments of Inertia about Coordinate System Axes.
- Product of Moment of Inertia about Coordinate System Axes.
- Radii of Gyration with respect to the (common) Center of Area.
- Angle between Coordinate System and (common) Principal Axes.

To display face area properties:

1. In the **Utilities** group, click  **Properties**.
2. Click  **Area**.
3. Type the density value in the user input line and press ENTER.
The default density value is 1.
Alternatively, select a face or faces.
4. Click  **Confirm**.
The area properties are displayed in a standard text editor.
Optionally, you can save the list of area properties in a file.
5. Exit the standard text editor to return to Creo Elements/Direct Drafting.

Example

Figure 13. I-Section Face on page 113 shows an I-section face.

Figure 13. I-Section Face

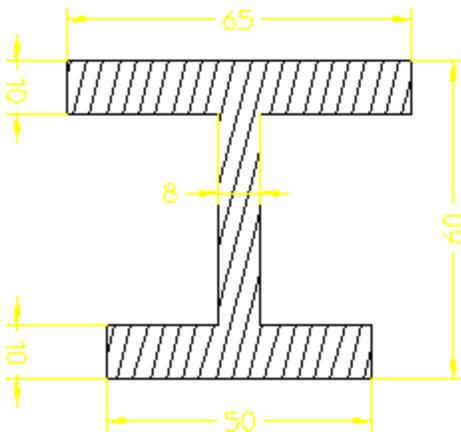
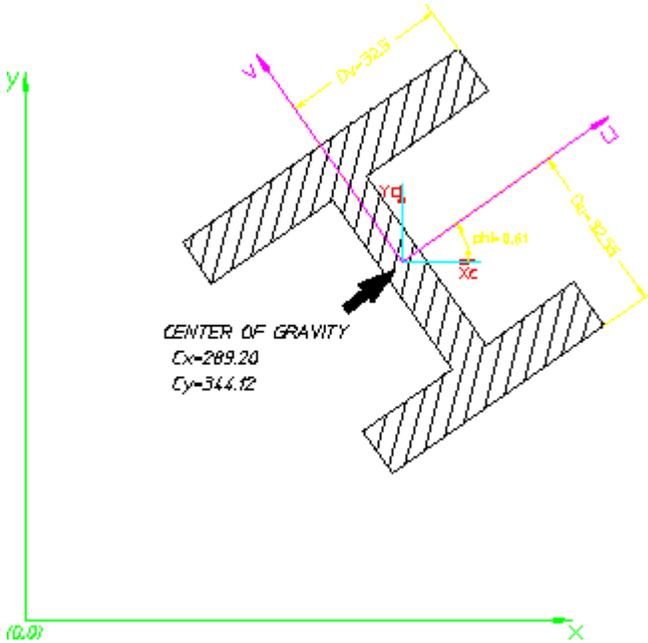


Figure 14. Resultant Diagram for I-Section Face on page 114 is a resultant diagram for the I-section face.

Figure 14. Resultant Diagram for I-Section Face



A listing of the Creo Elements/Direct Drafting output for this single face is given below:

ALL VALUES REFER TO THE FOLLOWING UNITS:

LENGTH = 1 MM

ANGLE = 1 DEG

FACE 1:

NUMBER OF HOLES

noh = 0

DENSITY

rho = 1

PERIMETER LENGTH

P = 334

AREA

A = 1470

CENTER OF AREA = CENTER OF MASS

(Cx, Cy) = (289.2011071614633, 344.1233317074434)

PRINCIPAL AXES OF INERTIA THROUGH THE CENTER OF AREA

(DIRECTIONS)

u = (0.8191520442890209, 0.5735764363510046)

v = (-0.5735764363510046, 0.8191520442890209)

SECOND MOMENTS OF AREA (ABOUT PRINCIPAL AXES)

Icu = 761433.6734694783

Icv = 334727.5

SECOND MOMENTS OF AREA (ABOUT COORDINATE SYSTEM AXES)

Ix = 174699726.7554225

Iy = 123421911.6970526

PRODUCT OF SECOND MOMENT OF AREA (ABOUT COORDINATE SYSTEM AXES)

$I_{xy} = 146095161.0177031$
MOMENTS OF INERTIA (ABOUT PRINCIPAL AXES)
 $J_{cu} = 761433.673469478$
 $J_{cv} = 334727.5$
MOMENTS OF INERTIA (ABOUT COORDINATE SYSTEM AXES)
 $J_x = 174699726.755422$
 $J_y = 123421911.697053$
PRODUCT OF MOMENT OF INERTIA (ABOUT COORDINATE SYSTEM AXES)
 $J_{xy} = 146095161.017703$
SECTION MODULI ABOUT PRINCIPAL AXES
 $Z_{cu} = 23392.00626959447$
 $Z_{cv} = 10299.30769231674$
DISTANCE FROM NEUTRAL AXIS u TO EXTREME FIBER
 $D_u = 32.55102040816436$
DISTANCE FROM NEUTRAL AXIS v TO EXTREME FIBER
 $D_v = 32.5$
RADIOI OF GYRATION WITH RESPECT TO THE CENTER OF AREA
 $R_{cu} = 22.7592199074477$
 $R_{cv} = 15.08992320434836$
ANGLE BETWEEN COORDINATE SYSTEM AND PRINCIPLE AXES
 $\phi = 35$

Using the Calculator

Creo Elements/Direct Drafting provides a standard MS Windows calculator.

To use the scientific calculator:

1. In the **Utilities** group, click  **Calculator**.
The **Calculator** application opens.
2. On the **View** menu of the **Calculator**, click **Scientific**.

Using the Standard Text Editor

Creo Elements/Direct Drafting provides a standard text editor.

To use the standard text editor:

- Click **Miscellaneous** and then, in the **System** group, click  **Edit File**.
A blank text file opens.

Using the Ruler or Grid

The ruler simulates the T-square traditionally used with manual drafting methods. When used together with the grid point catch mode, the ruler allows you to draw with the help of a grid.

The grid is a useful drawing aid for alignment purposes.

Displaying a Ruler and/or a Grid

To display the ruler and/or the grid:

1. Click **File** ► **Settings** ► **Grid**.
The **Grid Settings** dialog box opens.
2. In the **Grid** area, click **Dot** or **Line** to display a dot or line grid, respectively.
3. In the **Ruler** area, click **Ruler** to display the ruler.
4. Click **OK**.

Displaying the Origin

To display the coordinate origin:

1. Click **File** ► **Settings** ► **Grid**.
The **Grid Settings** dialog box opens.
2. In the **Ruler** area, click **Origin**.
The coordinate origin is indicated in the chosen viewport by an axes symbol.
If the ruler is active in the chosen viewport, it becomes unavailable.
3. Click **OK**.

Setting the Ruler or Grid Spacing

To change the spacing between the tick marks of the ruler or between the grid lines or dots:

1. Click **File** ► **Settings** ► **Grid**.
The **Grid Settings** dialog box opens.
2. In the **Options** area, type the required distance (for example, 0 . 5) in the **Spacing** box.

Note that this entry is interpreted in terms of the current length unit.

If the spacing is very small, it may not be shown on the screen but it will still be used for catch when in ruler or grid catch mode. Use the buttons on the **View** tab if you want an enlarged view.

Setting Cursor Size

You can use either a small cursor or a full screen cursor. A full screen cursor is particularly recommended for use with the ruler.

To change the cursor size:

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. Click **Environment**.
3. In the **Cursor** area, click the **Large** check box to use the large (full screen) cursor.
4. In the **Cursor Size** box, type the size for the cursor.

The catch range is not affected by the cursor size. Optionally, you can specify a catch range in the **Catch Range** box.

Using Follow

The ruler is always located at the origin. The origin always tracks the ruler, and the ruler always tracks the origin. Optionally, you can use the **Follow** check box to move the coordinate system origin with the origin of the ruler.

To make the origin follow the ruler:

1. Click **View** and then, in the **Viewing** group, click  **Grid**.
2. Click the **Follow** check box.

The ruler automatically moves to each new point you create.

Moving the Ruler

To move the ruler:

1. Click **View** and then, in the **Viewing** group, click  **Grid**.
2. Click **Move**.
3. Specify the new origin.

The orientation of the ruler axes remains the same.

Placing the Ruler or Grid

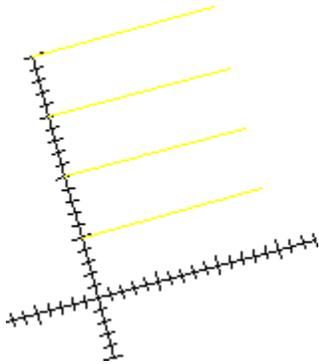
To change the orientation of the axes:

1. Click **View** and then, in the **Viewing** group, click  **Grid**.

2. Click **Place**.
3. Specify the new origin and two more points that define the new axis orientation.

Figure 15 on page 118 shows the ruler axes.

Figure 15. Ruler Axes



Rotating the Ruler or Grid

With Respect to the X-Axis

To rotate the ruler (or grid) so that the x-axis is at a given angle to the horizontal:

1. Click **View** and then, in the **Viewing** group, click  **Grid**.
2. Click **Turn**.
3. Click **Absolute**.
4. Type the required angle in the user input line and press ENTER.

Through a Given Angle

To rotate the ruler (or grid) through a given angle:

1. Click **View** and then, in the **Viewing** group, click  **Grid**.
2. Click **Turn**.
3. Click **Relative**.
4. Type the required rotation angle in the user input line and press ENTER.

A positive value results in counterclockwise rotation.

Hiding the Ruler or Grid

To hide the ruler or grid:

1. Click **View** and then, in the **Viewing** group, click  **Grid**.
2. Click **Hide**.

Hide disables the ruler and/or grid. Alternatively, you can click **None** in the **Ruler** area of the **Grid Settings** dialog box.

Drawing an Isometric Ruler or Grid

To redraw the ruler or grid according to the isometric grid required for drawing top, front, or side views:

1. Click **View** and then, in the **Viewing** group, click  **Grid**.
2. Click **Isometric** and then click any of the following:
 - Click **Front** to draw a front view.
 - Click **Top** to draw a top view.
 - Click **Side** to draw a side view.

Selecting What is Displayed on Your Screen

This section describes how to use the **Show** dialog box to control exactly what is displayed on your screen.

To open the **Show** dialog box:

- Click **File** ► **Settings** ► **Show**.

Alternatively, in the **Utilities** group, click **Show**.

Show operates in the current viewport.

Before you plot your complete drawing, you are recommended to check what is included by clicking **Show All** in the **General** tab of the **Show** dialog box.

You can selectively display:

- Construction geometry
- Hatch
- Dimension
- Text
- Geometry

-
- Vertex
 - Info
 - Layers
 - URLs

 **Note**

- The **Show** settings apply to the current viewport.
 - Any new viewports created automatically adopts the settings of the viewport that was active when they were created.
-

Showing All Features

To display all the features of your drawing:

1. Click **File** ► **Settings** ► **Show**.

Alternatively, in the **Utilities** group, click **Show**.

The **Show** dialog box opens.

2. On the **General** tab, click **Show All**.

It is useful to start with everything displayed, and then clear the check box of items that are not required.

Showing Points

To display all existing points on the drawing:

1. Click **File** ► **Settings** ► **Show**.

Alternatively, in the **Utilities** group, click **Show**.

The **Show** dialog box opens.

2. On the **General** tab, click the **Vertex** check box.

This is useful if you want to identify a point more accurately, or to check if two lines overlap. Points are displayed in the following colors (by default):

- GREEN – end points
- YELLOW – common end points
- BLUE – single center points
- MAGENTA – multiple center points

Showing Dimensions

To display all dimensions on a drawing:

1. Click **File ▶ Settings ▶ Show**.

Alternatively, in the **Utilities** group, click **Show**.

The **Show** dialog box opens.

2. On the **General** tab, click the **Dimension** check box to display dimensions as a group.

Any dimensions created after you clear the **Dimension** check box are not displayed on the screen till the **Dimension** check box is clicked.

Changing the Layer Color

To change the color of a specified layer:

1. Click **File ▶ Settings ▶ Show**.

Alternatively, in the **Utilities** group, click **Show**.

The **Show** dialog box opens.

2. Click the **Layer** tab.
3. Select the layer number.
4. Select a color from the **Color** box.
5. Click **Color**.

Displaying Infos

To display elements that contain attached infos:

1. Click **File ▶ Settings ▶ Show**.

Alternatively, in the **Utilities** group, click **Show**.

The **Show** dialog box opens.

2. Click the **Info Elem.** tab.
3. Select the info to be displayed.
4. Click **On** to display elements that have information attached.

Viewing a Drawing

This section describes how to use the normal and dynamic **View** functions to see details in your drawing more clearly. You can zoom-in on the area concerned, or reposition the drawing on the screen.

The buttons in the **Viewing** group on the **View** tab operate on either the current viewport (default) or on a viewport that you select.

Selecting a Viewport

You can select the viewport in which the **Viewing** group buttons operate.

Current Viewport

If you want to use buttons in the **Viewing** group on the **View** tab for operating in the current viewport:

1. Click **View** and then, in the **Window** group, click **Current**.
2. Click in the viewport or type the viewport number in the user input line and press ENTER.

Creo Elements/Direct Drafting uses the current viewport for any buttons in the **Viewing** group on the **View** tab.

Select Viewport

To select a viewport:

1. Click **View** and then, in the **Window** group, click **Select**.
2. Click in the viewport to select it.

Using View Functions

This section describes how to control the area of the drawing shown in the current viewport using the buttons in the **Viewing** group on the **View** tab.

Viewing a Drawing in the Full Screen Mode

To view your drawing in the **Full Screen** mode:

- Click **View** and then, in the **Viewing** group, click  **Full Screen**.

Alternatively, press F11 to toggle between the **Full Screen** mode and the normal mode.

In the **Full Screen** mode, you can view the maximized drafting window. In addition, the ribbon is minimized and the **Part Browser** is hidden during the **Full Screen** mode.

Fitting a Drawing

To fit the current drawing in the selected viewport:

-
- Click **View** and then, in the **Viewing** group, click  **Fit**.

Restoring the Last View

To return to the previous view:

- Click **View** and then, in the **Viewing** group, click  **Last**.

Redrawing

To redraw the drawing in the chosen viewport:

- Click **View** and then, in the **Viewing** group, click  **Redraw**.

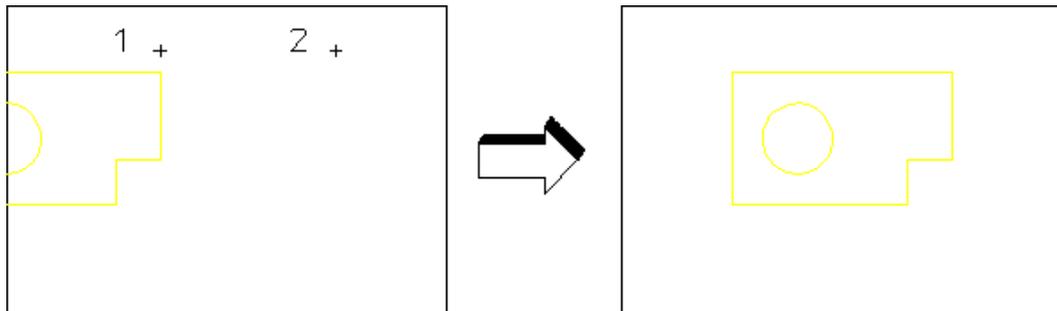
This is sometimes necessary if you have deleted several elements and traces are still visible on the screen. The contents of other viewports will not be redrawn.

Panning Across a Drawing

Panning is a very useful function that enables you to move across the drawing to view those areas not currently displayed.

Figure 16 on page 123 illustrates the use of panning.

Figure 16. Panning Across a Drawing



To pan in any direction:

1. Click **View** and then, in the **Viewing** group, click  **Pan**.
2. Click a point in your drawing.
3. Specify the new screen position for this point.

Alternatively, press the middle mouse button to pan to the new position.

Creating a New Center Point for Your Drawing

To specify the new center point for the display:

-
- Click **View** and then, in the **Viewing** group, click  **Center**.

Your drawing is repositioned so that this point is in the center of the chosen viewport.

 **Note**

 **Center** uses the current catch mode to identify the specified point.

Zooming In or Out

A window is always zoomed about its center point.

To zoom your window:

1. Click **View** and then, in the **Viewing** group, click  **Zoom**.
2. Type a zoom factor in the user input line and press ENTER.

A number greater than 1 results in a window that covers a smaller area (zooming in). A number less than 1 (>0) or negative (<-1) gives a larger coverage (zooming out).

Alternatively, move the mouse towards the screen to zoom in or move the mouse away from the screen to zoom out and click a point in the viewport.

Creating a New View in a Window

To create a new window:

1. Click **View** and then, in the **Viewing** group, click  **Window**.
2. Specify the two points (corners) of the area you want to enlarge.

Storing and Recalling Windows

If you are likely to need the same view again, you can store it and recall it as required

To store the current window coordinates:

- Click **View** and then, in the **Viewing** group, click  **Store**.

To recall the window that was stored last:

- Click **View** and then, in the **Viewing** group, click  **Recall**.

Searching for Text and Dimensions in the Viewport

You can search for text and dimensions in a drawing in the viewport using the buttons in the **Search** group on the **View** tab.

Searching for text in the viewport

1. Click **View** and then, in the **Search** group, click **Text**.
The **Search Text** dialog box opens.
2. In the **Text** box, type the text to be searched for.
Select the **Match case** and **Match whole word** check boxes as required.
3. The **Index** box indicates the current occurrence of the text.
4. Click **Next** or **Previous** to search for the next or previous occurrence of the text, respectively.

Searching for dimensions in the viewport

1. Click **View** and then, in the **Search** group, click **Dimension**.
The **Search Dimension** dialog box opens.
2. In the **Dimension** box, type the dimension to be searched for.
3. In the **Dim Type** box, select the dimension type.
4. In the **Search In** box, select the scope of the search, that is, the main dimension or a dimension text extension.
5. The **Index** box indicates the current occurrence of the dimension.
6. Select the **Match case** and **Match whole word** check boxes as required.
7. Click **Next** or **Previous** to search for the next or previous occurrence of the dimension, respectively.

Using Viewports to Display Several Views

You can divide the drawing area of the screen so that more than one view can be displayed. This is a very useful feature when you want to work on a detail drawing. The changes made to the detail are automatically repeated in the main drawing. Each division of the screen is called a viewport.

Viewports can be overlapped with other viewports, menus, and programmable tables. All overlapping viewports are continuously updated during your drawing session.

In the following example, the view in viewport 1 can be imagined as the view you would see if you were examining your drawing with a magnifying glass. To achieve this effect, simply zoom in to the drawing displayed in viewport 1 until you can see the required level of detail.

- Viewport 1 — contains a magnified view showing the required level of detail.
- Viewport 2 — contains the entire drawing and is used as a positional reference for the magnifying glass.

Examine the zoomed drawing in viewport 1 while moving the cursor (as a positional reference) in viewport 2. See [Figure 17 on page 126](#) and [Figure 18 on page 127](#).

Figure 17. Using Viewport 2 as a Reference

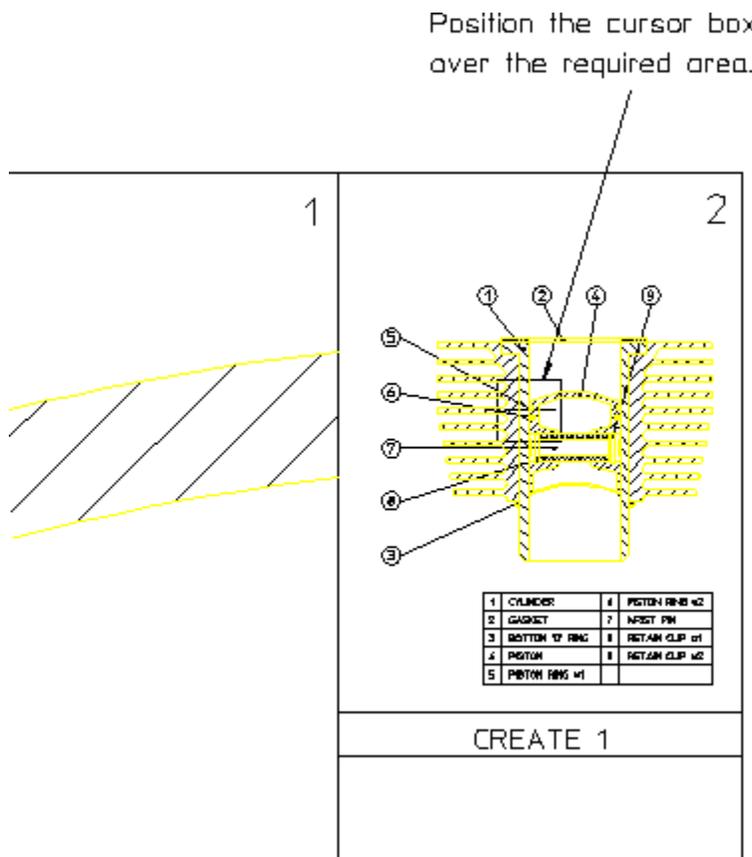
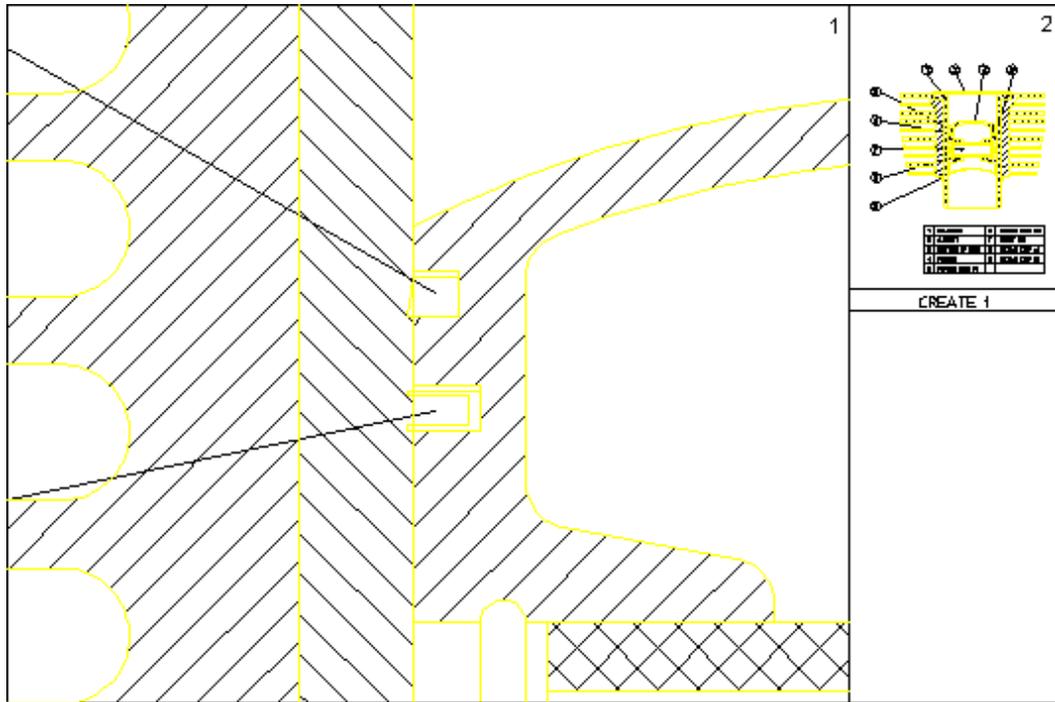


Figure 18. Viewing Piston Assembly Detail in Viewport 1



Creating a Viewport

To create a new viewport:

1. Click **View** and then, in the **Window** group, click  **New**.
2. Specify the new corner points for the new viewport.

The new viewport is marked by a sequential number in the top-right corner.

Making a Viewport Current

If a viewport is partially or totally covered by other viewports, it is made inactive. Its visual contents are not updated by any subsequent operations unless it is made the current viewport.

The current viewport is the one that is currently active, fully visible and up-to-date. The viewport number is displayed in inverse video in the top right corner.

To change the current viewport:

1. Click **View** and then, in the **Window** group, click **Current**.
2. Click the viewport required.

Alternatively, type the viewport number in the user input line and press ENTER.

Changing the Size of a Viewport

To change the size of a viewport:

1. Click **View** and then, in the **Window** group, click **Change**.
2. Click the required viewport.
3. Specify the new corner points of the viewport.

Moving a Viewport

To move a viewport:

1. Click **View** and then, in the **Window** group, click **Move**.
2. Click the viewport.
3. Drag the cursor to the new position in the viewport.

The contents of the viewport follow the viewport.

Maximizing the Size of a Viewport

You can maximize the size of any viewport such that the viewport occupies the entire Creo Elements/Direct Drafting drawing area.

To maximize the size of the viewport:

1. Click **View** and then, in the **Window** group, click **Large**.
2. Click the viewport.

Any other viewport are now hidden behind the maximized viewport. To bring any of these hidden viewport to the foreground, make it current. To change the size of the maximized viewport, use **Change**.

Redrawing a Viewport

To redraw a viewport:

1. Click **View** and then, in the **Window** group, click **Redraw**.
2. Click the viewport.

The contents of the viewport will be redrawn.

Deleting a Viewport

To delete a viewport:

1. Click **View** and then, in the **Window** group, click  **Delete**.
2. Click the viewport.

Alternatively, type the viewport number in the user input line and press ENTER.

Deleting All Viewports

To delete all viewports:

1. Click **View** and then, in the **Window** group, click  **Delete**.
2. Type ALL in the user input line and press ENTER.

All viewports are removed from the Creo Elements/Direct Drafting drawing area.

Note

To continue drawing or to view your current Creo Elements/Direct Drafting drawing after you have deleted all viewports, you need to create at least one new viewport.

Viewport Settings

The viewport settings are defined in the **Viewport Settings** dialog box.

To define the viewport settings:

1. Click **File** ► **Settings** ► **Viewport**.
The **Viewport Settings** dialog box opens.
2. In the **Border** area:
 - a. In the **Color** box, select a color for the viewport border.
 - b. In the **Width** box, type the viewport border width.
3. In the **Color of** area:
 - a. In the **Background** box, select the background color.
 - b. In the **Text** box, select the color of the text.
 - c. In the **Cursor** box, select the color of the cursor.

-
- Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.
4. In the **Invert** area, select one of the following in the **Invert Color** box to invert display colors in a viewport:
 - **White elements**

If you select **White elements**, the elements that are white (or almost white) are displayed in black. This is the default for the **Creo Colors** color scheme.
 - **Black elements**

If you select **Black elements**, the elements that are black (or almost black) are displayed in white. This is the default for the **CoCreate Colors** color scheme.
 - **Off**

If you select **Off**, element colors are not inverted in the viewport.
 5. In the **Print Preview** area:
 - a. In the **Width** box, type the preview width.
 - b. In the **Height** box, type the preview height.
 - c. In the **Background** box, select the background color.
 - d. Select the **Show Ruler** check box, to display the ruler.
 6. Select the **Smooth Window Update** check box, to provide more visual guidance when zooming into a selected area or when defining the window view.

The time duration for window update is ranged from 30ms to 3000ms.
 7. Click **OK**.

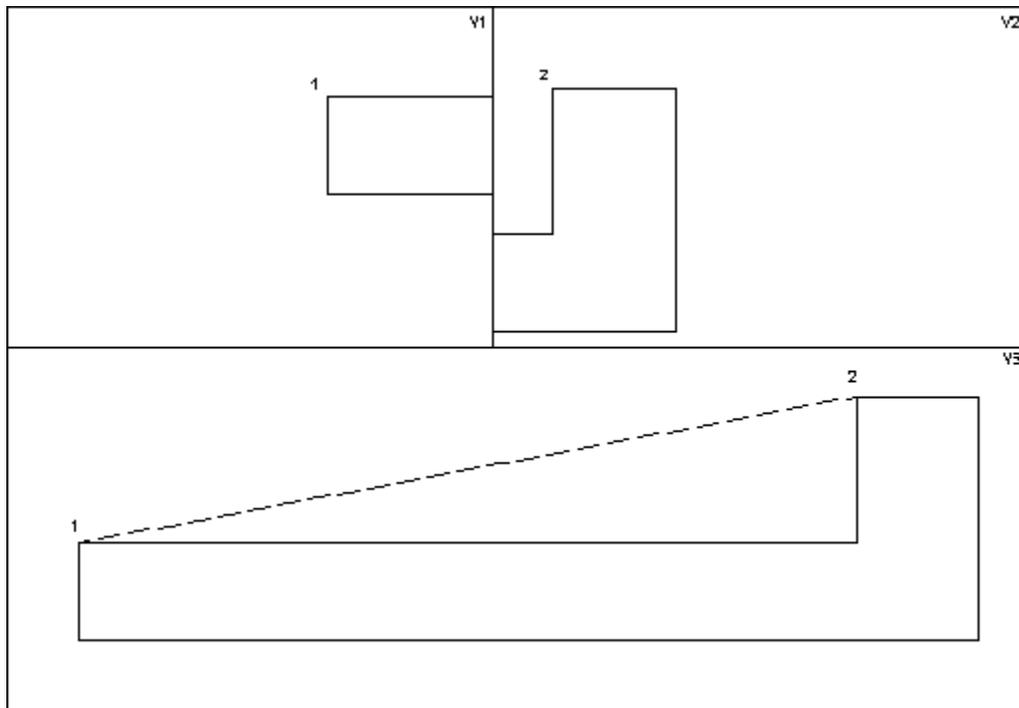
 **Note**

- The viewport colors are defined by the color scheme. The internal viewports (for example, the viewport in the **Symbols** dialog box) also display the default colors for the selected color scheme.
 - The smart zoom feature for files with large size takes longer than the specified time.
-

Viewport Example

A common application using viewports is identifying two (or more) widely separated points on a large and complex drawing. Refer to [Figure 19 on page 131](#).

Figure 19. Joining Two Widely Spaced Points Using Viewports



To draw a line between points 1 and 2 in [Figure 19 on page 131](#):

1. Create viewport V3.
2. Then create viewport V1 and V2.
3. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
4. Click  **2 Points**.
5. Specify point 1 in V1 and point 2 in V2.

Creo Elements/Direct Drafting draws a line between points 1 and 2.

Using Properties to Change Elements and Text

Line geometry, text and hatch have associated properties.

Geometry Properties

Geometry properties are:

- Color

-
- Linetype
 - Line width (pen size) and line size

These can be predefined using the **Geometry Settings** dialog box.

Text Properties

Text properties are:

- Font
- Origin vertex
- Angle
- Fill
- Frame
- Linespace
- Ratio
- Size
- Slant

These can be predefined using the **Text Settings** dialog box.

Hatch Properties

Hatch properties are:

- Pattern
- Angle
- Distance
- Reference Point

These can be predefined using the **Hatch Settings** dialog box.

Getting and Putting Properties

It is possible to retrieve properties (of  **Text**, for example) and use them either in subsequent work or to change existing properties.

Note

You cannot change dimension properties using  **Get** and  **Put**.

For example:

1. In the **Utilities** group, click  **Properties**.
2. Click  **Get**.
3. Select a block of text.

All properties of that text are retrieved. Any further text will now have these properties.

To change the properties of an existing text block to those of another:

1. In the **Utilities** group, click  **Properties**.
2. Click  **Get**.
3. Select the text whose properties you want to retrieve.
4. In the **Utilities** group, click  **Properties**.
5. Click  **Put**.
6. Select the text whose properties are to be changed.

Using the Toolbox

Creo Elements/Direct Drafting provides a special dialog box from which you can run, view, and administer macros. Up to 24 macros can be assigned to the **TOOLBOX** dialog box and you can define any number of macro-sets that can be called in the **TOOLBOX** dialog box.

To open the **TOOLBOX** dialog box:

- Click **Miscellaneous** and then, in the **System** group, click  **Toolbox**.

The **TOOLBOX** dialog box opens.

Recording a Macro

To record a macro using the **TOOLBOX** dialog box:

1. Click **ADD**.

Numbers appear in the boxes to help you identify the correct box. If macros are already assigned, their names are shortened to make space for the box numbers.

2. Click one of the 24 macro boxes.

-
3. Type a title for the macro between single quotes (' ') in the user input line and press ENTER.

The macro title can be up to six characters long.

4. Perform the required operations in Creo Elements/Direct Drafting.

Creo Elements/Direct Drafting records all your operations.

5. After you finish your operations, click **END**.

Creo Elements/Direct Drafting stops recording the macro and displays the macro name in the box you assigned to it.

Example

This example shows you how to record a macro named RT4020 that draws a rectangle and dimensions it (rectangle size is 40x20mm). The macro is assigned to box number 3.

To record the RT4020 macro using the **TOOLBOX** dialog box:

1. Click **ADD**.
2. Click box number 3.
3. Type `RT4020` in the user input line and press ENTER.
4. Click **Drafting** and then, in the **Draw** group, click **Rectangle**.
5. Type 0, 0 in the user input line and press ENTER.
6. Type 40, 20 in the user input line and press ENTER.
7. Click **OK**.
8. Click **END**.

The macro is now defined.

Running a Macro

To run a macro:

- In the **TOOLBOX** dialog box, click the macro name.

Renaming Macros

To rename a macro:

1. In the **TOOLBOX** dialog box, click **RENAME**.
2. Select the macro.
3. Type a new name for the macro in the user input line and press ENTER.

Deleting Macros

To delete a macro:

1. In the **TOOLBOX** dialog box, click **DELETE**.
2. Select the macro.

Editing Macros

After you have recorded a macro, you can modify its contents using **EDIT** in the **TOOLBOX** dialog box. For example, suppose you want to change the size of the rectangle you created in [Recording a Macro on page 133](#) to measure 38x22 instead of 40x20.

To edit the RT4020 macro:

1. In the **TOOLBOX** dialog box, click **EDIT**.
2. Select RT4020.

Creo Elements/Direct Drafting displays the macro string in the user input line.

3. Modify the old values and press ENTER.

The next time you use the RT4020 macro, the new rectangle is drawn. You may change the macro's name to reflect the new size.

Loading Macro-Sets

Although the **TOOLBOX** dialog box lets you display a maximum of 24 macros, you can group macros together into a file. Each file can contain up to 24 macros that can be loaded by the **TOOLBOX** dialog box. For example, if you have a file called `tb_rectangle` containing the macros:

```
RT4020
RT4025
.
.
.
RT5020
```

you can tell  **TOOLBOX** to use this file by clicking **INPUT** and then specifying the filename `tb_rectangle`.

Note

The **TOOLBOX** dialog box is a fixed size. Although you can reposition it around the screen, you cannot change its size.

Setting the Auto Save Time for Drawings

In addition to the **File** functionality described in [Storing, Loading and Filing on page 528](#), Creo Elements/Direct Drafting provides an autosave feature to automatically save a drawing at routine intervals.

- The name of the autosaved file is `me_backup.mi`. This file is put in the working directory as specified by `ME_WORKING_DIR`.
- If `ME_WORKING_DIR` is not defined, the file is stored in `\temp` on Windows-based systems.
- The autosave interval is a time interval in minutes. Possible values are 10 and 0 (0 means off; this is the default).
- While the autosave is running, a message pops up and the blue store bar displays the progress.

To set a time interval (in minutes) at which your drawing will be automatically saved to a file named `me_backup.mi`:

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. Click **Save**.
3. In the **Auto Save** area, click the **Auto Save Drawing** check box.
4. In the **Type** box, type a time interval (in minutes).
5. Click **OK**.

Setting the Delay Time for Cascaded Menus

If you find that cascaded menus are displayed too quickly or too slowly, proceed as follows to change the time delay for such menus:

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. Click **UI**.
3. In the **Menus** area, type the time delay (in milliseconds) in the **Cascade Timeout** box.
4. Click **OK**.

Setting the Redraw Mode

When dealing with large or complex drawings, you may want to change the redraw mode in order to enhance the graphics performance.

To define settings for the redraw mode:

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. Click **Redraw Mode**.
3. In the **Parts as Box** area:
 - a. In the **Smaller than** box, type a size threshold (in pixels) below which parts are only shown as boxes.
 - b. Optionally, in the **Box Color** box, select a color.
4. In the **Draw Dynamic** area:
 - a. Click the check box of the items that you want to include in the redraw.
 - b. Clear the check box of the items that you want to exclude from the redraw to enhance performance.
5. Click **OK**.

Saving the Current Drawing Environment

The current setup of your drawing environment can be saved to a file for re-use.

To save the current drawing environment:

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. Click **Save**.
3. In the **Save Settings** area, click the check box of the settings you want to save.
4. Click **OK**.

Further Information

For further technical information about the drawing environment and making measurements in Creo Elements/Direct Drafting, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

-
- help edit_environment
 - help wui_construction_settings
 - help wui_geometry_settings
 - help wui_dimension_settings
 - help wui_hatch_settings
 - help wui_text_settings
 - help wui_prp_settings
 - help wui_refline_settings
 - help wui_symmetry_line_settings
 - help wui_center_line_settings
 - help wui_viewport_settings
 - help wui_grid_settings
 - help wui_show
 - help wui_hidden_settings
 - help measure_angle
 - help measure_area
 - help measure_coordinate
 - help measure_distance
 - help measure_length
 - help measure_radius
 - help measure_tang_ang_at_pnt
 - help multiple_measure
 - help calculator
 - help catch
 - help window
 - help line_grid
 - help dot_grid
 - help cs_set

Creo Elements/Direct Drafting displays further information about the drawing environment and making measurements.

7

Using the Creo Elements/Direct Drafting Copilot Functionality

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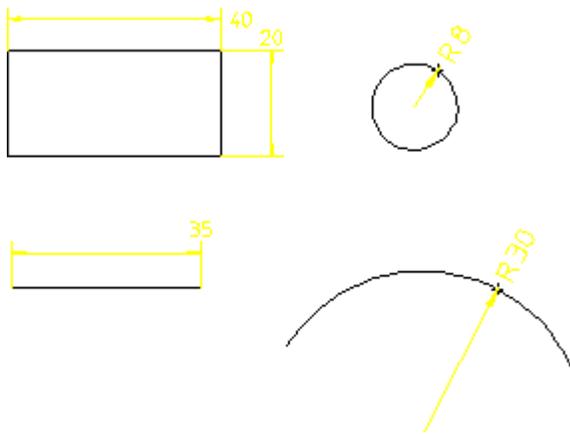
Introduction

The Creo Elements/Direct Drafting copilot functionality provides dynamic visual feedback on an element's attributes (angles and dimensions) before you fix the position. This dynamic feedback lets you create designs quickly and minimizes the need for construction geometry.

You can also enter values using the keyboard as usual if you need greater precision in size and positioning.

The following figure shows an example of the on-screen feedback provided by the copilot functionality when you create geometry.

Figure 20. Copilot Functionality: Typical Feedback



The snap marker (magenta box) over the cursor indicates that you are in copilot mode and about to click a start-point.

Enabling or Disabling the Copilot Functionality

To enable or disable the copilot functionality:

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. Click **Copilot**.
3. Click or clear the **Enable Copilot** check box to enable or disable the copilot, respectively.
4. Click **OK**.

Copilot Settings

You can also define the following settings for the copilot using the **Copilot** pane in the **System** dialog box:

- Snap settings
- Catch range settings
- Text settings

Snap Settings

In the **Snap** area, define the following snap settings:

Angle	In the Angle box, type the angle that the cursor snaps to.
Distance	In the Distance box, type the distance that the cursor snaps to.

For example, if the snap angle is set to 15° and the snap distance is set to 1 mm, specifying lengths is restricted to multiples of 1 mm and angles to multiples of 15°. This means that movement of the cursor over empty areas occurs in increments rather than as a smooth action.

Catch Range Settings

In the **Catch Range** area, define the catch distance:

Perpendicular	Type the catch distance from perpendicular elements.
Tangent	Type the catch distance from tangent elements.
Center	Type the catch distance from center points.

Text Settings

In the **Text Box** area, click the **Long Prefixes** check box to display a full description of the circular element in the text box. For example:

- **Long Prefixes** on = Radius
- **Long Prefixes** off = R

Defining other settings related to Copilot

To define other settings related to copilot:

1. Click **File** ► **Settings** ►  **System**.

-
- The **System** dialog box opens.
2. Click **Snapping**.
 3. Define the following settings for snapping functionality:
 - Snap points
 - Snap marker
 - Snap line
 - Constraint signs

Snap Points Settings

In the **Snap Points** area, define the following:

- | | |
|--------------|---|
| Count | In the Count box, type the count value.
This value sets the maximum number of points simultaneously present in the drawing. |
| Delay | In the Delay box, type the delay (in seconds).
This value sets the delay for how long a cursor is caught on an element before a snap point for a snap line appears. |
| Color | In the Color box, select a color for the last visited snap point. |

Snap marker Settings

In the **Snap marker** area, select a color for the snap marker.

Snap Line Settings

In the **Snap Line** area, define the following

- **Color**
 - **Line type**
- | | |
|-----------------|--|
| Color | In the Color box, select a color for the snap line. |
| Linetype | In the Line type box, select a line type for the snap line. |

Constraint Signs Settings

In the **Constraint Signs** area, define the following:

-
- Enable** Click the **Enable** check box to enable constraint signs.
A constraint sign is displayed when the element you are creating is horizontal, vertical, tangent, or perpendicular to another element.
- Color** In the **Color** box, select a color for the constraint sign.

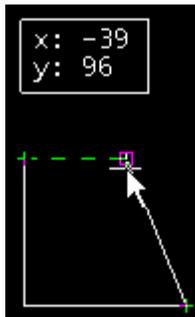
Copilot Features

To improve your speed and accuracy when working with geometry, use the following features:

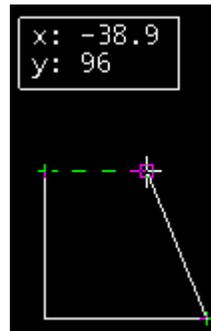
- Incremental movement
- Size indication
- Snap lines

Incremental movement

You can specify a length increment to restrict the length of a line or the radius of a circle using the copilot. The length and angle increments are defined by the snap angle and the snap distance settings available in the **Snap** area of the **Copilot** pane in the **System** dialog box.



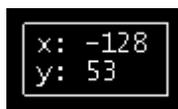
Increment = 1 mm



Increment = 0.1 mm

Size indication

When you create or modify elements, a text box with coordinates shows the element's size or modification distance.



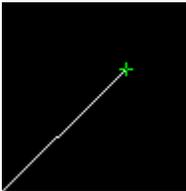
Snap lines

Snap lines are similar to temporary auxiliary lines; they appear when you move the cursor close enough to catch them.



Snap line base points

When you hold the cursor at a vertex for 0.5 seconds (the default setting), a small plus sign appears. This is the snap point for snap lines.



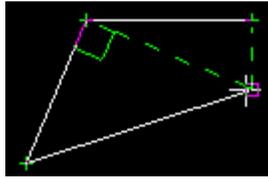
On geometry elements, you can have snap points at the start point, the end point, the midpoint (of lines), and the intersection between two elements.



There is a limit to the number of snap points that can exist at one time; when it is reached, the snap point that hasn't been visited for the longest time disappears. You can change this number in the **Snapping** pane of the **System** dialog box.

Snap line orientation

Snap lines may be horizontal, vertical, parallel, tangent, or perpendicular to an element. You can catch the cursor to a snap line, to the intersection of two snap lines, to the intersection of a snap line and a geometry element.

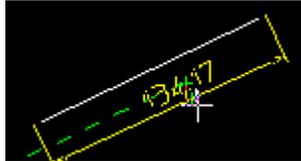


Snap lines on text

The snap point for text is on the attachment point of the text.

Snap lines on dimensions

The snap line snap points for dimensions appear on the center of the dimension text, and the snap line appears on and is parallel to the dimension line.



To align a new dimension with an existing dimension, the text and lines of the two dimensions must be parallel.

Snap lines on reference lines

On reference lines, the snap points appear on the vertices. You can snap the cursor to the reline on any of the 9 text adjust points.

Example — Creating a Slotted Plate

Suppose you want to create an engineering drawing from a sketch made at a design meeting. With the copilot functionality, you can quickly turn a sketch in an engineering design without the need for unnecessary construction geometry.

Figure 21. Design Sketch

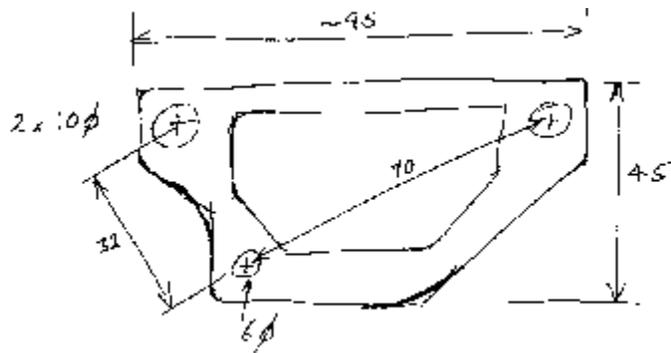
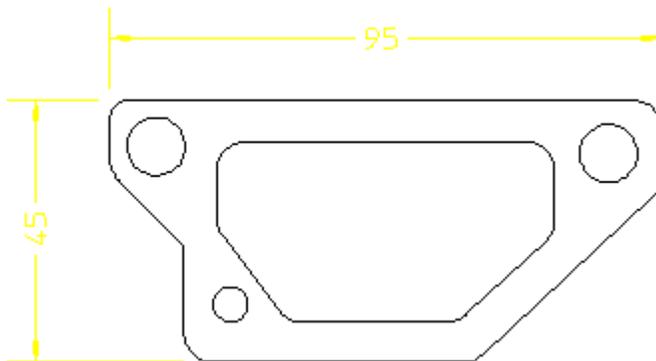


Figure 22. Design Reality



An example of constructing the plate with the minimum of construction geometry follows. As you draw, you will notice that line colors may change as you intersect with existing geometry and the status line gives feedback on your current activity. During certain actions, Creo Elements/Direct Drafting displays negative values. Ignore these for this example.

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
2. Click  **Horizontal** and draw a horizontal line of length 50 mm (line 1 in [Figure 23 on page 148](#)).
3. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
4. Click  **Vertical** and draw a vertical line of length 20 mm (line 2).
5. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
6. Click  **Angle & Length** and click point A.
7. Move the cursor in an arc and specify an angle of 135°.
8. Drag the cursor and specify a length of 18 mm.

9. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **2 Points**.
10. Click  **Parallel** and then click line 1.
11. Drag the cursor vertically and click a point at 45.
12. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
13. Click  **Vertical** and draw line 4 to meet the construction line.
14. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
15. Click  **Horizontal** and draw a horizontal line of length 95 mm.
16. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
17. Click  **Vertical** and draw a vertical line of length 15 (line 6).
18. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
19. Click  **2 Points** and draw line 7.

The next step is to construct the holes. The hole at H1 is to be offset at 10 and 8 with diameter 6 mm. Holes H2 and H3 constructed on arcs 32 and 70 mm from H1.

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Circle**.
2. Click  **Center & Radius** (see [Figure 23 on page 148](#)).
3. Type `OFFSET` or `offset` in the user input line and press ENTER.
4. Click line 1 and then line 2.
5. Click the center point at 8 (x-axis) and 10 (y-axis).
6. Drag the cursor and specify a radius of 3.
7. In the **Draw** group, click the arrow next to  **Center & Radius**.
8. Click  **Center & Radius** and draw a construction circle at 32 with H1 as the center.
9. Similarly, draw a construction circle at 70 with H1 as the center (see [Figure 24. Slotted Plate - 2 on page 148](#)).
10. In the **Draw** group, click the arrow next to  **Circle**.

11. Click  **Center & Radius**.
12. Draw hole H2 having radius 5.
13. Similarly, draw hole H3 also having radius 5.

 **Note**

The precise position for holes H2 and H3 is not important.

14. Delete the circular construction geometry.

Figure 23. Slotted Plate - 1

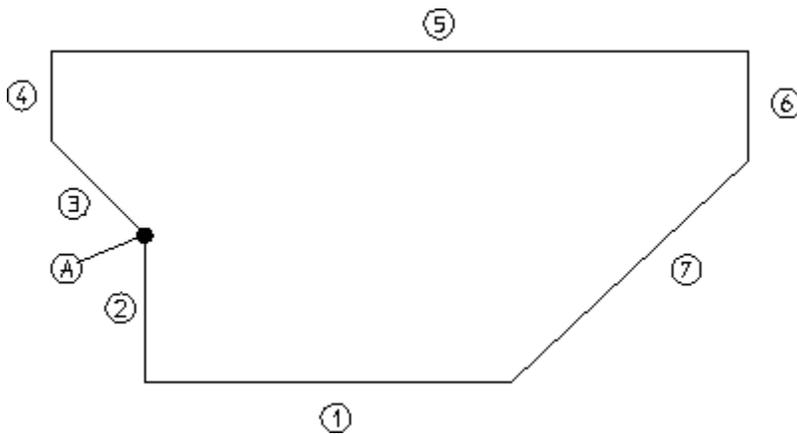
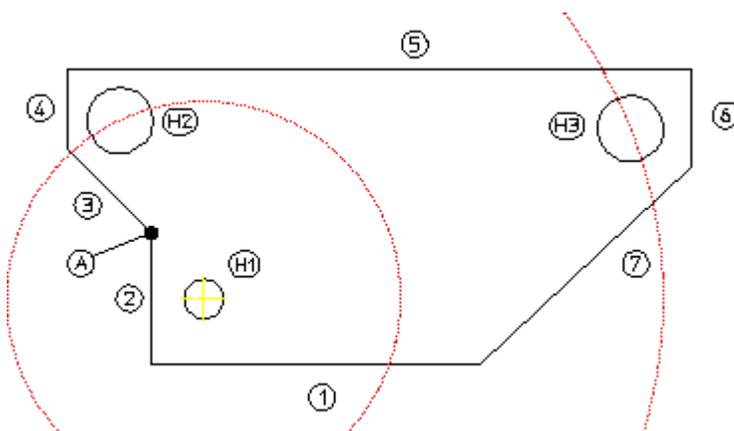


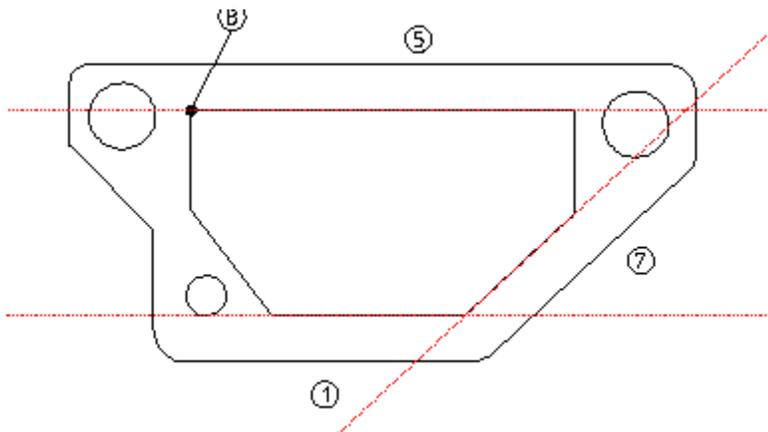
Figure 24. Slotted Plate - 2



The slot can be drawn in freehand mode using  **Polygon**. The only criteria is that the inside edge is parallel to line 7.

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **2 Points**.
2. Click  **Parallel** and draw parallel construction lines at a distance of 7 from lines 1, 5, and 7.
3. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
4. Click  **Polygon** and draw the slot starting at point B moving anticlockwise.
5. Delete the construction geometry to clean up the drawing.
6. Click **Drafting** and then, in the **Engineering** group, click the arrow next to  **Fillet**.
7. Click  **Radius**, type 5 in the user input line, and press ENTER.
8. Click the corners of the plate to add fillets.

Figure 25. Slotted Plate - 3



Further Information

For further technical information about the copilot, refer to the Creo Elements/Direct Drafting help system.

Type `help copilot` in the Creo Elements/Direct Drafting user input line and press ENTER.

Creo Elements/Direct Drafting displays further information about the copilot.

8

Drawing Methods

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This chapter describes how to create a drawing using construction geometry and "real" geometry.

Drawing Geometry

On the **Drafting** tab, the following groups contain the buttons to draw geometry:

- **Draw**
- **Engineering**
- **Annotate**

Drawing Lines

Creo Elements/Direct Drafting allows you to draw linear elements in your drawing using  **Line**.

To draw a linear element:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
2. Click the required **Line** button and perform the associated procedure.

Line Button	Purpose	Procedure
 Polygon	Draw a polygon.	Specify the start point, and then a sequence of end points. The result does not have to be a closed polygon. Creo Elements Direct/Drafting treats the lines as individual elements and hence you must delete them individually or enclose them in a box for deletion. Once you close a polygon, you can continue by indicating the first point of a new polygon.
 2 Points	Draw a line with two points.	Specify the start and end points of the line.
 Horizontal	Draw a horizontal line.	<ol style="list-style-type: none"> a. Specify the start point of the line. b. Type the length of the line in the user

Line Button	Purpose	Procedure
		<p>input line and press ENTER.</p> <p>or</p> <p>Specify the end point of the line.</p> <p>If you specify an end point, the line is drawn through that point.</p> <p>This operation is similar to line by two points but only draws a horizontal line.</p>
 Vertical	<p>Draw a vertical line.</p>	<ol style="list-style-type: none"> a. Specify the start point of the line. b. Type the length of the line in the user input line and press ENTER. <p>or</p> <p>Specify the end point of the line.</p> <p>If you specify an end point, the line is drawn through that point.</p> <p>This operation is similar to line by two points but only draws a vertical line.</p>
 Parallel	<p>Draw a line parallel to an existing line.</p>	<ol style="list-style-type: none"> a. Click an existing line. b. Specify a point on the parallel line. <p>Alternatively:</p> <ol style="list-style-type: none"> a. Type the distance in

Line Button	Purpose	Procedure
		<p>the user input line and press ENTER.</p> <ol style="list-style-type: none"> b. Click an existing line. c. Specify the side on which you want to draw the parallel line. <p>You can use the ruler to ensure that the line is drawn at a given distance (see Using the Ruler or Grid on page 115).</p>
 Perpendicular	<p>Draw a perpendicular line to an existing line.</p>	<ol style="list-style-type: none"> a. Click the existing line. b. Click a point on the perpendicular line. <p>Alternatively, to use a point on the existing line as the starting point of the perpendicular line:</p> <ol style="list-style-type: none"> i. Specify the point on the existing line. ii. Specify the end point of the perpendicular line or type the length of the perpendicular line in the user input line and press ENTER. <p>If the perpendicular line appears on</p>

Line Button	Purpose	Procedure
		<p>the wrong side of the line, repeat the operation using the opposite sign for the specified value.</p>
 Angle & Length	<p>Draw a line with a given length and angle.</p>	<ol style="list-style-type: none"> a. Specify the start point of the line. b. Type the angle of the line with respect to the x-axis in the user input line and press ENTER. c. Type the length of the line in the user input line and press ENTER. <p>The line is drawn accordingly.</p>

Line Button	Purpose	Procedure
 Tangent	Draw a tangent to a circular element (arc or circle) from any given point.	a. Specify the approximate point of tangency on the arc or circle. b. Specify the point from which the tangent should be drawn.  Note Creo Elements/ Direct Drafting informs you if such a tangent is not possible.
 Tangent by 2	Draw a tangent to two arcs or circles (from one circular element to another).	Specify the approximate point of tangency on each of the two arcs or circles. The tangent drawn is the one with points of tangency closest to those indicated.

Drawing Rectangles

You can draw rectangles as whole entities. However, Creo Elements/Direct Drafting treats them as four individual line elements. The sides of a rectangle can be deleted or manipulated individually.

To draw a rectangle:

1. Click **Drafting** and then, in the **Draw** group, click  **Rectangle**.
2. Specify two diagonal corner points.

The rectangle size varies with the movement of the cursor until the second point is defined.

Drawing Circles

Creo Elements/Direct Drafting allows you to draw circular elements in your drawing using  **Circle**.

To draw a circular element:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Circle**.
2. Click the required **Circle** button and perform the associated procedure.

Circle Button	Purpose	Procedure
 Center & Radius	Draw a circle by indicating the center and circumference point.	<ol style="list-style-type: none"> a. Specify the center of the circle. b. Specify a point on the circumference or Type the length of the radius in the user input line and press ENTER. The circle moves with the cursor till you define the circumference point. <p>You can draw additional concentric circles by indicating additional points or radii.</p>
 3 Points	Draw a circle by indicating three points on the circumference.	<p>Specify three circumference points.</p> <p>The circle drawn varies with the movement of the cursor till you define the third point.</p>
 Concentric	Draw concentric circles.	<ol style="list-style-type: none"> a. Click the existing circle. b. Specify the desired circumference point. or Type the radial offset

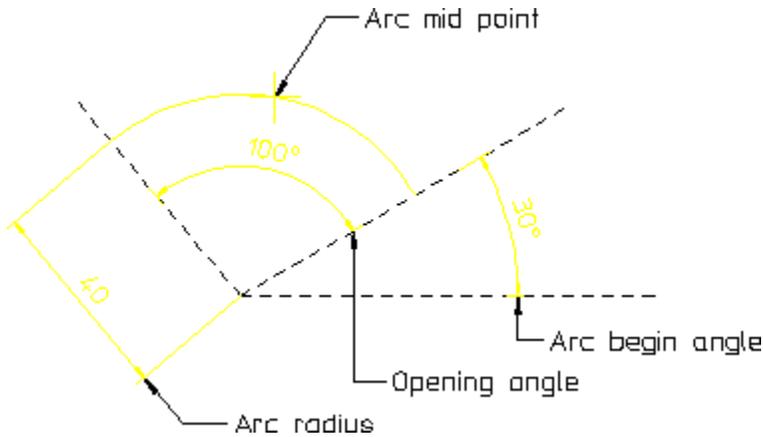
Circle Button	Purpose	Procedure
		<p>in the user input line and press ENTER.</p> <p>To draw further concentric circles, simply specify further circumference points.</p>
 Diameter	Draw a circle by diameter.	Specify the two diameter end points for the circle.
 Tangent	Draw a circle of given radius tangential to two existing elements.	<p>a. Specify the approximate tangent points on the two elements.</p> <p>The elements can be lines, arcs, or circles.</p> <p>b. Type the required radius in the user input line and press ENTER.</p> <p>Creo Elements/Direct Drafting calculates the best solution, or suggests a valid radius, if no solution is possible.</p>

Drawing Arcs

Creo Elements/Direct Drafting allows you to draw arcs in your drawing using  **Arc**. Arcs are normally drawn in the counterclockwise direction.

Figure 26 on page 158 shows arc geometry.

Figure 26. Arc Geometry



To draw an arc:

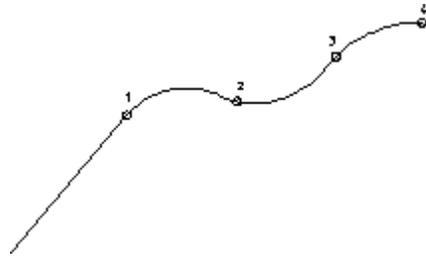
1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Arc**.
2. Click the required **Arc** button and perform the associated procedure.

Arc Button	Purpose	Procedure
 3 Points	Draw an arc through three points.	<ol style="list-style-type: none"> Specify the two end points of the required arc. Specify a third point on the arc. The arc drawn varies with the movement of the cursor till you define the third point.
 Center & 2 Points	Draw an arc with a given center and radius.	<ol style="list-style-type: none"> Specify the center point. Type the arc radius in the user input line and press ENTER. Type the start angle of the arc in the user input line and press ENTER. Type the end angle of the arc in the user

Arc Button	Purpose	Procedure
		<p>input line and press ENTER.</p> <p>Alternatively, you can define the angles by specifying two points.</p>
 Center, Radius & 2 Points	<p>Draw an arc by center and end points.</p>	<ol style="list-style-type: none"> Specify the center of the arc. Specify the arc start point (radius). Specify the arc end point (third point). <p>The third point may not lie on the arc plane, but its line defines the end of the arc.</p>
 Concentric	<p>Draw concentric arcs.</p>	<ol style="list-style-type: none"> Click an existing arc. Specify a point on the required arc. Specify further points to draw further concentric arcs.
 Point, Radius & Angle	<p>Draw an arc by specifying a point, radius, and angle.</p>	<ol style="list-style-type: none"> Specify a peripheral arc point. Type the arc radius in the user input line and press ENTER. Type the start or end angle of the arc in the user input line and press ENTER.

Arc Button	Purpose	Procedure
		<p> Note</p> <p>Angles are measured counterclockwise from the horizontal.</p> <p>d. Type the opening arc angle in the user input line and press ENTER.</p>
 Diameter	<p>Draw an arc by specifying diameter.</p>	<p>Specify the diameter end points. An arc is drawn between the end points.</p> <p>The resulting position of the arc depends upon which diameter point you specify first.</p>
 Smooth	<p>Draw a smoothly connecting arc.</p>	<p>a. Click an element end point (line, circle, and so on).</p> <p>b. Specify the end point of the arc.</p> <p>The arc is drawn tangential to the element you have clicked.</p> <p>With each further point indicated, another arc is drawn tangential to the previous one.</p> <p>In Figure 27 on page 161, points 1 to 4 have been specified in sequence.</p>

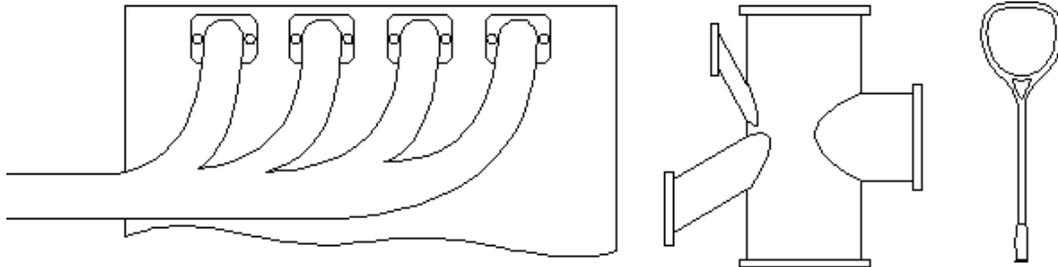
Figure 27. Smoothly Connecting Arcs



Drawing Splines

Spline curves are special types of smooth curves that pass through given data points. When you have specified the data points, Creo Elements/Direct Drafting uses a powerful technique (B-spline mathematics) to produce the spline shape. The following diagram shows examples of uses of spline curves.

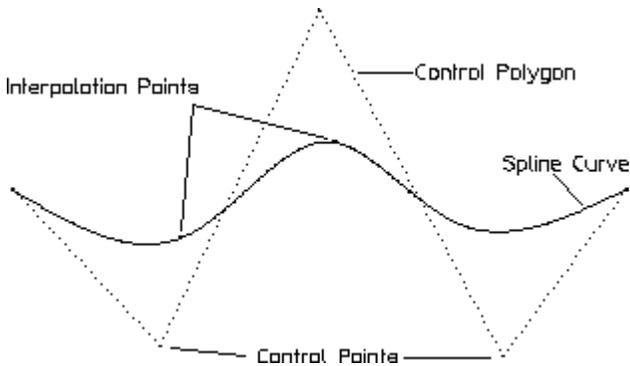
Figure 28. Typical Spline Curves: Engine Manifold, Pipe Tree, Racquet Head



Spline Components

Figure 51 on page 162 shows a simple spline with its control polygon visible. The control polygon can be thought of as a convex hull containing the spline. Note that both interpolation and control splines have a control polygon. Points on the interpolation spline curve are called interpolation points. Points on the control polygon are called control points.

Figure 29. Spline Components



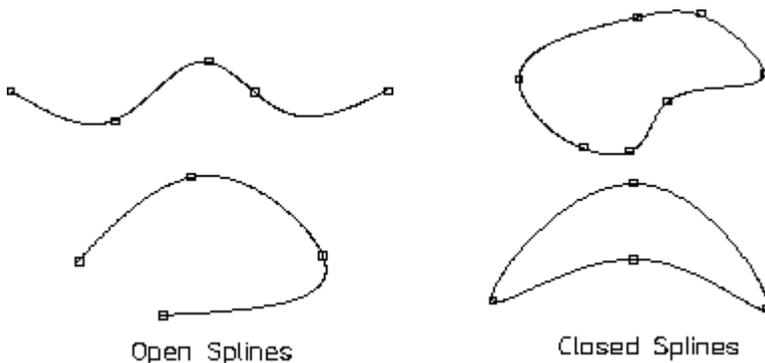
Interpolation Spline vs. Control Spline

You can create splines based on interpolation points or control points. If you want a spline to pass through defined points, use interpolation points. If the spline does not need to pass through defined points, you can use interpolation-point created splines or control-point created splines. Control-point created splines are more appropriate for creative design work where curved molded surfaces are required.

Open Spline vs. Closed Spline

By default, the interpolation or control splines you create in Creo Elements/Direct Drafting are open. The **Closed** check box under  **Spline** enables you to create closed splines.

Figure 30. Open Spline vs. Closed Spline



Drawing a Spline

Creo Elements/Direct Drafting allows you to draw splines in your drawing using **Spline**.

To draw a spline:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Spline**.
2. Click the required **Spline** button and perform the associated procedure.

Spline Button	Purpose	Procedure
 Interpolation	<p>Draw an interpolation spline using interpolation points.</p> <p>Points on an interpolation spline (I spline) are called interpolation points. A spline created with interpolation points passes exactly through the interpolation points used to define it.</p>	<ol style="list-style-type: none"> Specify a sequence of interpolation points. Click  OK. <p>While creating an interpolation spline, you can also change the order of spline, append the spline to an existing geometry, and specify the slope of an interpolation spline. See Modifying Splines on page 223 for more information.</p>
 Control	<p>Draw a control spline.</p> <p>When you draw a spline using a control polygon, the shape of the spline is influenced by the control polygon formed by the control points.</p>	<ol style="list-style-type: none"> Specify a sequence of control points. <p> Note</p> <p>These points make up a control polygon which "controls" the shape and size of the resulting control spline. The first control point serves as the begin point of the control spline. The last control point serves as the end point of the control spline.</p> <ol style="list-style-type: none"> Click  OK. <p>Creo Elements/Direct</p>

Spline Button	Purpose	Procedure
		<p>Drafting draws a spline curve based on the control points you specified.</p> <p>While creating a control spline, you can also change the order of the spline. See Modifying Splines on page 223 for more information.</p>

 **Note**

- To display the control polygon of an interpolation spline, click **File** ► **Settings** ► **Show** and then click the **Vertex** check box.
- To display or hide the spline control points, click **Drafting** and then, in the **Draw** group, click the arrow next to  **Spline**. Click  **Display On** or  **Display Off** to display or hide the spline control points, respectively, and then select the spline.

Drawing Ellipses

Creo Elements/Direct Drafting allows you to draw ellipses in your drawing using the **Ellipse** buttons under  **Spline** in the **Draw** group on the **Drafting** tab.

To draw an ellipse:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Spline**.
2. Click the required **Ellipse** button and perform the associated procedure.

Ellipse Button	Purpose	Procedure
 Axis & Point	Draw an ellipse by axis end points and a peripheral point.	<ol style="list-style-type: none"> a. Specify the two vertices of the major axis of the required ellipse. b. Specify a point on the circumference.
 Center, Angle & Radius	Draw an ellipse by center, angle and radii.	<ol style="list-style-type: none"> a. Specify the center of the required ellipse. b. Type the angle of the major axis with respect to the x axis (30° in Figure 31 on page 150) in the user input line and press ENTER. c. Type the major radius (30 mm in Figure 31 on page 167) in the user input line and press ENTER. d. Type the minor radius (15 mm in Figure 31 on page 167) in the user input line and press ENTER.
 Center & 2 Points	Draw an ellipse using the center and two points.	<ol style="list-style-type: none"> a. Specify the center point of the ellipse. b. Specify the first point on the circumference. c. Specify the second point on the circumference.

Ellipse Button	Purpose	Procedure
 2 Focal & Circumference	Draw an ellipse by specifying the focus. See Figure 32 on page 168 (drawing A).	<ol style="list-style-type: none"> Specify the first focal point (1). Specify the second focal point (2). Specify a point on the circumference (3).
 Focal, Angle & Radius	Draw an ellipse by specifying the focal points and a direction. See Figure 32 on page 168 (drawing B).	<ol style="list-style-type: none"> Specify the first focal point (point 1). Specify the direction of the major axis (point 2) Type a numerical eccentricity value for the ellipse (for example, 0.8) in the user input line and press ENTER. <p> Note The eccentricity of ellipse is greater than 0 but less than 1.</p> Type a value for the major radius (for example, 15) in the user input line and press ENTER.

Figure 31. Ellipse by Center, Angle and Radii

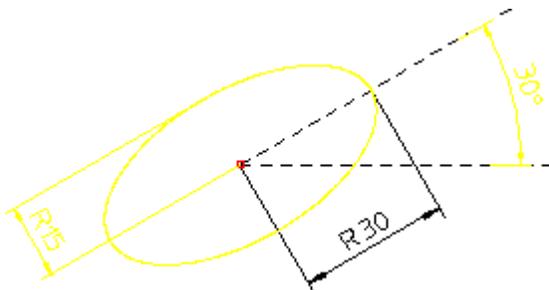
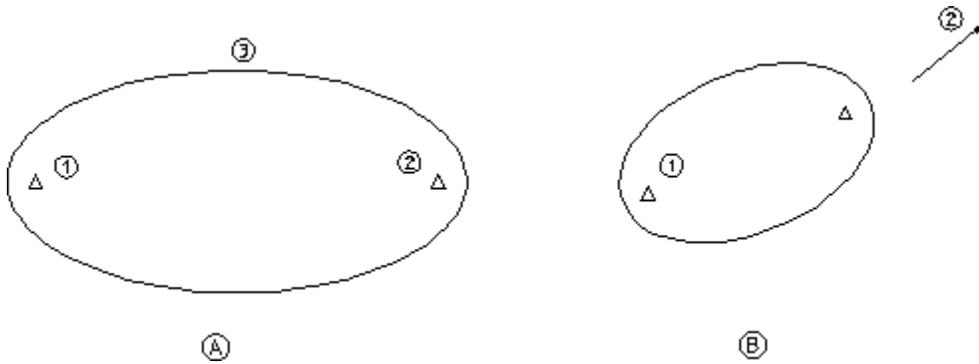


Figure 32. Ellipses by Focus, Point, Angle



Drawing Hyperbolas

Creo Elements/Direct Drafting allows you to draw hyperbolas in your drawing using the **Hyperbola** buttons under  **Spline** in the **Draw** group on the **Drafting** tab.

To draw a hyperbola:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Spline**.
2. Click the required **Hyperbola** button and perform the associated procedure:

Hyperbola Button	Purpose	Procedure
 Center, Focus & Point	Draw a hyperbola using center, focus, and point.	<ol style="list-style-type: none"> Specify the center of the hyperbola (point 1). Specify the focus of the hyperbola (point 2). Specify one end of the hyperbola (point 3). <p>The other end point of the hyperbola is computed by symmetry.</p> <p>See Figure 33 on page 169.</p>
 Center, Tangent & Focus	Draw a hyperbola tangential to existing geometry.	<ol style="list-style-type: none"> Specify the center of the hyperbola (point 1). Click a tangential point (point 2). Specify the focus of the hyperbola (point 3). <p>See Figure 34 on page 170.</p>

Figure 33. Hyperbola by Center, Focus, and End Point

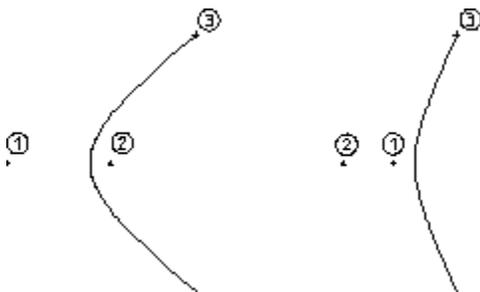
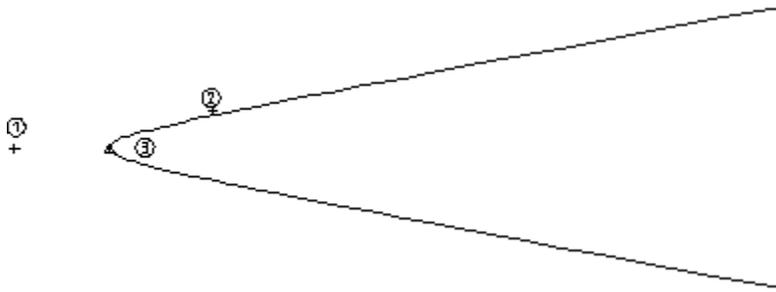


Figure 34. Hyperbola by Center, Tangent, and Focus



Drawing Parabolas

Creo Elements/Direct Drafting allows you to draw parabolas in your drawing using the **Parabola** buttons under  **Spline** in the **Draw** group on the **Drafting** tab.

To draw a parabola:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Spline**.
2. Click the required **Parabola** button and perform the associated procedure:

Parabola Button	Purpose	Procedure
 Center & Focus	Draw a parabola using its center and focus.	a. Specify the center of the parabola (point 1) b. Specify the focus of the parabola (point 2) See Figure 35 on page 171 .
 Center, Direction & Point	Draw a parabola in specific direction.	a. Specify the focus of the parabola (point 1). b. Specify the direction of the parabola (point 2). c. Specify the end of the parabola (point 3). Note that the other end point of the parabola is computed by symmetry. See Figure 36 on page 172 .

Figure 35. Parabola by Center and Focus

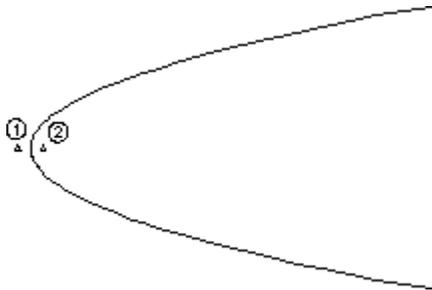
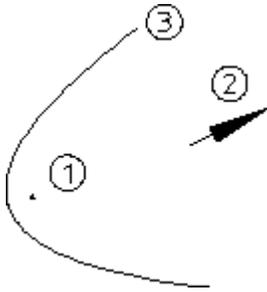


Figure 36. Parabola by Focus, Direction and End Points



Drawing Points

Creo Elements/Direct Drafting allows you to draw points anywhere on your drawing.

To draw a point:

1. Click **Drafting** and then, in the **Draw** group, click  **Point**.
2. Click the position in a viewport.

or

Type the required coordinates in the user input line and press ENTER.

If you have difficulty deleting points, try enclosing them in a box for deletion.

Drawing Polylines

Creo Elements/Direct Drafting allows you to draw polylines from polygons using



Polyelement.

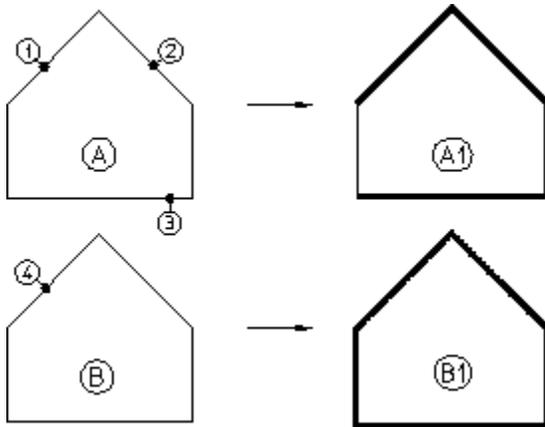
Polylines facilitate design modifications.

The following example shows some of the differences of working with polyline elements and non-polyline elements. Drawing A is a standard polygon created by

clicking  **Polygon** under  **Line** in the **Draw** group on the **Drafting** tab. To modify the pensizes of drawing A to produce drawing A1, you need to click points 1, 2, and 3 (for the purpose of this example, it is assumed that only individual elements can be clicked).

Drawing B is similar to drawing A but has been converted into a polyline. Because drawing B is a polyline, it acts like a single element and you need only click point 4 to modify the complete shape (drawing B1).

Figure 37. Modifying Polyline and Non-Polyline Elements



The following procedure describes how to draw a polyline from a polygon and then modify the polyline by changing its pensize and shape.

To draw a polyline:

1. Draw a polygon.
2. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Polyelement**.
3. Click  **Create**.
4. Click a vertex of the existing polygon.
5. Click a polygon element that ends at the vertex.
6. Click adjacent elements or click the last element.
7. Click  **OK** when the desired shape is defined.

The polygon is now a polyline. To test the polyline, change the pensize.

To change the pensize of the polyline:

1. Select the polyline and right-click.
2. Click  **Change Geo Penwidth**.
3. Type a value (for example, 0.5) in the user input line and press ENTER.
4. Select one element of the polyline.

Note that the complete shape changes except the selected element.

Similarly, you can modify the shape of the polyline.

To change the shape of the polyline:

1. Click **Modify** and then, in the **General** group, clear the **Keep** check box.

2. In the **General** group, click **More**.
3. Click  **Stretch**.
4. Click  **Free**,  **Horizontal**, or  **Vertical**.
5. Click a vertex (or enclose it in a box).
6. Specify the new position of the vertex.

Smashing a Polyline

You can smash the polyline to individual elements using  **Smash**.

To smash the polyline:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Polyelement**.
2. Click  **Smash**.
3. Select the polyelement.
4. Click  **OK**.

Drawing Equidistant (Parallel) Contours

To draw a contour equidistant to an existing contour:

1. Click **Drafting** and then, in the **Draw** group, click  **Equidistance**.
2. Type the separation distance required for the equidistant or parallel contour in the user input line and press ENTER.
3. Click the start point of the existing contour.
4. Select the elements to be copied (Creo Elements/Direct Drafting allows you to skip to the final element, provided you do not pass more than one branch element that is not to be copied).
5. Specify the side on which the equidistant contour is to be drawn.

If you require a particular section of a contour to be copied:

1. Follow steps 1 to 4 above.
2. When you have indicated the desired section, click  **OK**.
3. Specify the side on which the equidistant contour is to be drawn.

Using Overdraw

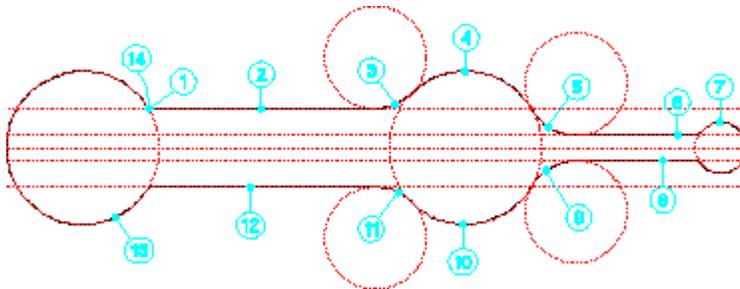
Creo Elements/Direct Drafting allows you to draw a contour between intersection points in the construction geometry using  **Overdraw**. The lines are drawn over the existing construction lines and circles just as you would line in a contour by hand.

Additionally, it is not necessary to click each construction geometry intersection point before clicking the line or arc you want to draw over. You simply click the next construction geometry element and Creo Elements/Direct Drafting calculates the intersection point. This saves one click for every overdraw action.

To use  **Overdraw**:

1. Draw the construction geometry as shown in [Figure 37 on page 175](#).
2. Click **Drafting** and then, in the **Draw** group, click  **Overdraw**.
3. Specify the start point (1).
4. Specify points 2 to 14 (end point).

Figure 38. Using the Overdraw Command



 **Overdraw** supports multiple  **Undo** operations within the command.

Drawing Construction Geometry

Construction geometry is used for positioning and as an aid to drawing.

This section describes how to draw construction lines and circles, just as you would normally do when working on a drawing board.

Unlike real lines, construction lines are considered to be of infinite length. Only full construction circles can be drawn.

When drawing construction lines, you can indicate a point for the line or enter the exact coordinates via the keyboard.

Enter coordinates using the following format:

2.5, 3

12, 4.8

5.35, 6.1

The x and y coordinates are separated by a comma. After you type the coordinates in the user input line, press ENTER.

Drawing Construction Lines

Creo Elements/Direct Drafting allows you to draw construction lines using **Construction Line** buttons in the **Draw** group on the **Drafting** tab.

To draw construction lines:

1. Click **Drafting** and then in the **Draw** group, click the arrow next to  **2 Points**.
2. Click the required **Construction Line** button and perform the associated procedure.

Construction Line Button	Purpose	Procedure
 2 Points	Draw construction lines using two points.	<ol style="list-style-type: none">a. Type the angle for the construction line in the user input line and press ENTER.b. Specify the second point. Alternatively, click the two points for the construction line.
 Horizontal	Draw horizontal construction lines.	<ol style="list-style-type: none">a. Type the y coordinate of the horizontal construction line in the user input line and press ENTER. or Click a point through which the horizontal construction line must pass.

Construction Line Button	Purpose	Procedure
		b. To draw several horizontal construction lines, type their y coordinates, separated by spaces, in the user input line and press ENTER.
 Vertical	Draw vertical construction lines.	a. Type the x coordinate of the vertical construction line in the user input line and press ENTER. or Click a point through which the vertical construction line must pass. b. To draw several vertical construction lines, type their x coordinates, separated by spaces, in the user input line and press ENTER.
 Parallel	Draw construction lines parallel to an existing line.	a. Type the separation distance of the line in the user input line and press ENTER. b. Click any existing line and then click the side on which the parallel construction line should be drawn. Multiple lines can be drawn at the

Construction Line Button	Purpose	Procedure
		<p>previously set separation distance by clicking the original lines and the desired sides.</p> <p>Alternatively:</p> <ol style="list-style-type: none"> Click any existing line. Specify a point for the parallel construction line to pass through. <p>You can draw multiple parallel construction lines by specifying further points for the lines to pass through.</p> <p>Alternatively, draw a set of parallel lines from an original line.</p> <p>For example, if you specify line separations as 10 20 30 40, four parallel lines are drawn at the given distances from the indicated line.</p>
 Perpendicular	Draw a construction line perpendicular to an existing line.	<ol style="list-style-type: none"> Click any existing line. Specify a point through which the perpendicular construction line must pass.
 Point & Angle	Draw a construction line at an angle.	<ol style="list-style-type: none"> Specify a point through which the construction line must pass.

Construction Line Button	Purpose	Procedure
		<p>b. Specify a second point for the construction line.</p> <p>or</p> <p>Type the angle of the line with respect to the x axis in the user input line and press ENTER.</p> <p>A positive angle is measured in a counterclockwise arc from the horizontal. You can set the angle units in your Creo Elements/Direct Drafting environment.</p>
 Tangent	<p>Draw a construction line tangent to a circle or a spline and passing through a given point or at a set angle.</p>	<p>a. Click the approximate point of tangency.</p> <p>b. Type the angle of the construction line in the user input line and press ENTER.</p> <p> Note</p> <p>The angle is measured in a counterclockwise arc from the horizontal.</p> <p>Alternatively, click the approximate point of tangency on the circle or spline and then specify a point through which the construction line must pass.</p>

Construction Line Button	Purpose	Procedure
 Tangent by 2	Draw a construction line as a tangent to two circles or splines.	Click the approximate point of tangency on each of the two circles or splines. The construction line is drawn through the points of tangency that are closest to the points you indicated.
 Bisector	Draw a bisecting construction line.	Specify the two end points of the line. A construction line is drawn perpendicular to the line at the appropriate position. To divide the line in two unequal parts: <ol style="list-style-type: none"> Type the bisector factor (value for the ratio of division) in the user input line and press ENTER. Specify first point on the line. Specify second point on the line.
 Bisector Angle	Draw a construction line bisecting an angle.	<ol style="list-style-type: none"> Specify the first line. Specify the second line. A construction line, defining the bisector angle, is drawn.

Drawing Construction Circles

Creo Elements/Direct Drafting allows you to draw construction circles using **Construction Circle** buttons in the **Draw** group on the **Drafting** tab.

To draw a construction circle:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Center & Radius**.
2. Click the required **Construction Circle** button and perform the associated procedure.

Construction Circle Button	Purpose	Procedure
 Center & Radius	Draw a construction circle by center and circumference point.	<ol style="list-style-type: none"> a. Specify the center of the construction line. b. Type the radius of the construction circle in the user input line and press ENTER. <p>Alternatively, specify a point on the circumference.</p> <p>If multiple concentric circles are required, type the desired radii in the user input line and press ENTER or specify multiple peripheral points.</p>
 3 Points	Draw a construction circle by three points on the circumference.	Specify two circumference points. The construction circle drawn varies with the movement of your cursor till you specify the third point.
 Tangent & Center	Draw a construction circle by a center and element.	<ol style="list-style-type: none"> a. Click the approximate point of tangency on the existing element. b. Specify the center of the construction circle to be drawn.
 Concentric	Draw concentric circles.	<ol style="list-style-type: none"> a. Click the existing circle.

Construction Circle Button	Purpose	Procedure
		<p>b. Type the radial offset of the new construction circle from the existing one, that is, the difference between the two radii, in the user input line and press ENTER.</p> <p>c. To draw several concentric construction circles, type the appropriate radial offsets separated by spaces in the user input line and press ENTER.</p> <p>or</p> <p>Click existing points through which the construction circles must pass.</p>
 Diameter	Draw a construction circle by diameter.	Specify two points as the two ends of the diameter for the required circle.
 Tangent & 2 Points	Draw a construction circle by radius, point (or two points) and element.	<p>a. Click the approximate point of tangency of the construction circle on the existing element (which must not be a spline).</p> <p>b. Specify a point on the circumference.</p> <p>c. Type the required radius in the user input line and press ENTER.</p> <p>or</p> <p>Specify another point.</p>

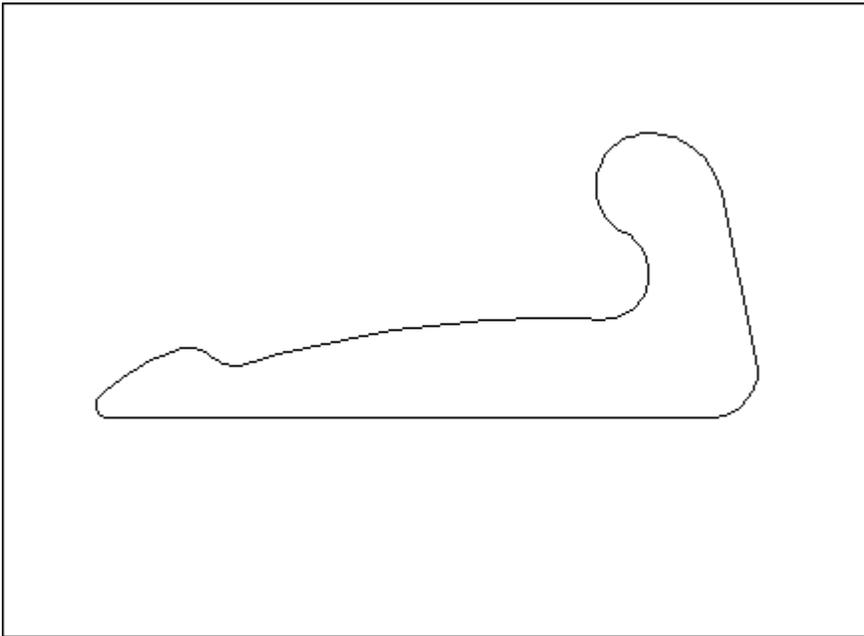
Construction Circle Button	Purpose	Procedure
 Tangent by 2	Draw a construction circle by radius or point and tangential to two elements.	<ol style="list-style-type: none"> a. Click the approximate points of tangency (either lines or circles but no splines). b. Type the desired radius in the user input line and press ENTER. <p>or</p> <p>Specify a peripheral point on the construction circle.</p> <p>Creo Elements/Direct Drafting calculates the best solution, or informs you that no solution is possible.</p>
 Tangent by 3	Draw a construction circle by three elements.	<p>Click the approximate point of tangency of the construction circle on each of the three existing elements (none of which must be a spline).</p> <p> Note</p> <p>If the third tangency point for a construction circle is located such that no circle is tangent to all the three points, the first two points moves along the projection (notional extension) of the associated elements.</p>

Drawing a Lever Using Construction Geometry

A tutorial example to familiarize you with the various techniques for creating a drawing using construction geometry follows.

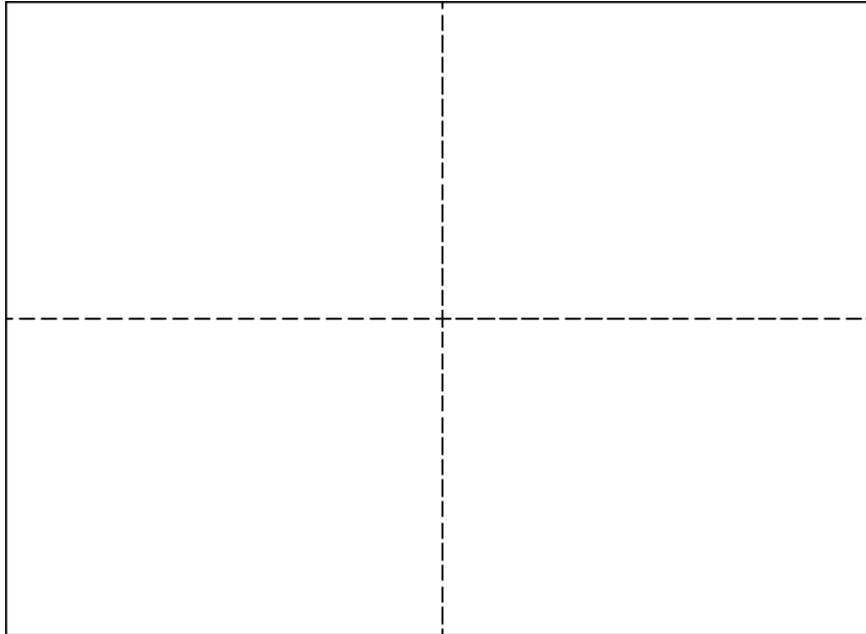
[Figure 39. Lever on page 184](#) shows the finished drawing of the lever.

Figure 39. Lever



1. Draw horizontal and vertical construction lines. See [Figure 40. Horizontal and Vertical Construction Lines on page 185](#).

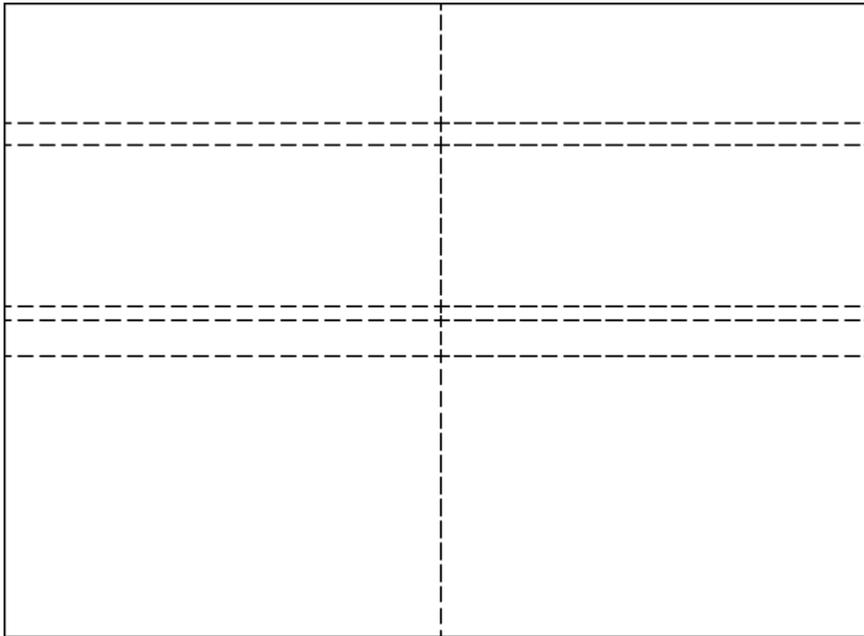
Figure 40. Horizontal and Vertical Construction Lines



- a. In the **Utilities** group, click **Show**.
or
Click **File** ► **Settings** ► **Show**.
The **Show** dialog box opens.
- b. Click the **Construction Geometry** check box and the **Geometry** check box.
- c. Clear the **Vertex** check box.
- d. Click **Close**.
- e. In the Catch box located at the bottom of Creo Elements/Direct Drafting window, select **ALL**.
- f. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **2 Points**.
- g. Click  **Horizontal**.
- h. Type 0, 0 in the user input line and press ENTER.
- i. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Horizontal**.
- j. Click  **Vertical**.
- k. Type 0, 0 in the user input line and press ENTER.

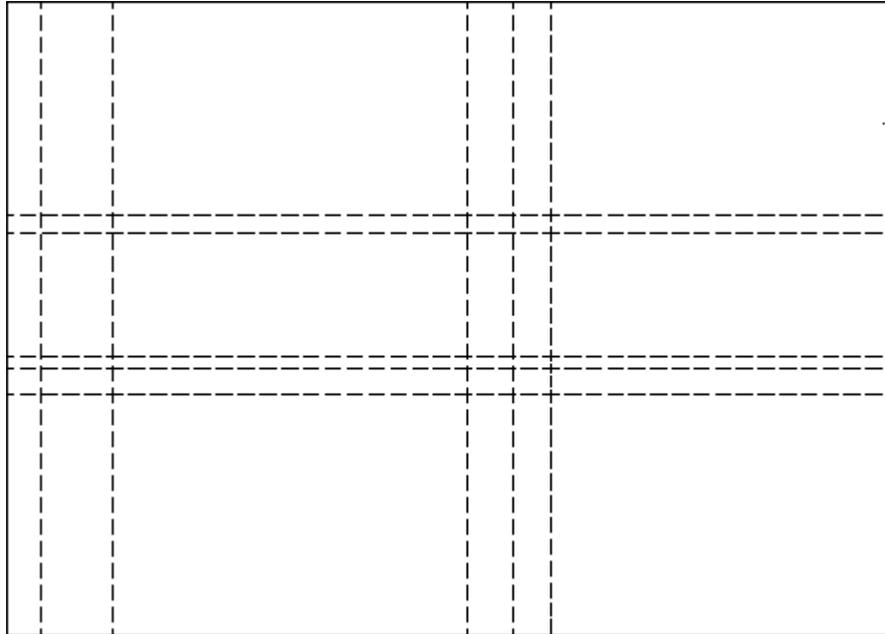
- l. Click **View** and then, in the **Viewing** group, click  **Fit**.
 - m. Click  **OK**.
2. Draw some parallel horizontal construction lines. See [Figure 41. Parallel Construction Lines on page 186](#).

Figure 41. Parallel Construction Lines



- a. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Vertical**.
 - b. Click  **Parallel**.
 - c. Type `-8 3 40 45` in the user input line and press ENTER.
 - d. Select the horizontal construction line and then click somewhere above it.
 - e. Click **View** and then, in the **Viewing** group, click  **Fit**.
3. Draw some parallel vertical construction lines. See [Figure 42. Parallel Vertical Lines on page 187](#).

Figure 42. Parallel Vertical Lines



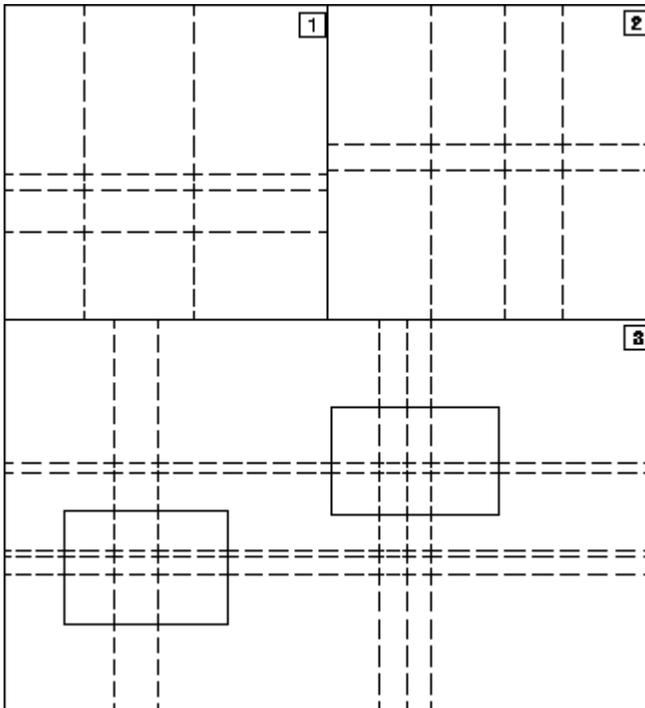
- a. Type 11 25 130 151 in the user input line and press ENTER.

 **Note**

The command for the parallel construction line is still active.

- b. Click the vertical construction line and click a point to the left of it.
 - c. Click **View** and then, in the **Viewing** group, click  **Fit**.
4. Create three viewports in the drawing area. See [Figure 43. Three Viewports on page 188](#).

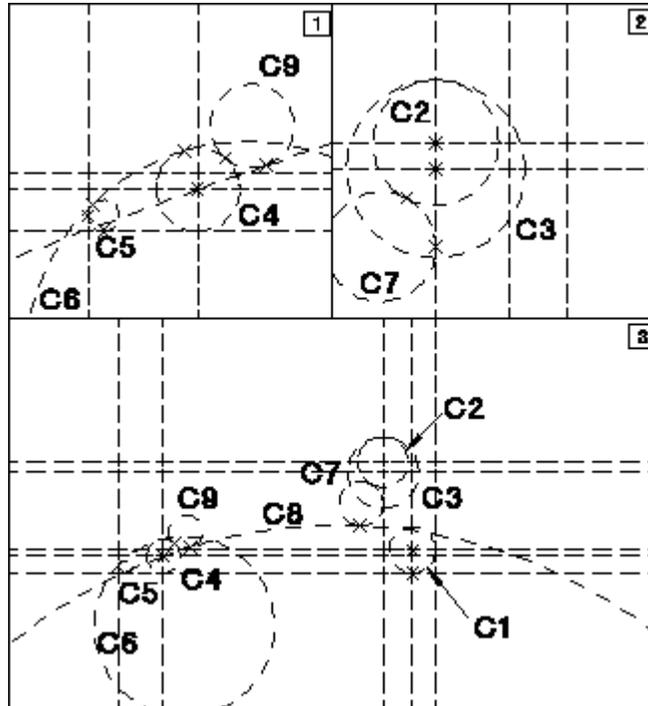
Figure 43. Three Viewports



- a. Click **View** and then, in the **Window** group, click  **Delete**.
- b. Type ALL in the user input line and press ENTER.
- c. Click the viewport to delete it.
- d. Click **View** and then, in the **Window** group, click  **New**.
- e. Create viewport 1 as shown above by clicking two opposite corner points of the viewport.
- f. Repeat the earlier two steps to create viewports 2 and 3.
Viewports 1 and 2 give you a closer look at certain areas of your drawing where you do more complex work. Viewport 3 displays the entire drawing.
- g. Click **View** and then, in the **Window** group, click **Current**.
- h. Click viewport 1.
- i. Click **View** and then, in the **Window** group, click  **New** and click two points in viewport 3 as defined by the rectangle on the left.
- j. Click **View** and then, in the **Window** group, click **Current**.
- k. Click viewport 2.

1. Click **View** and then, in the **Window** group, click  **New** and click two points in viewport 3 as defined by the rectangle on the right.
5. Now create some construction circles. See [Figure 44. Construction Circles](#) on page 189.

Figure 44. Construction Circles

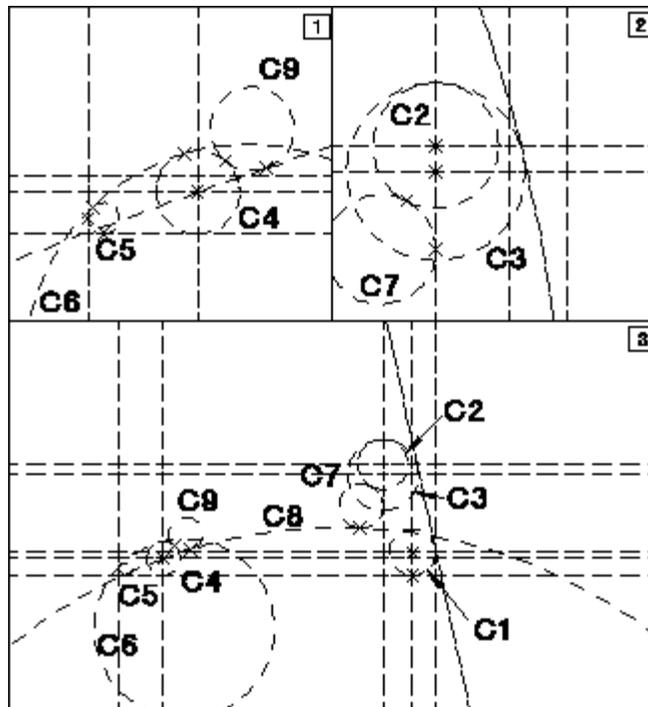


- a. In the Catch box located at the bottom of Creo Elements/Direct Drafting window, select **ALL**.
- b. Draw construction circle C1.
 - i. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Center & Radius**.
 - ii. Click  **Center & Radius**.
 - iii. Click the center point and lower circumference point (in viewport 3).
- c. Draw construction circle C2.
 - i. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Center & Radius**.
 - ii. Click  **Center & Radius**.

-
- iii. Click the center point, type 12 in the user input line, and press ENTER (in viewport 2).
- d. Draw construction circle C3.
- Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Center & Radius**.
 - Click  **Center & Radius**.
 - Click the center point, type 17 in the user input line, and press ENTER (in viewport 2).
- e. Draw construction circle C4.
- Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Center & Radius**.
 - Click  **Center & Radius**.
 - Click the center point, type 8 in the user input line, and press ENTER (in viewport 1).
- f. Draw construction circle C5.
- Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Center & Radius**.
 - Click  **Tangent by 2**.
 - Click the two tangent points, type 3 in the user input line, and press ENTER (in viewport 1).
- g. Draw construction circle C6.
- Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Tangent by 2**.
 - Click  **Tangent by 2**.
 - Click the two tangent points, type 43 in the user input line, and press ENTER (in viewport 1).
- h. Draw construction circle C7.
- Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Tangent by 2**.
 - Click  **Tangent by 2**.
 - Click the two tangent points, type 10.5 in the user input line, and press ENTER (in viewport 2).

- i. Draw construction circle C8.
 - i. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Tangent by 2**.
 - ii. Click  **Tangent & 2 Points**.
 - iii. Click the tangent point on C7 and the center point of C4.
 - iv. Type 246 in the user input line and press ENTER (in viewport 3).
 - j. Draw construction circle C9.
 - i. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Tangent & 2 Points**.
 - ii. Click  **Tangent by 2**.
 - iii. Click the two tangent points, type 8 in the user input line, and press ENTER (in viewport 3).
6. Draw a construction line tangent to C1 and C3. See [Figure 45. Construction Line Tangent](#) on page 191.

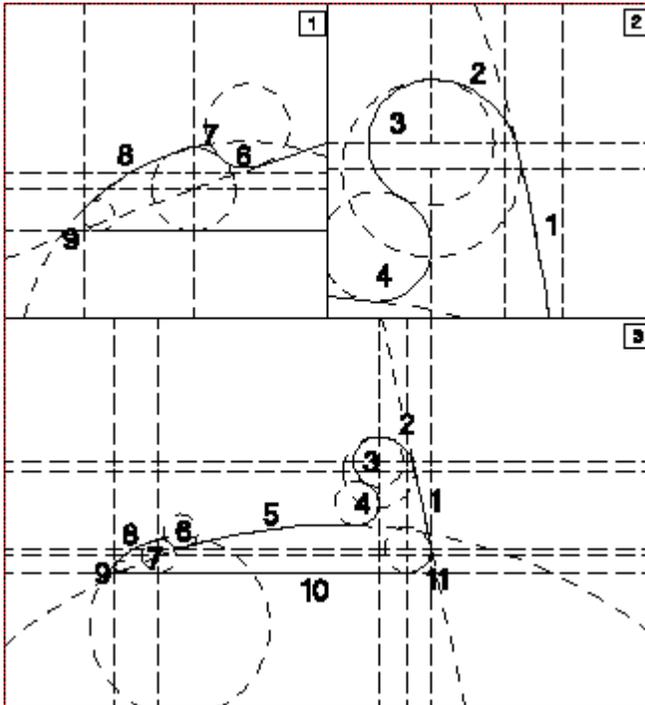
Figure 45. Construction Line Tangent



- a. Click **Drafting** and then in the **Draw** group, click the arrow next to  **2 Points**.

- b. Click  **Tangent by 2**.
 - c. Click the tangent points on C1 and C3 (in viewport 3).
7. Click  **Overdraw** to draw the lever itself. See [Figure 46. Drawing the Lever](#) on page 192.

Figure 46. Drawing the Lever



- a. Click **Drafting** and then, in the **Draw** group, click  **Overdraw**.
- b. Click point 11/1 as the start point.
- c. Click a point on element 1 and click point 1/2.
- d. Click a point on element 2 and click point 2/3.
- e. Click a point on element 3 and click point 3/4.
- f. Click a point on element 4 and click point 4/5.
- g. Click a point on element 5 and click point 5/6.

 **Note**

When drawing the arc on C9, ensure that the cursor is nearer to C9 than to any other line. If you make a mistake, click **Drafting** and then, in the **Draw** group, click  **Overdraw** and zoom-in on the area by clicking  **New** in the **Windows** group on the **View** tab.

- h. Continue the same procedure for elements 6, 7, 8, 9, 10, and 11.

Specifying an Offset

Creo Elements/Direct Drafting allows you to specify an offset distance for the following geometrical elements:

- Lines
- Rectangles
- Circles
- Arcs
- Points
- Construction lines
- Construction circles

You can also specify an offset while modifying the geometrical elements. The offset distance is defined as the distance of the geometrical element from two existing linear elements.

You can use  (Offset) on the status bar to define the offset.  is disabled by default. It is active when you use following buttons:

- **Line** buttons ( **Polygon**,  **2 Points**,  **Horizontal**,  **Vertical**,  **Parallel**,  **Perpendicular**,  **Angle & Length**,  **Tangent**, or  **Tangent by 2**)
-  **Rectangle**
- **Circle** buttons ( **Center & Radius**,  **3 Points**,  **Concentric**,  **Diameter**, or  **Tangent**)

- Arc buttons ( 3 Points,  Center & 2 Points,  Center, Radius & 2 Points,  Concentric,  Point, Radius & Angle,  Diameter, or  Smooth)
 -  Point
 - Construction Line buttons ( 2 Points,  Horizontal,  Vertical,  Parallel,  Perpendicular,  Point & Angle,  Tangent,  Tangent by 2,  Bisector, or  Bisector Angle)
 - Construction Circle buttons ( Center & Radius,  3 Points,  Tangent & Center,  Concentric,  Diameter,  Tangent & 2 Points,  Tangent by 2, or  Tangent by 3)
 - Move buttons ( 2 Pnts,  Horizontal,  Vertical, or  Multiple)
-  is disabled after you finish the above operations.

 **Note**

To use , you must enable **Copilot** in Creo Elements/Direct Drafting. See [Enabling or Disabling the Copilot Functionality on page 140](#) for more information.

To define an offset for geometrical elements in Creo Elements/Direct Drafting:

1. Click the required button (for example,  **2 Points** in the **Draw** group on the **Drafting** tab).
2. In the status bar, click .
3. Select the first offset element.
4. Type the distance from the first offset element in the user input line and press ENTER.
5. Click a point on one side of the first offset element to indicate the positive side.
6. Select the second offset element.

7. Type the distance from the second offset element in the user input line and press ENTER.
8. Click a point on one side of the second offset element to indicate the positive side.

The offset point thus defined serves as the current definition point (such as the begin point or end point) of the geometrical element.

9. Complete the operation.

10. Click  **OK**.

Alternatively:

1. Click the required button (for example,  **2 Points** in the **Draw** group on the **Drafting** tab).
2. In the status bar, click .
3. Select the first offset element.
4. Select the second offset element.
5. Click between the two elements to specify the offset point.
6. Complete the operation.
7. Click  **OK**.

After you click  **OK**, Creo Elements/Direct Drafting ends  and it is no longer active in the status bar.  also ends if you click  **Cancel**.

Drawing Fillets

Creo Elements/Direct Drafting allows you to draw fillets in your drawing using

 **Fillet**.

To draw a fillet:

1. Click **Drafting** and then, in the **Engineering** group, click the arrow next to  **Fillet**.
2. Click  **Radius**.
3. Type the radius in the user input line and press ENTER.
4. Click the two elements between which the fillet should be drawn, or click the vertex.

The fillet radius is drawn and by default the corner point is deleted. If you do not want the corner to be deleted, click the **Keep Corner** check box before clicking the elements or vertex.

Changing Fillets

To change fillet radii:

1. Click **Drafting** and then, in the **Engineering** group, click the arrow next to  **Fillet**.
2. Click  **Modify**.
3. Type the fillet radius in the user input line and press ENTER.

Note

To remove all fillets, type 0 in the user input line and press ENTER.

4. Select the fillet or enclose it in a box.

To select all fillets, click  **Select** ►  **All** in the **Utilities** group. Then click  **Confirm**.

Tip

If fly-by highlighting is enabled, Creo Elements/Direct Drafting highlights the fillets when you hover over them. You can use the CMT and the context menu to modify the general properties of a fillet (for example, to change the color, to change the line type, or to move the fillet) after you select it.

For more information about fly-by highlighting, CMT, and the context menu, see [Fly By Highlighting on page 26](#), [Command Mini Toolbar on page 52](#), and [Context Menu Context Menu Commands on page 55](#).

Drawing Chamfers

Creo Elements/Direct Drafting allows you to draw and modify chamfers in your drawing using  **Chamfer**.

By default, the corner is deleted after the chamfer is drawn. If you do not want the corner to be deleted, click the **Keep Corner** check box in the **Engineering** group on the **Drafting** tab before you choose a  **Chamfer** button.

To draw a chamfer:

1. Click **Drafting** and then, in the **Engineering** group, click the arrow next to  **Chamfer**.
2. Click the required **Chamfer** button and perform the associated procedure.

Chamfer Button	Purpose	Procedure
 Vertex	Draw a chamfer at a given distance from the vertex along each line.	<ol style="list-style-type: none"> a. Type the distance of the chamfer from the vertex in the user input line and press ENTER. b. Click the vertex (the point where the two lines intersect).
 2 Points	Draw a chamfer by two points.	Click the start point and end point of the chamfer.

Chamfer Button	Purpose	Procedure
 Dist. Dist.	Draw a chamfer by specifying two different distances.	<ul style="list-style-type: none"> a. Type the distance to chamfer along the first element in the user input line and press ENTER. b. Type the distance to chamfer along the second element in the user input line and press ENTER. c. Click the first element. d. Click the second element.
 Dist. Ang.	Draw a chamfer at a given angle to the first line and at a given distance from the vertex.	<ul style="list-style-type: none"> a. Type the distance to chamfer along the first element in the user input line and press ENTER. b. Type the angle between the chamfer and the first element in the user input line and press ENTER. c. Click the first element. d. Click the second element.

Changing Chamfers

To change a chamfer:

1. Click **Drafting** and then, in the **Engineering** group, click the arrow next to  **Chamfer**.
2. Click  **Modify**.
3. Type the revised chamfer distance in the user input line and press ENTER.

4. Select the chamfer.

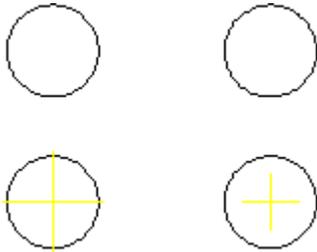
Drawing a Centerline

Creo Elements/Direct Drafting allows you to draw a centerline for a circular element using  **Center Line**.

The centerline becomes part of the circular element and is moved or deleted if the associated circular element is moved or deleted.

[Figure 47 on page 199](#) shows the top circles before and after adding a centerline. The different appearance of the centerlines is done by changing absolute and relative values. Refer to the online help on how to do this ([help centerline](#)).

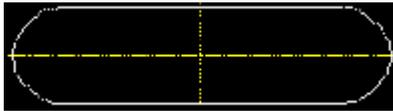
Figure 47. Creating a Centerline



When you dimension centerlines and then make a modification, the dimension is automatically updated. When asked to click the first element, click the center point of the circle (use Catch Center).

To draw a centerline:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Center Line**.
2. Click the required **Center Line** button and perform the associated procedure.

Center Line Button	Purpose	Procedure
 Create	Create a centerline.	Click the circle. See Figure 47 on page 199 .
 Slot Single	Create single slot centerlines.	Specify a circular element of a slot. Creo Elements/Direct Drafting draws the single slot centerlines. 
 Slot Double	Create double slot centerlines.	Specify a circular element of a slot. Creo Elements/Direct Drafting draws the double slot centerlines. 

Limitations of slot centerlines:

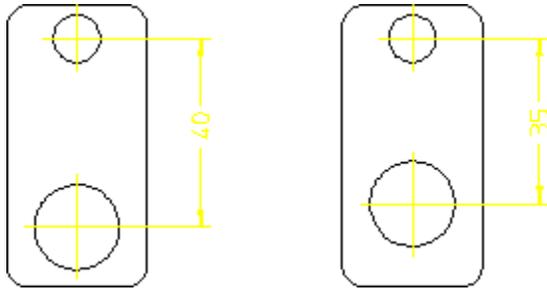
- Slot centerlines can be created only between two 180 degree arcs.
- The start and end points of the two arcs must be on a line that is perpendicular to the line connecting the center points of the arcs. After the center lines are created, you can modify the slot element.

Changing Position of Centerlines

To change the position of centerline:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Cen Line**.
2. Click  **Modify**.
3. Select an end of the centerline.
4. Specify the new position of the centerline.

Figure 48. Effect of Changing the Position



Creating Symmetric Elements

Creo Elements/Direct Drafting allows you to use  **Symmetry Line** for creating elements that are:

- Symmetric with existing elements
- Associated (or anchored) to the defining elements.

When you modify the defining elements, the associated element is modified accordingly.

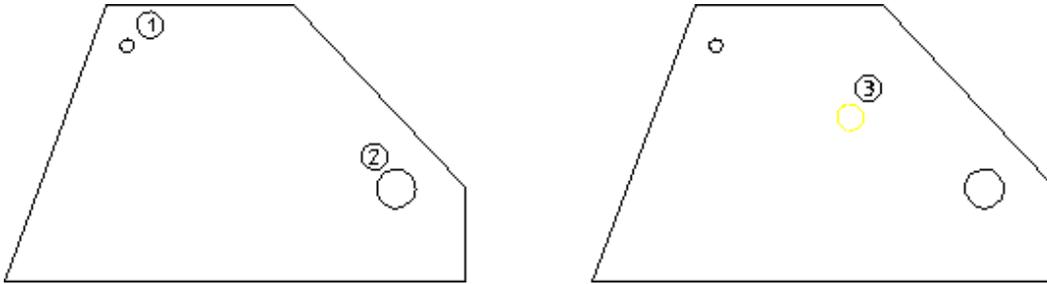
To draw an associated element:

1. Draw a polygon similar to that shown in [Figure 49 on page 202](#).
2. Create circle 1 having diameter 1 mm.
3. Create circle 2 having diameter 3 mm.
4. Click **Drafting** and then, in the **Annotate** group, click  **Symmetry Line**.
5. Click circle at point 1.
6. Click circle at point 2.

Creo Elements/Direct Drafting draws the associated circle (point 3) that is:

- Midway between circles 1 and 2.
- 2 mm in diameter (half the sum of the defining diameters).

Figure 49. Associated Symmetric Elements



To change the extension of the symmetry line:

1. Click **Modify** and then, in the **Annotate** group, click  **Sym Line**.
2. Select the end of the symmetry line and then select the new end position of the symmetry line.

To change the position of the symmetry line, move the circular or linear elements to which the symmetry line is associative. So try moving the circle at position 2 and notice how the associated element also moves.

Figure 50. Effect of Moving the Defining Circle

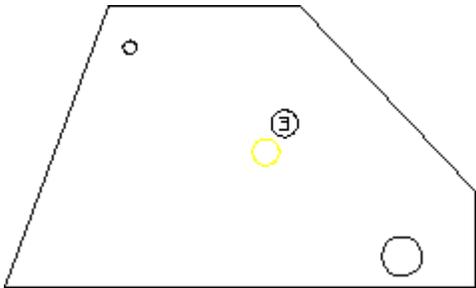
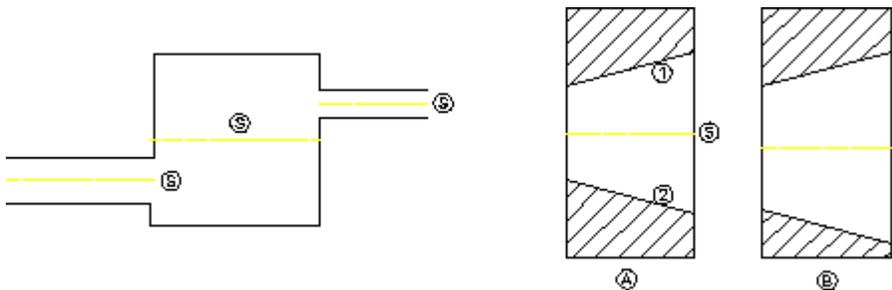


Figure 51. Finding Lines of Symmetry on page 202 shows another example of where to use  **Symmetry Line**.

Figure 51. Finding Lines of Symmetry



Drawing the Projected Reference Point

The "projected reference point" is the point at which the extended traces of two existing elements (line-line, line-arc, and arc-arc) would meet. To draw this point in your drawing:

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Projected Reference Point**, click  **New**.
3. Click the first element.
4. Click the second element.

The point where the traces of the two elements meet is now visible in your drawing.

When one or both the elements are arcs, the intersection point that is closest to both the elements is the projected reference point.

Note that this reference point (plus the associated traces) are fully associative:

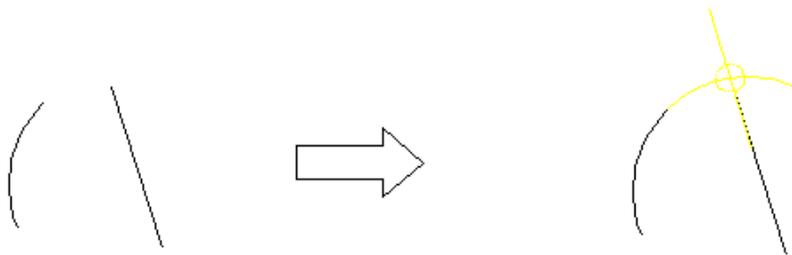
- These items can be included in dimensions.
- These items are preserved even when you perform **Modify** operations.

Note

You cannot have projected reference points across parts.

Figure 43 on page 203 shows an example of a projected reference point.

Figure 52. Projected Reference Point



Changing the Settings of the Projected Reference Point

To change the settings of the projected reference point and its associated elements:

-
1. Click **File** ► **Settings** ►  **Projected Reference Points**.

The **Projected Reference Point Settings** dialog box opens.

2. Specify the type, color, width, length, radius, and **Show** options for the projected reference point.

Further Information

For further technical information about drawing methods, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- `help line`
- `help circle`
- `help arc`
- `help ellipse`
- `help hyperbola`
- `help parabola`
- `help point`
- `help polyelem`
- `help equidistance`
- `help overdraw`
- `help c_line`
- `help c_circle`
- `help fillet`
- `help chamfer`
- `help centerline`
- `help symline`
- `help prp`

Creo Elements/Direct Drafting displays further information about drawing methods.

9

Modifying a Drawing

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This chapter describes how to use the buttons and options that allow you to reposition or alter your drawing.

You can modify single drawing elements (for example, lines and circles) or enclose the elements you want to modify in a box.

For a description of the **Picture** command, see [Importing a Picture File](#) on page 544.

Keeping or Removing the Original Elements

For all the buttons in the **General** group on the **Modify** tab, (except  **Isometric**), you can click or clear the **Keep** check box on the **Modify** tab. This determines whether the original drawing elements are kept or deleted after the modification.

For example, using  **Move** (with the **Keep** check box clicked) copies the drawing elements rather than just moving them. By default, the **Keep** check box is cleared.

Note

You can also click or clear the **Keep** check box when any button in the **General** group on the **Modify** tab is active.

Repeating a Modification

You can use the **Repeat** check box to repeat a modification a given number of times. For example, by clicking the **Repeat** check box and specifying a repeat factor of 5, you can perform the same **Move** operation five times.

Note

You can also click or clear the **Repeat** check box when any button in the **General** group on the **Modify** tab is active.

Modifying Using Selection

The **Selection** mechanism described in [Commonly Used Commands on page 72](#) can also be used with all the buttons on the **Modify** tab (except  **Isometric** and  **Stretch**).

Using  **Select** provides various ways to select elements to be modified and view them highlighted on the screen before the modification is made.

To use  **Select** with **Modify** tab:

-
1. Click the required modification button (for example, in the **General** group, click  **Move** ►  **Horizontal**) on the **Modify** tab.
 2. In the **Utilities** group, click  **Select**.
 3. Click **Advanced**.
The **Select** dialog box opens.
Alternatively, press F2 to open the **Select** dialog box.
 4. Select the elements to be modified using the buttons available in the **Select** dialog box. For example, you can click **Circle**, then **All**, and then **OK** in the **Select** dialog box to perform the same modification on all circles.

 **Note**

Each time you select elements for modification, you can decide whether you want to select:

- From the current part only (this is the default)

or

- From the entire parts tree (by clicking **Select** ►  **Global**).

Refer to [Handling Parts and Assemblies on page 429](#) for details on handling parts and assemblies.

Changing the Number of Elements in the Visual Feedback

You can change the number of elements displayed in the visual feedback during a **Modify** operation. For example, you may want to unclutter the screen during a **Move** of a complex assembly.

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. In the **Environment** settings, type the required number of elements in the **Max feedback** box.

Moving Elements

Creo Elements/Direct Drafting allows you to move or copy elements to another position using the Handles or the various  **Move** buttons.

To move an element with the handles:

1. Click an element in the drawing area.
The handles appear.
2. Click the element again and drag it to the new position.

To move multiple elements with handles:

1. Press SHIFT and click all the elements in the drawing area.
The handles appear.
2. Release SHIFT and click one of the elements again.
3. Drag the elements to a new location.

Click the  **Move** buttons to move or copy drawing elements to another position.

Note

Creo Elements/Direct Drafting shows a visual feedback that realistically changes when you drag a geometry.

Click the  **Move** buttons with the **Keep** check box clicked to produce a copy. The original remains unchanged.

Click the **Repeat** check box, type a repeat factor in the user input line and press ENTER. This repeats the next operation the specified number of times.

To move an element using  **Move**:

1. Click **Modify** and then, in the **General** group, click the arrow next to  **Move**.
2. Click the required **Move** button and perform the associated procedure.

Move Button	Purpose	Procedure
 2 Points	Move or copy a drawing element or elements using a reference point	<ol style="list-style-type: none"> a. Select the element or elements to be moved. b. To define the move vector, specify a reference point and then a destination point. <p>You can click any two points to define a move vector.</p> <p>Creo Elements/ Direct Drafting moves the element or elements to the new position.</p>
 Horizontal	Move or copy a drawing element or elements to a new position in the horizontal direction.	<ol style="list-style-type: none"> a. Select the element or elements to be moved. b. Type the required horizontal distance in the user input line and press ENTER. <p>or</p> <p>Specify a reference point and then a destination point to define the horizontal distance.</p> <p>If a repeat factor has been specified, the same horizontal distance will be taken for each move or copy.</p>

Move Button	Purpose	Procedure
		<p> Note</p> <p>You can interrupt the command with the  Length or  Distance button if the required distance can be measured from existing features on the drawing.</p> <p>Alternatively, you can specify two points defining the horizontal distance required.</p>
 Vertical	<p>Move or copy a drawing element or elements to a new position in the vertical direction.</p>	<p>a. Select the element or elements to be moved.</p> <p>b. Type the required vertical distance in the user input line and press ENTER.</p> <p>or</p> <p>Specify a reference point and then a destination point to define the vertical distance.</p> <p>If a repeat factor has been specified, the same vertical distance will be taken for each move or copy.</p>

Move Button	Purpose	Procedure
 Multiple	Interactively move or copy a drawing element or elements to any number of new positions.	<ol style="list-style-type: none"> Select the element or elements to be moved. Specify a reference point for the move. Specify the destination point for the move. The selected element or elements moves to the new position defined by these two points. Specify as many further destination points as required. Terminate the move or copy by clicking another button, another option, or  OK.

Rotating Drawings

Creo Elements/Direct Drafting allows you to rotate elements using **Rotate**.

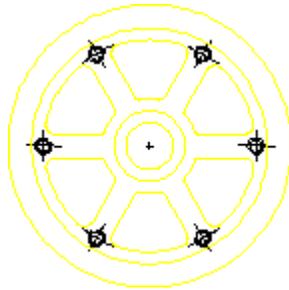
Elements can be rotated:

- About a Center



Center is especially useful when combined with a **Repeat** factor for repeated rotation. The flywheel shown in [Figure 57. Flywheel on page 213](#) was drawn by rotating the holes with a repeat factor of 5.

Figure 57. Flywheel



- By Reference point ( **2 Points**)

Both of these methods can be used with the **Keep** check box clicked or cleared on the **Modify** tab.

To rotate a drawing:

1. Click **Modify** and then, in the **General** group, click  **Rotate**.
2. Click the required **Rotate** button and perform the associated procedure.

Rotate Button	Purpose	Procedure
 Center	Rotate a drawing or drawing element about a center.	<ol style="list-style-type: none"> a. Specify the center of rotation. b. Type the angle of rotation in the user input line and press ENTER. or Define the angle by specifying two points. A positive angle results in a counterclockwise rotation. c. Select the element or elements to be rotated.
 2 Points	Rotate a drawing or drawing element using a reference point.	<ol style="list-style-type: none"> a. Specify a rotation reference point on the drawing or element. b. Specify the destination for the rotation reference point. c. Type the angle of rotation in the user input line and press ENTER. or Define the angle by specifying two points.

Rotate Button	Purpose	Procedure
		<p>A positive angle results in a counterclockwise rotation.</p> <p>d. Select the element or elements to be rotated.</p>

Mirroring

Creo Elements/Direct allows you to mirror a drawing using  **Mirror**.

To save time when drawing symmetrical shapes, you can mirror them about a line of symmetry.

On the **Modify** tab,  **Mirror** is usually used with the **Keep** check box clicked. The repeat factor is set to 1. Wait until your drawing is nearly complete before mirroring, otherwise you may miss a feature and have to draw it in both halves.

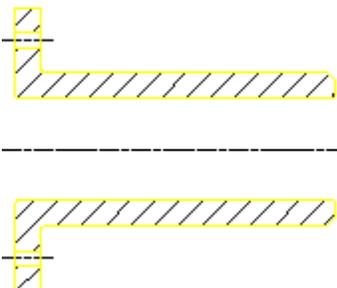
To mirror a drawing:

1. Click **Modify** and then, in the **General** group, click  **Mirror**.
2. Click the required **Mirror** button and perform the associated procedure.

Mirror Button	Purpose	Procedure
 2 Points	Mirror a drawing about two points.	<p>a. Specify two points to define the line of symmetry.</p> <p>b. Select the elements to be mirrored.</p> <p>The line of symmetry in Figure 58 on page 216 is the dashed line.</p>
 Horizontal	Mirror a drawing about a horizontal line.	<p>a. Specify a point on the horizontal line of symmetry.</p> <p>b. Select the elements to be mirrored.</p>

Mirror Button	Purpose	Procedure
 Vertical	Mirror a drawing about a vertical drawing.	<ol style="list-style-type: none"> Specify a point on the vertical line of symmetry. Select the elements to be mirrored.
 Center	Rotate a drawing through 180 degrees.	<ol style="list-style-type: none"> Specify the center point of rotation. Select the elements to be mirrored. <p>The elements are rotated 180 degrees in a clockwise direction.</p>

Figure 58. Line of Symmetry



Making Elements Larger or Smaller

Creo Elements/Direct Drafting allows you to make drawing elements and text larger or smaller using  **Scale**. Note that dimensions will be changed accordingly. If you want to keep the original drawing elements, click the **Keep** check box on the **Modify** tab.

To scale your drawing:

1. Click **Modify** and then, in the **General** group, click the arrow next to  **Scale**.
2. Click the required **Scale** button and perform the associated procedure.

Scale Button	Purpose	Procedure
 Center	Scale a drawing by center.	<ol style="list-style-type: none"> Specify the center for the scaling operation. Type a scale factor. or Specify two points to define the scale factor. Select the elements to be scaled. <p>When drawing elements are scaled using Center, one point on the drawing remains fixed.</p>
 2 Points	Scale your drawing using a reference point.	<ol style="list-style-type: none"> Specify a reference point and then specify the new position for this point. Type a scale factor. or Specify two points to define the scale factor. Select the elements to be scaled. <p>The selected elements are moved and scaled.</p>

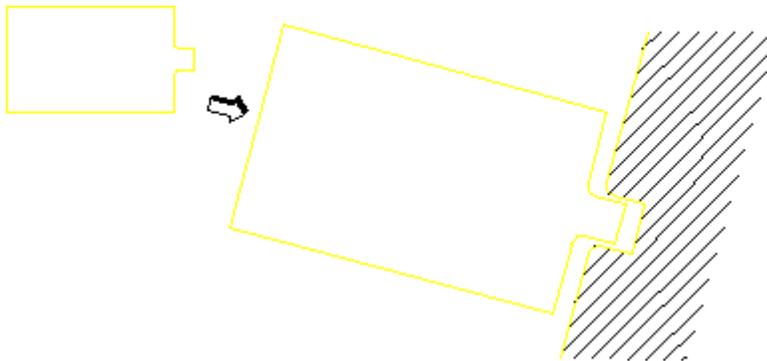
Drawing Similar Elements

 **Similar** is a combination of  **Move**,  **Rotate** and  **Scale**. It allows you to fit specified drawing elements between two given points. This involves rescaling.

To fit a specified drawing element between two given points:

1. Click **Modify** and then, in the **General** group, click **More**.
2. Click  **Similar**.
3. Specify two reference points on the drawing.
4. Specify two destination points for those reference points.
5. Select the elements of which you want to create a similar copy. These drawing elements will then be scaled to fit the new position. [Figure 59 on page 218](#) shows an example.

Figure 59. Fitting Drawing Elements Between Two Points



Creating a Parallel Perspective (Oblique) View

Creo Elements/Direct Drafting allows you to create a parallel perspective view using  **Transform**.

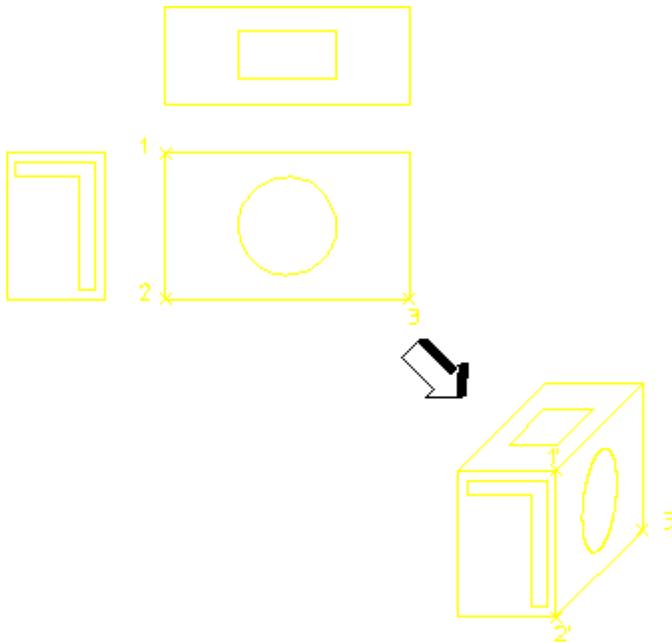
Parallel perspective means that three view points for the perspective view are taken to be at infinity, and parallel lines will therefore remain parallel.

Example

[Figure 60 on page 219](#) shows an oblique view created by copying the end elevation, then the side and top created by  **Transform**.

Points used for the first  **Transform** operation are shown.

Figure 60. Creating an Oblique View



Position the End View

1. Click **Modify** and then, in the **General** group, click the arrow next to  **Move**.
2. Click  **2 Points**.
3. Specify a reference point on the drawing (end view).
4. Specify a destination point.
5. Select the end view (marked by an upside-down L in [Figure 60 on page 219](#)).

Position the Side View

1. Click **Modify** and then, in the **General** group, click **More**.
2. Click  **Transform**.
3. Click three reference points on the drawing (1, 2, and 3).
4. Click three destination points (1', 2', and 3') to which the three reference points will be mapped in the parallel perspective view.
5. Select the side view (marked by a circle in [Figure 60 on page 219](#)).

Position the Top View

1. Click **Modify** and then, in the **General** group, click **More**.
2. Click  **Transform**.

3. Click three reference points on the drawing.
4. Click three destination points to which the three reference points will be mapped in the parallel perspective view.
5. Select the top view (marked by a rectangle in [Figure 60 on page 219](#)).

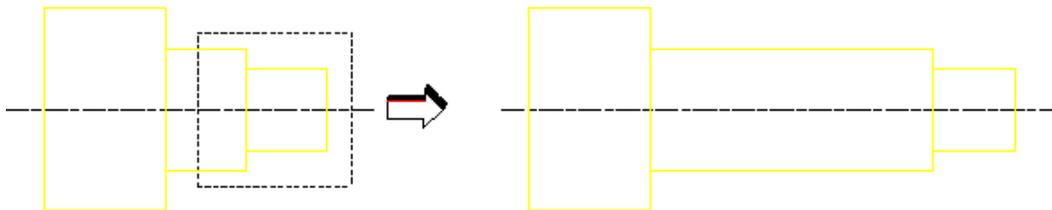
Stretching a Drawing

Creo Elements/Direct Drafting allows you to alter the shape of your drawing by stretching it in a certain direction using  **Stretch**. All the dimensioning, hatching, and finishing information will be modified to fit the new stretched shape.

Example

The following example demonstrates the use of  **Stretch**. See [Figure 61 on page 220](#).

Figure 61. Stretching a Drawing



1. Click **Modify** and then, in the **General** group, clear the **Keep** check box.
2. In the **General** group, click **More**.
3. Click  **Stretch** and then click the required button.
4. Specify a reference point on the drawing.
5. Specify a destination point.
6. Select the elements to be stretched.

You can select the elements by clicking a single vertex or enclosing several elements in a box (the vertex is moved and the element is stretched accordingly).

Alternatively, you can preselect the elements that you want to stretch. In the preselection mode, the elements are stretched after you specify a reference point and a destination point.

You can combine stretching with any of the other **Modify** buttons (except  **Isometric**).

To stretch a drawing:

1. Click **Modify** and then, in the **General** group, clear the **Keep** check box.
2. In the **General** group, click **More**.
3. Click  **Stretch**.
4. Click the required  **Stretch** button and perform the associated procedure.

Stretch Button	Purpose	Procedure
 Free	Stretch drawing elements in an arbitrary direction.	a. Select the drawing elements to be stretched. b. Specify a reference point and a destination point for the stretch operation.
 Horizontal	Stretch the drawing horizontally.	a. Select the drawing elements to be stretched. b. Type a horizontal distance in the user input line and press ENTER. or Specify a reference point and a destination point for the horizontal distance.
 Vertical	Stretch the drawing vertically.	a. Select the drawing elements to be stretched. b. Type a vertical distance in the user input line and press ENTER.

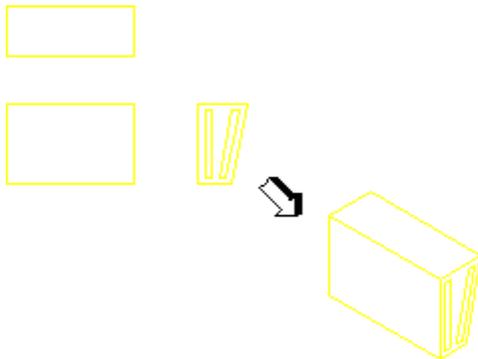
Stretch Button	Purpose	Procedure
		<p>or</p> <p>Specify a reference point and a destination point for the vertical distance.</p> <p> Note</p> <p>You can interrupt the command by clicking the  Length button, if the distance you need can be measured from existing features on the drawing.</p>

Creating an Isometric View

To draw an approximate isometric view of a three-dimensional object, you must have drawn the front, side, and top views already.

The following example shows how to create an isometric view. Refer to [Figure 62. Creating an Isometric View on page 222](#).

Figure 62. Creating an Isometric View



1. Click **Modify** and then, in the **General** group, click **More**.
2. Click  **Isometric**.
3. Select the front view by enclosing it in a box.

4. Click a reference point on the front view.
5. Specify the destination for the front view reference point.
6. Select the side view by enclosing it in a box.
7. Click a reference point on the side view.
8. Specify the destination for the side view reference point.
9. Select the top view by enclosing it in a box.
10. Click a reference point on the top view.
11. Specify the destination for the top view reference point.

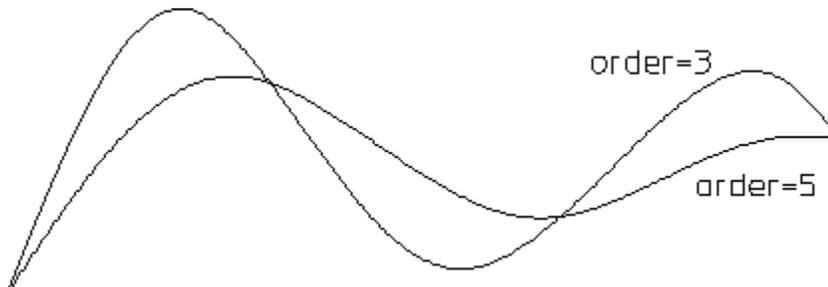
Modifying Splines

Creo Elements/Direct Drafting allows you to specify the order of spline and modify the interpolation and control splines.

Specifying the Order of a Spline

The spline order controls the smoothness of the curve. The higher the order, the smoother the spline. [Figure 63. Spline Order on page 223](#) shows two splines drawn through the same control points.

Figure 63. Spline Order



Be sure to specify the order of the spline before you specify the last interpolation point (for interpolation splines) or control point (for control splines) for the spline.

Creo Elements/Direct Drafting allows you to change the order of the splines using

 **Order** while creating splines (see [Drawing Splines on page 161](#)).

To change the order of the interpolation spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify General** click  **Order**.
3. Type the order of the spline in the user input line and press ENTER.

To change the order of the control spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Control**.
2. Under **Modify General**, click  **Order**.
3. Type the order of the spline in the user input line and press ENTER.

Appending an Interpolation Spline

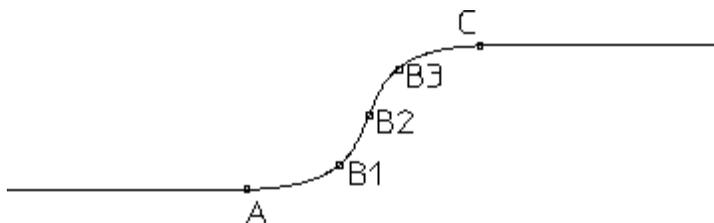
To append an interpolation spline to existing geometry:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify General**, click  **Append**.
3. Click the end point of the line to which you want to append an interpolation spline.
4. Specify a sequence of points.
5. Click  **OK**.

To append the begin-point and end-point of an interpolation spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify General**, click  **Append**.
3. Specify point A and a series of points B1, B2, B3, till Bn.
4. Click  **Append**.
5. Specify point C.
6. Click  **OK**.

Figure 64. Appending an Interpolation Spline



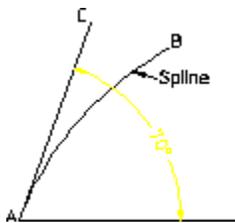
Specifying the Slope of an Interpolation Spline

While drawing an interpolation spline using  **Interpolation** (see [Drawing a Spline on page 162](#)), you can specify the slope angle at which the interpolation spline leaves or approaches the current interpolation point using  **Tangent**.

To specify the slope angle at which the interpolation spline leaves or approaches the current interpolation point:

1. Define the first interpolation points (such as point A in [Figure 65. Specifying the Slope of an Interpolation Spline on page 225](#)) when creating or appending an interpolation spline.
2. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol.**
3. Under **Modify General**, click  **Tangent**
4. Type in the slope angle (for example 70) and press ENTER.
Alternatively, you can specify a point at the required angle, for example, point C.
5. Specify the second point (B in [Figure 65. Specifying the Slope of an Interpolation Spline on page 225](#)).
6. Click  **OK**.

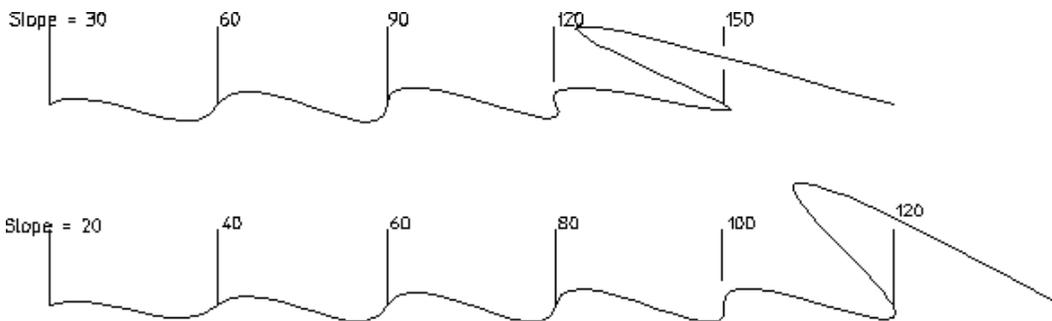
Figure 65. Specifying the Slope of an Interpolation Spline



By specifying a slope at a given interpolation point, you add a tangency condition to the spline. This tangency condition can be modified or deleted

[Figure 66. Effects of Different Slope Angles on page 226](#) shows the effects of specifying different slope angles:

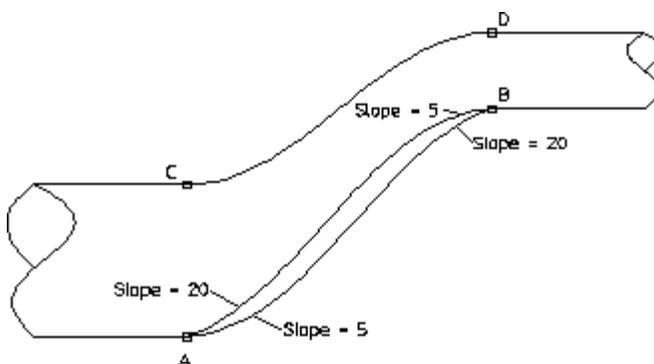
Figure 66. Effects of Different Slope Angles



Being able to specify the slope of a spline is useful when joining unequal diameters. The following diagram shows the difference between curves created

with  **Append** (curve CD) and  **Tangent** (curve AB).

Figure 67. Slope of Interpolation Splines



To draw curve C to D, do the following:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify General**, click  **Append**.
3. Click point C.
4. Click the arrow next to  **Interpol**.
5. Click point D.
6. Click  **OK**.

To draw the lower curve A to B, do the following:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Spline**.
2. Click  **Interpolation**.

-
3. Click point A.
 4. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
 5. Under **Modify General**, click  **Tangent**.
 6. Type 5 in the user input line and press ENTER.
 7. Click point B.
 8. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
 9. Under **Modify General**, click  **Tangent**.
 10. Type 20 in the user input line and press ENTER.
 11. Click  **OK**.

Modifying Interpolation Points

Creo Elements/Direct Drafting allows you to modify the interpolation points using **Modify Point** buttons under  **Interpol** in the **Draw** group on the **Modify** tab.

Adding an Interpolation Point

To add an interpolation point to an interpolation spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify Point**, click  **Add**.
3. Click the spline to which you want to add an interpolation point.
4. Click the position on the spline where you want to add the interpolation point.

When you add an interpolation point, the shape of the spline changes accordingly.

Moving an Interpolation Point

To change the position of an interpolation point of a spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify Point**, click  **Change**.
3. Click the spline you want to modify.
4. Click the interpolation point you want to move.
5. Click the new position of the interpolation point.

When you move an interpolation point, other interpolation points also move. The shape of the spline changes accordingly.

Deleting an Interpolation Point

To delete an interpolation point of a spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify Point**, click  **Delete**.
3. Click the spline you want to modify.
4. Click the interpolation point you want to delete.

When you delete an interpolation point, the shape of the spline adjusts to the positions of the remaining interpolation points.

Changing the Spline Shape by Moving an Interpolation Point

To change the shape of an interpolation spline by moving one of its interpolation points:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify Point**, click  **Shape**.
3. Click the spline you want to modify.
4. Click the interpolation point you want to move.
5. Specify the new position of this interpolation point.

When you move an interpolation point, the shape of the spline changes accordingly.

Changing the Slope of an Interpolation Spline

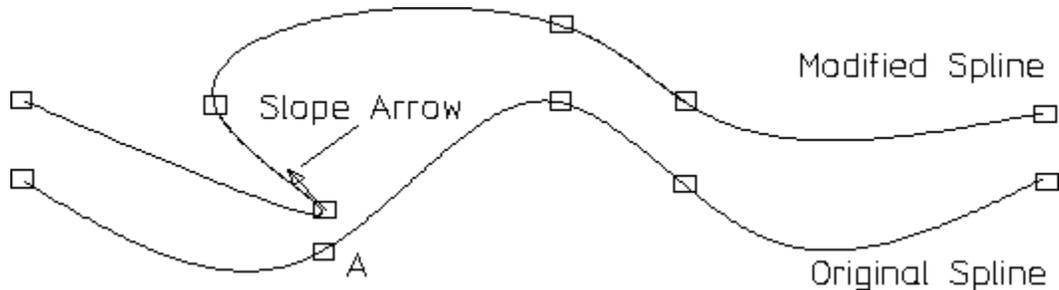
Creo Elements/Direct Drafting allows you to change the slope of an interpolation spline using **Modify Tangential** buttons under  **Interpol** in the **Draw** group on the **Modify** tab.

Changing the Slope at an Interpolation Point

You can change the spline slope at any interpolation point without moving interpolation points. The spline slope defines the angle at which a spline approaches an interpolation point. This is useful if you are merging a spline with

another line. [Figure 68. Effect of Changing the Slope at an Interpolation Point on page 229](#) shows a simple spline and the same spline with the tangent of interpolation point A changed.

Figure 68. Effect of Changing the Slope at an Interpolation Point



To change the spline slope at an interpolation point:

1. Click the **Vertex** check box in the **Show** dialog box.

The interpolation points and any points where you changed the spline slope are displayed.

2. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
3. Under **Modify Tangential**, click  **Add**.
4. Click the interpolation spline.
5. Click an interpolation point of the spline.

Creo Elements/Direct Drafting displays an arrow to indicate the spline slope at that point.

6. Type the required slope angle (for example, 70) and press ENTER.

Alternatively, you can specify a point at the required angle.

The spline slope at other interpolation points changes accordingly. However, the interpolation points do not move.

Changing the Slope at Any Point on an Interpolation Spline

You can change the spline slope at any point on the interpolation spline. This operation will move the interpolation points of the spline.

To change the spline slope at any point on the spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify Tangential**, click  **Change**.
3. Click the interpolation spline.

4. Click an interpolation point of the spline.

Creo Elements/Direct Drafting displays an arrow to indicate the spline slope at that point.

5. Type the required slope angle (for example, 70) and press ENTER.

Alternatively, you can specify a point at the required angle.

The interpolation points of the spline move to new positions. The spline slope at the interpolation points changes accordingly.

Deleting a Tangency Condition of an Interpolation Spline

You can undo any spline slope change (see [Specifying the Slope of an Interpolation Spline on page 225](#), [Changing the Slope at an Interpolation Point on page 228](#), and [Changing the Slope at Any Point on an Interpolation Spline on page 229](#)) by deleting the associated tangency condition.

To delete a tangency condition:

1. Click the **Vertex** check box in the **Show** dialog box.

The interpolation points and any points where you changed the spline slope are displayed.

2. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.

3. Under **Modify Tangential**, click  **Delete**.

4. Click the spline.

5. Click the point where you defined or changed the slope angle.

The spline returns to the shape it had before you defined or specified a slope angle at a given point on the spline.

Changing the Shape of an Interpolation Spline

You can change the shape of an interpolation spline without moving any of its interpolation points.

To change the shape of an interpolation spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.

2. Under **Modify Tangential**, click  **Shape**.

3. Click the spline.

4. Click the interpolation at which you want to change the spline slope in order to change the shape of the spline.

Note that the shape of the spline changes but none of its interpolation points are moved.

Modifying Control Points

Creo Elements/Direct Drafting allows you to modify the size and shape of control spines by modifying its control points using **Modify Point** buttons under  **Control** on the **Draw** group of **Modify** tab.

Adding a Control Point

To add a control point to a control spline:

1. Click **Modify** and then in the **Draw** group, click the arrow next to  **Control**.
2. Under **Modify Point**, click  **Add**.
3. Click the spline you want to modify.
4. Click a position between two existing control points.
5. Move the cursor to the position where you want to add the new control point.
6. Click to add the new control point.

When you add a control point, the shape of the spline changes accordingly.

Moving a Control Point

To change the position of a control point of a spline:

1. Click **Modify** and then in the **Draw** group, click the arrow next to  **Control**.
2. Under **Modify Point**, click  **Change**.
3. Click the spline you want to modify.
4. Click the control point you want to move.
5. Click the new position of the control point.

When you move a control point, the shape of the spline changes accordingly.

Deleting a Control Point

To delete a control point of a control spline:

1. Click **Modify** and then in the **Draw** group, click the arrow next to  **Control**.
2. Under **Modify Point**, click  **Delete**.

3. Click the spline you want to modify.
4. Click the control point you want to delete.

When you delete a control point, the shape of the spline changes accordingly.

Changing the Shape of a Control Spline

You can change the shape of a control spline by moving one or multiple control points.

To change the shape of a control spline:

1. Click **Modify** and then in the **Draw** group, click the arrow next to  **Control**.
2. Under **Modify Point**, click  **Shape**.
3. Click the spline.
4. Click the control point you want to move in order to change the shape of the spline.

Note that the shape of the spline changes when you move one of its control points.

Changing the Slope at a Point on the Control Spline

Creo Elements/Direct Drafting allows you to change the spline slope at any point on the control spline using **Modify Tangential** button under  **Control** in the **Draw** group on the **Modify** tab. This operation moves the control points of the spline.

To change the spline slope at any point on the spline:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Control**.
2. Under **Modify Tangential** click  **Change**.
3. Click the control spline.
4. Click any point on the spline. Creo Elements/Direct Drafting displays an arrow to indicate the spline slope at that point.
5. Type in the required slope angle (for example 70) in the user input line and press ENTER.

Alternatively, you can specify a point at the required angle.

The spline slope changes accordingly. The control points of the spline move to new positions.

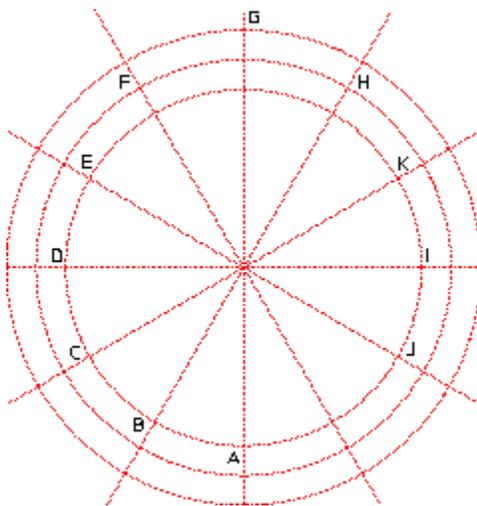
Example — Cam Profile

An example describing how to create a cam profile and then modify it follows:

Construct the following geometry (see [Figure 69. Cam Profile - 1](#) on page 233):

- Horizontal and vertical lines at 0 0
- Circles at 35 40 45
- Lines at angles 30 60 120 150

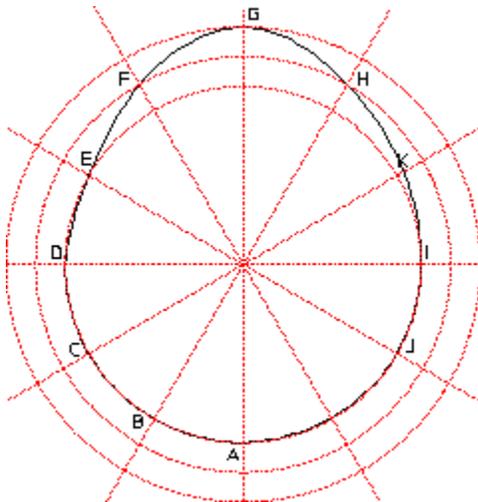
Figure 69. Cam Profile - 1



To create the initial profile:

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Spline**.
2. Click  **Interpolation**.
3. Click the **Closed** check box.
4. Specify points A, B, till J.
5. Click  **OK**.

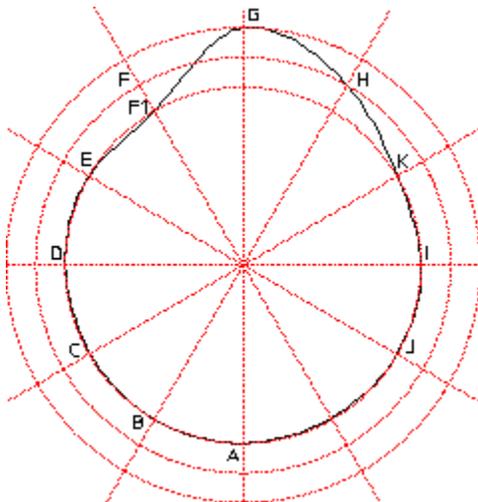
Figure 70. Cam Profile - 2



You now want to add an interpolation point at K:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify Point**, click  **Add**.
3. Click the spline.
4. Click a point on the spline that is close to point K.
5. Click the intersection point at K.

Figure 71. Cam Profile - 3



To delete an interpolation point:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.

2. Under **Modify Point**, click  **Delete**.
3. Click the spline.
4. Click point C.

The point is deleted.

To change the position of an interpolation point:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify Point**, click  **Change**.
3. Click the spline.
4. Click point F.
5. Move the point and click point F1.

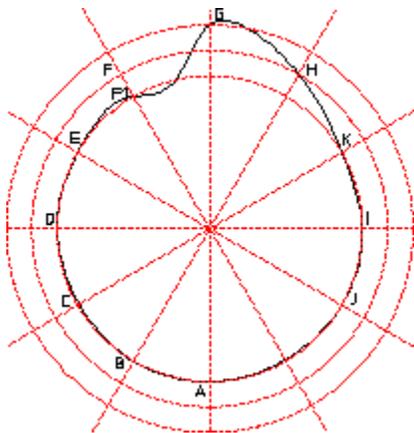
To change the angle at point F1:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Interpol**.
2. Under **Modify Tangential**, click  **Change**.
3. Click the spline.
4. Click point F1.

The system displays an arrow indicating the slope direction. You can now either click a point to indicate the direction or type a value in the user input line. (This example uses an angle of 10°.)

5. Type 10 in the user input line and press ENTER.

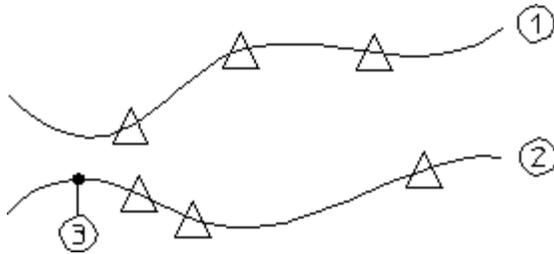
Figure 72. Cam Profile - Final Shape



Subdividing a Spline

 **Subdivide** either divides a spline into section of equal length (spline 1 in Figure 73. Dividing a Spline on page 236) or splits a spline at points along its length (spline 2 in Figure 73. Dividing a Spline on page 236).

Figure 73. Dividing a Spline



Dividing a Spline in Segments of Equal Length

Dividing splines into sections of equal lengths is useful, for example, when designing the shape of a ship's hull and you need to divide the ribs (rib-cage) into equal lengths.

To divide a spline into segments of equal length:

1. Click **Modify** and then, in the **Draw** group, click  **Subdivide**.
2. Click the spline you want to split.
3. Type an integer for the number of segments you want to produce (for example 4) in the user input line and press ENTER.

Creo Elements/Direct Drafting divides the spline into the specified number of segments.

Dividing a Spline by Specifying a Split Point

You can define a split point by specifying a length value. The spline splits at the end point of the spline segment defined by this length.

To divide a spline by specifying a split point:

1. Click **Modify** and then, in the **Draw** group, click  **Subdivide**.
2. Click the spline you want to split.
3. Click any point on the spline you want to split.
4. Specify a length.

A split point is created at the end point of the spline segment defined by this length.

If the **Trim** check box is clicked (see [Using the Trim Function on page 237](#)), the spline segment "beyond" this split point is hidden.

Converting Splines

Splines need to be converted only if you want to export them to devices (such as numerical control machines) that do not recognize spline curves.

Creo Elements/Direct Drafting allows you to convert splines to arcs and lines by approximation using  **Convert**.

To convert splines:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Convert**.
2. Click the required **Convert** button and perform the associated procedure.

Convert Button



Lines

Purpose

Convert splines to lines.

Procedure

Select the spline for converting it to a line.



Arcs & Lines

Convert splines to arcs and lines.

Select the spline for converting it to arcs and lines.

Using the Trim Function

Splines can be split in the same way as other line elements. However, you can click or clear the **Trim** check box, which affects the way in which a spline is split as follows:

- Click** Two new (independent) splines are created, each with their own mathematical behavior.
- Clear** A given section of the spline is hidden following the split. This means that the spline is drawn from the start (or end) point to the split point. Curve points beyond the visibility range are kept but are hidden.

[Figure 74. Splitting Splines on page 238](#) shows the effects of splitting a spline with the **Trim** check box:

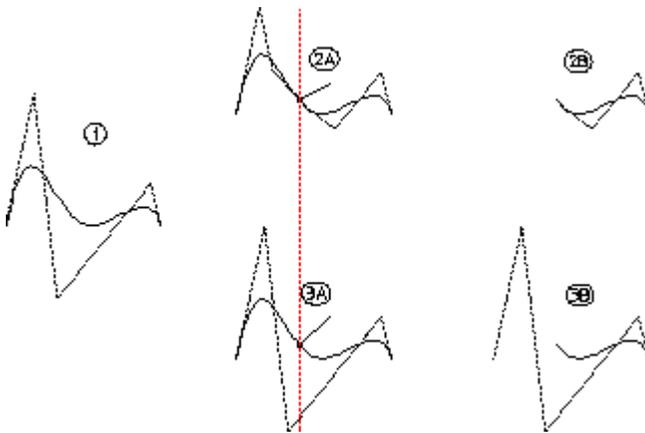
Click Splines 2A and 2B

The spline is split in two — with each spline having a new end-point, a new start-point, and individual control polygons.

Clear Splines 3A and 3B

The spline is split but the control polygon retains its shape. When the left part of the spline is deleted, the curve points are retained (but hidden).

Figure 74. Splitting Splines



In drawings 2B and 3B, the left spline has been deleted. Notice in drawing 3B how the control polygon thinks that the spline starts at its original point (drawing 1).

Creo Elements/Direct Drafting allows you to split and trim a spline using the **Trim** check box and  **Subdivide** in the **Draw** group on the **Modify** tab.

To split and trim a spline:

1. Click **Modify** and then, in the **Draw** group, click the **Trim** check box.
2. Click  **Subdivide**.
3. Divide the spline.

Proceed as described in [Subdividing a Spline on page 236](#).

Trimming or Extending Elements

Creo Elements/Direct Drafting allows you to trim or extend overlapping elements using  **Trim/Extend**. This is useful when you want to clean up a drawing. Note that construction geometry can be used as reference elements (indicating where an element is to be trimmed).

To trim overlapping elements:

1. Click **Modify** and then, in the **Draw** group, click  **Trim/Extend**.
2. Click the required **Trim/Extend** button and perform the associated procedure.

Trim/Extend Button	Purpose	Procedure
 1 Element	Trim one overlapping element.	<ol style="list-style-type: none">a. Click the first element (point 1).b. Click the intersecting element (point 2). See Figure 75 on page 241 .
 2 Elements	Trim two overlapping elements.	<ol style="list-style-type: none">a. Click the first element (point 1).b. Click the intersecting element (point 2). See Figure 76 on page 241 .

Trim/Extend Button	Purpose	Procedure
 Cut Middle	Cut out the middle of an element.	<ol style="list-style-type: none"> Click element (point 1). Click element (point 2). Click element (point 3). See Figure 77 on page 241.
 Chain	Trim a series of elements.	After you click the second element, Creo Elements/Direct Drafting trims the line or lines and prompts you to click the next element: <ol style="list-style-type: none"> Click point 1. Click point 2. Click points 3, 4, 5, 6. See Figure 78 on page 241. <p> Note</p> Use  Chain only after clicking  1 Element or  2 Elements . <p> Note</p> You cannot trim elements such as closed splines, circles, and closed polygons because split points are not defined.

Figure 75. Trimming One Element

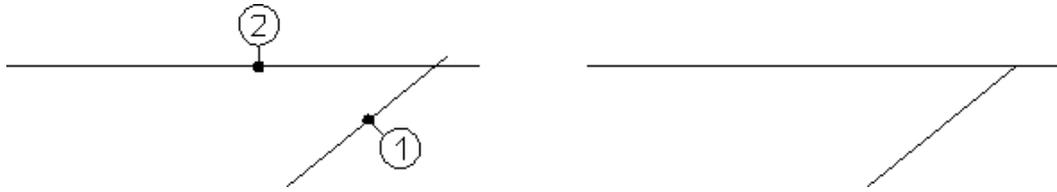


Figure 76. Trimming Two Elements

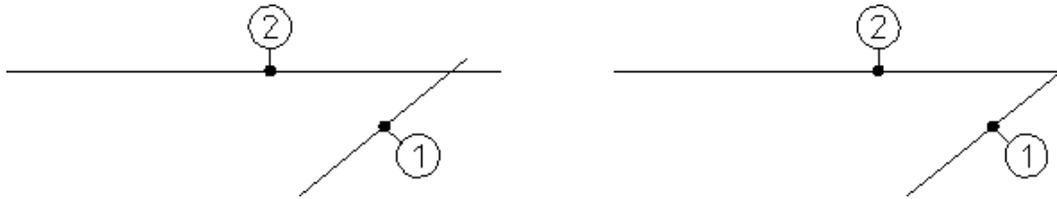
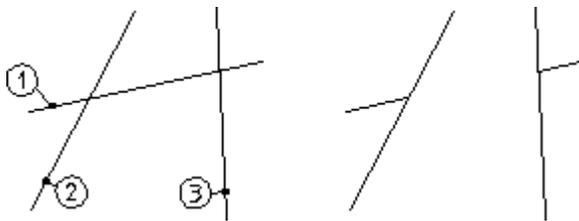
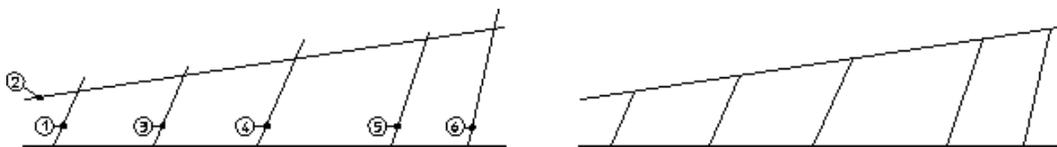


Figure 77. Cutting Out the Middle Section



Chain takes the last intersecting elements as the new intersecting elements.

Figure 78. Using the Chain Option



Grouping

You can group annotations or parts together.

- Single Dimension
- Multiple Text Elements
- Parts
- Multiple Symbols

Each group has one of its elements as the master element.

Creo Elements/Direct Drafting creates one invisible reline between each element in a group and its master element. These reflines can be shown with either the SHOW_INV_RTL or SHOW_INV_GEO command. Deleting an invisible reline deletes the group. You can change the flyby highlight color of the group box and of the group master element with the FLYBY_GROUP_MASTER_COLOR function.

Create a group

To create a group:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Groups**.
2. Click  **Create**.
3. Type GLOBAL in the user input line and press ENTER if you want to group elements from different parts.
4. Select the master element for the group.
5. Select additional elements to add them to the group.
6. Click  **OK**.

Ungroup all elements of a group

To ungroup all elements in an existing group:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Groups**.
2. Click  **Remove**
3. Select an element belonging to the group you wish to ungroup.
4. Click  **OK**.

Add elements to an existing group

To add elements to an existing group:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Groups**.
2. Click  **Add**.
3. Select an element belonging to the group to which you wish to add.
4. Select elements to add them to the group.
5. Click  **OK**.

Remove elements from an existing group

To remove elements from an existing group:

1. Click **Modify** and then, in the **Draw** group, click the arrow next to  **Groups**.
2. Click  **Delete**.
3. Select an element from the group that contains the elements, which you want to delete.
4. Select elements to remove them from the group.
5. Click  **OK**.

Limitations

- A group may contain only one master element.
- If a group contains a dimension, it automatically becomes the master element.
- Each element can belong to only one group.
- You cannot add an active part to a group.

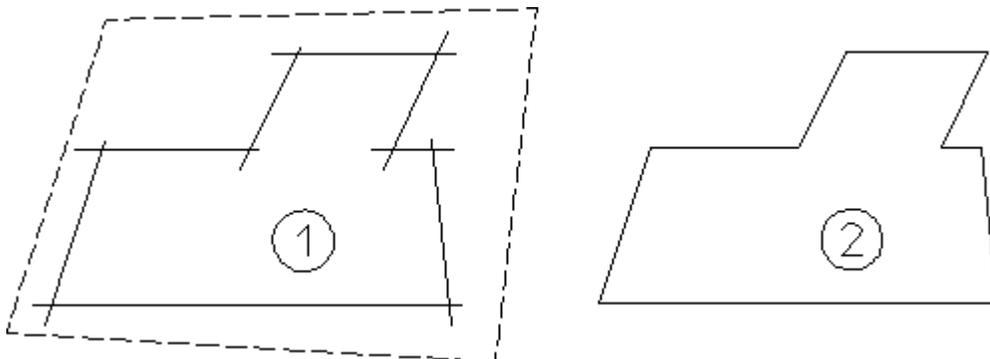
Creating a Contour

Creo Elements/Direct Drafting allows you to draw a smooth contour by deleting overlapping elements using  **Contour**

To create a contour from a number of overlapping lines:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Modify**, click  **Contour**
3. Enclose the elements in a box.

Figure 79. Creating a Contour



Splitting Elements

It is often useful to split an element (line, circle, and so on) in two or more parts.

Following the split operation, the segments of lines that intersect can be treated as separate elements and deleted individually.

Splitting Elements Automatically

Before you begin to draw, you must click the **Splitting** check box to automatically split elements at their intersection points. This technique keeps contours clean, which is important for automatic hatching.

To click the **Splitting** check box:

1. Click **File** ► **Settings** ►  **Geometry**.

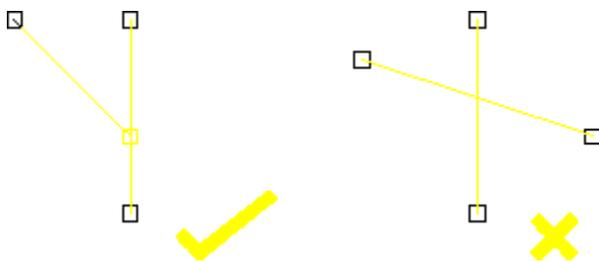
The **Geometry Settings** dialog box opens.

2. In the **Geometry** area, click the **Splitting** check box.

When a new element is created, Creo Elements/Direct Drafting checks to see if one of the end points of the new element lies on other elements. If so, it splits the other element at that point if possible. When the new element crosses the old element, the old element will not be split.

Figure 80. [Splitting Elements on page 244](#) shows the principle of splitting elements.

Figure 80. Splitting Elements



Note

If you are deleting an element whose end point lies on other elements and the **Splitting** check box is clicked, Creo Elements/Direct Drafting attempts to merge the remaining elements automatically into one.

Splitting Single Element or Elements in a Box

To split a single element:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Modify**, click  **Split**.
3. Click the element to be split. Creo Elements/Direct Drafting highlights the chosen element.
4. Click the point where the element should be split.

Alternatively, to split several elements automatically at their intersection points:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Modify**, click  **Split**.
3. Box the elements to be split.

Ensure that you fully enclose the elements in the box.

To verify that all elements have been split, click the **Vertex** check box in the **Show** dialog box. All existing points are highlighted.

Merging Elements

To merge elements that have been split:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Modify**, click  **Merge**.
3. Click the elements to be merged.

The elements will then be returned to their original state. Intersection points will no longer be marked when you click the **Vertex** check box in the **Show** dialog box.

Cleaning a Drawing

Inconsistencies in a part (for example, lines not quite parallel, small gaps between tangent objects, duplicate elements) are relatively common, but can cause problems for applications (like Parametric Design) that scan the part to extract information. To help eliminate drawing inconsistencies, Creo Elements/Direct Drafting provides the following several tools that can do limited cleaning of parts under **Clean**:

-  **Pnt**

-  **Duplicate**
-  **Stacked**
-  **Staggered**
-  **Gaps**

Points

Model points within a given distance of each other are merged, and the elements attached to the merged points are adjusted (moved, stretched, or rotated) as necessary to reflect the new positions of the points. For each set of points to be

merged,  **Pnt** chooses a location within the given tolerance and moves the points there. In general, there is no way to accurately predict or control which elements will be adjusted as a result of the merge. This should not be a limitation

in most cases, however, because the tolerance used is likely to be small.  **Pnt** is most useful for removing small gaps between elements.

To use  **Pnt**:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Clean**, click  **Pnt**.
3. Type a distance tolerance (current length units) in the user input line and press ENTER.

Points within the given tolerance are marked with a *, and will be merged. Elements that will be modified as a result of the merge are highlighted.

4. Click  **Confirm** to merge the marked points and modify the highlighted elements.

Duplicate

 **Duplicate** scans the part for duplicate elements and deletes them. It selects the element of a duplicate pair to delete. Elements are considered to be duplicates if their model points are identical. An exception is made for concentric circles (only); if the radii of concentric circles are within a given tolerance, they are

treated as duplicates. Normally,  **Duplicate** should be used after  **Pnt** to clean up duplicate elements that may result from merging model points.

To use  **Duplicate**:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Clean**, click  **Duplicate**.
3. Type a distance tolerance (current length units) for the removal of duplicate elements in the user input line and press ENTER.

Elements that will be deleted are highlighted.

4. Click  **Confirm** to remove duplicate elements.

Stacked

 **Stacked** finds stacked lines and arcs in the current part and calls  **Split** to split them.

To use  **Stacked**:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Clean**, click  **Stacked**.

Elements to be split are highlighted and the points to be used for the splits are marked with a *.

3. Click  **Confirm** to split elements.

Staggered

 **Staggered** finds staggered (partially overlapping) lines and arcs in the current part and merges these to form a single "new" line or arc. The "old" lines or arcs are removed by this operation.

To use  **Staggered**:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Clean**, click  **Staggered**.
3. Type a distance tolerance (current length units) in the user input line and press ENTER.

Elements that will be deleted are highlighted.

4. Click  **Confirm** to delete superfluous elements following the merge.

Gaps



Gaps uses the Creo Elements/Direct Drafting  **Trim/Extend** button to close "trivial" gaps between two or more elements, even between circular and linear elements.

- "Trivial" gaps are small gaps that can be closed automatically without any major impact on the drawing. These can be closed automatically.
- "Non-trivial" gaps cannot be eliminated automatically because their elimination may lead to unpredictable results. Therefore, "non-trivial" gaps cannot be closed automatically. "Non-trivial" gaps are typically characterized by distances and/or complexities greater than those found in "trivial" gaps.

To automatically close "trivial" gaps:

1. Click **Modify** and then, in the **Draw** group, click **More**.
2. Under **Clean**, click  **Gaps**.
3. Draw a selection box around the elements between which there are gaps that you want to close.

Note

If you use the  **Global** selection mode, you may unintentionally close "correct" gaps between elements

4. Type a distance tolerance (in current length units) in the user input line and press ENTER.

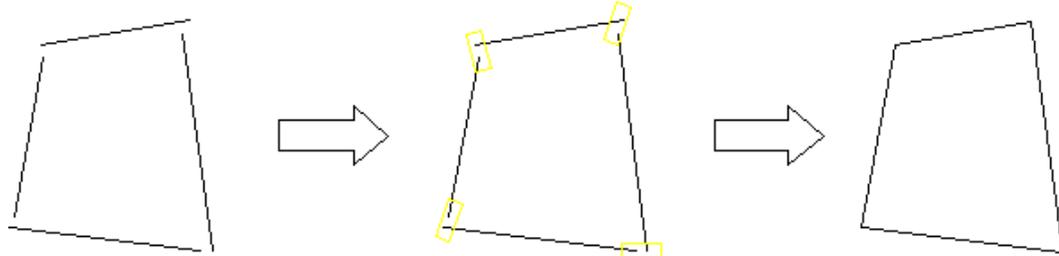
Note that this distance tolerance needs to be a "realistic" (small) value in order to prevent the elimination of required spacing between elements. If the distance tolerance is too big, a very large number of gaps may be shown. If all gaps are shown in the drawing, your drawing may be cluttered with color rectangles indicating "non-trivial" gaps.

The number of "trivial" and "non-trivial" gaps found in your selection is indicated in the prompt line. However, note that "non-trivial" gaps are not indicated in the drawing.

5. Check the "trivial gaps" preview for plausibility.
6. Click  **Confirm** to close any "trivial" gap or gaps.

Figure 81. Closing Trivial Gaps on page 249 shows an example of a set of elements separated by "trivial" gaps. These gaps are highlighted in the drawing and are then closed. Note that the gap sizes are exaggerated for visual clarity.

Figure 81. Closing Trivial Gaps



Show Gaps

Creo Elements/Direct Drafting allows you to box "non-trivial" gaps between two or more elements using  **Show Gaps**.

To mark "non-trivial" gaps:

1. Click **Modify** and then, in the **Draw** group, click  **Show Gaps**.
2. Type a box color for "trivial" gaps in the user input line and press ENTER.
3. Type a box color for "non-trivial" gaps in the user input line and press ENTER.
4. Draw a selection box around the elements between which there are or may be gaps.
5. Type a distance tolerance (current length units) in the use input line and press ENTER.

Note

This distance tolerance needs to be a "realistic" (small) value in order to prevent the elimination of required spacing between elements. If the distance tolerance is too big, a very large number of gaps may be shown. If all gaps are shown in the drawing, your drawing may be cluttered with color rectangles indicating "non-trivial" gaps.

The number of "trivial" and "non-trivial" gaps found in your selection is indicated in the prompt line.

6. Type the number of "non-trivial" gaps in the user input line and press ENTER.

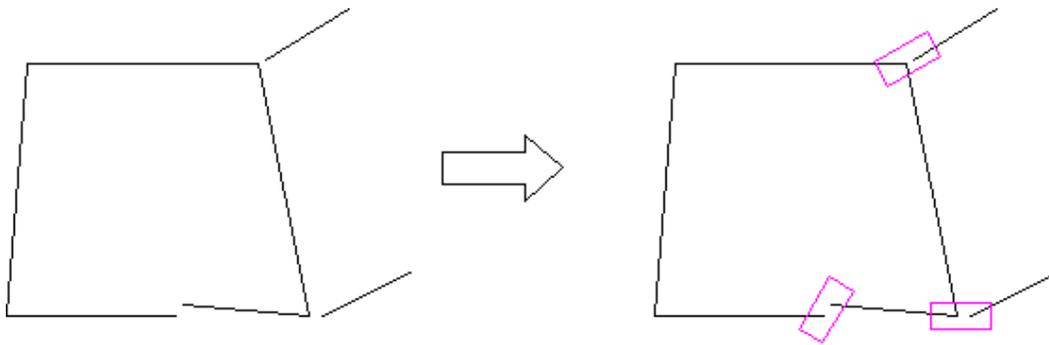
Optionally, by incrementing this number, you can display one "non-trivial" gaps after the other for further examination. For example, you may want to type 1 to box only a single "non-trivial" gap. Then type 2 to box the next one, and so forth.

Optionally, type the total number of "non-trivial" gaps to box all of these gaps at the same time.

- Optionally, close the "trivial" gaps indicated by  **Show Gaps** by clicking  **Confirm**.

Figure 82. Showing Non-Trivial Gaps on page 250 shows an example of a geometry that includes three "non-trivial" gaps.

Figure 82. Showing Non-Trivial Gaps



 **Tip**

To clean up most common problems in a part, click the **Clean** buttons in the following sequence:

 Pnt	Merge disconnected points.
 Gaps	Close gaps between adjoining elements.
 Duplicate	Remove existing duplicates and those caused by merging points.
 Stacked	Split overlapping elements.
 Duplicate	Remove duplicates caused by splitting.
 Staggered	Merge partially overlapping elements.

Further Information

For further technical information about modifying, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- `help modify`
- `help move`
- `help rotate`
- `help mirror`
- `help scale`
- `help similar`
- `help stretch`
- `help isometric`
- `help convert_spline`
- `help bspl_point_length`
- `help trim`
- `help contour`
- `help split`
- `help merge`
- `help clean_drawing`
- `help group`
- `help ungroup`
- `help group_add`
- `help group_remove`

Creo Elements/Direct Drafting displays further information about modifying your drawing.

10

Dimensioning a Drawing

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This chapter describes how to dimension your drawings.

Dimensioning Fundamentals

To clarify how the dimensioning commands work, it is necessary to define the terminology used when discussing dimensioning. Generally, a dimension consists of two parts:

- Dimension Text.
- Dimension Geometry.

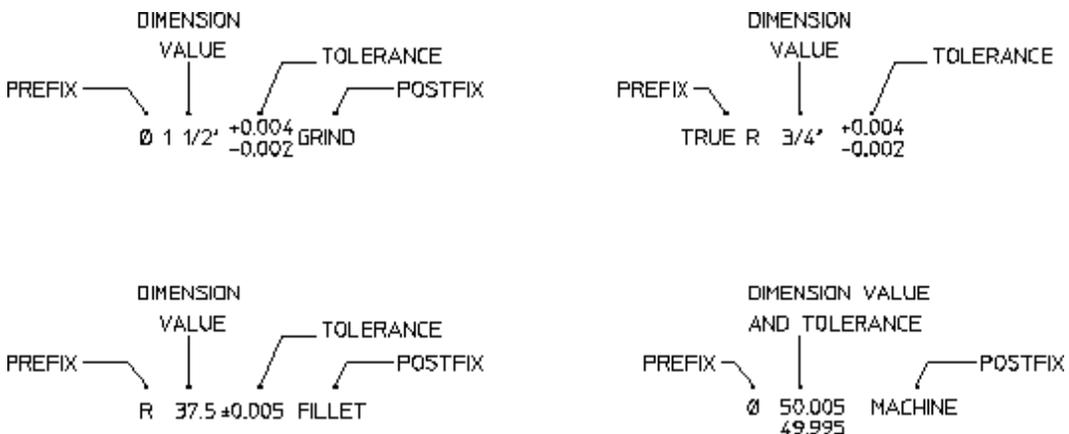
What is Dimension Text?

Dimension text describes the geometry feature that is marked by the dimension geometry. Dimension text is divided into the following fields:

- Prefix
- Dimension Value
- Tolerance
- Postfix
- Subfix
- Superfix

Figure 83 on page 253 shows several examples of dimension text with the fields indicated.

Figure 83. Dimension Text Examples



The fields prefix, tolerance, postfix, subfix and superfix are optional.

What is a Prefix?

Use a prefix to include additional information before the dimension value. Examples of prefixes are: DIA (diameter), TRUE R (true radius), and R (radius).

What is a Dimension Value?

A dimension value is the numerical value assigned to the geometry feature you are dimensioning. The value can be imperial, metric or angular depending on the geometry feature and the standard units you use.

What is a Tolerance?

A tolerance is used to include information about the tolerance allowed in the dimension value. Tolerance is the total amount by which a specific dimension is permitted to vary. Types of tolerances are:

- Limit tolerances
- Plus and minus tolerances
- Upper and lower tolerances

Limit Tolerances

Limit tolerancing is used to give the maximum and the minimum dimension values. The high limit (maximum value) is placed over the low limit (minimum value).

Figure 84 on page 254 shows an example of limit tolerancing.

Figure 84. Limit Tolerancing



Plus and Minus Tolerances

Plus and minus tolerancing is used to place the tolerance to the right of the dimension value as a plus and minus expression of the tolerance.

Figure 85 on page 254 shows an example of plus and minus tolerancing.

Figure 85. Plus and Minus Tolerancing

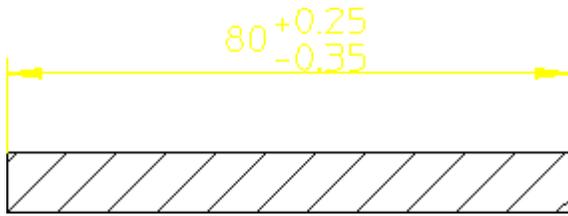


Upper and Lower Tolerances

Upper and lower tolerancing is used to place the tolerances to the right of the specific dimension value. They appear as upper and lower values of the permissible variation of the size of the feature.

Figure 86 on page 255 shows an example of upper and lower tolerancing.

Figure 86. Upper and Lower Tolerancing



What is a Postfix?

Use a postfix to include additional information after the dimension value. Examples of postfix information are: 3 HOLES, EQUALLY SPACED, and CONC (concentric).

What are Subfixes and Superfixes?

You can use subfixes (text below the dimension text) and superfixes (text above the dimension text) to add information such as manufacturing-related specifications to the actual dimension text.

Figure 87 on page 255 shows an example of a dimensioned part complete with dimension text, subfix and superfix.

Figure 87. Dimension Text plus Subfix plus Superfix



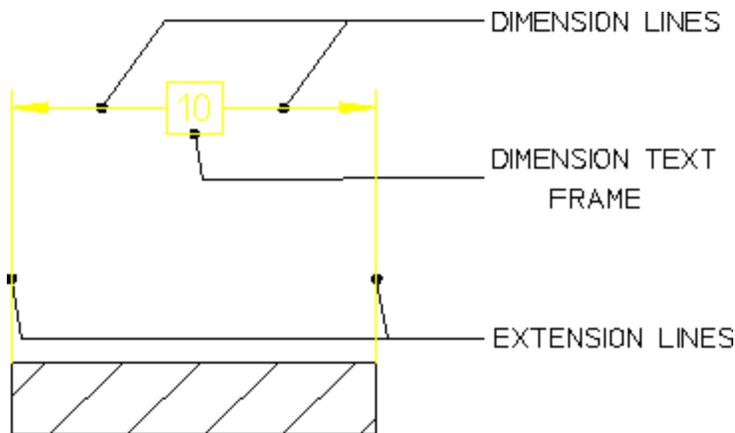
What is a Dimension?

Dimension marks the geometry feature corresponding to the dimension text. Dimension consists of:

- Dimension lines
- Extension lines
- A dimension text frame

Figure 88 on page 256 shows the three components of a dimension.

Figure 88. Dimension



Dimension Lines

Dimension lines show the direction and extent of a dimension. The dimension line may be broken and the dimension text inserted (referred to as on the dimension line), or the line may be a full unbroken line with the dimension text above or below the dimension line. The dimension line will normally end with a terminator.

Figure 89 on page 256 shows dimension line terminators.

Figure 89. Dimension Line Terminators



Extension Lines

Extension lines show the extension of a surface or point to a location outside the part outline. In most technical drawings, you draw them perpendicular to the dimension line with a visible gap from the geometry feature.

Dimension Text Frames

The dimension text frame surrounds all dimension text relating to the dimension geometry. Normally, the frame is switched off. You can make the frame appear as a box-shaped frame or a balloon-shaped frame or a flag-shaped frame by selecting the appropriate option (see Figure 125 on page 321).

Dimension Settings and Dimension Text Distances

As previously described, a dimension is made up of several parts:

- Dimension text
- Dimension lines
- Extension lines

You can set the distance between some of the parts making up a dimension.

Figure 90 on page 257 shows the most common distances that you can set. Each distance is identified by a letter. Refer to Table 1 on page for an explanation of the letters.

Figure 90. Setting Dimension Distances

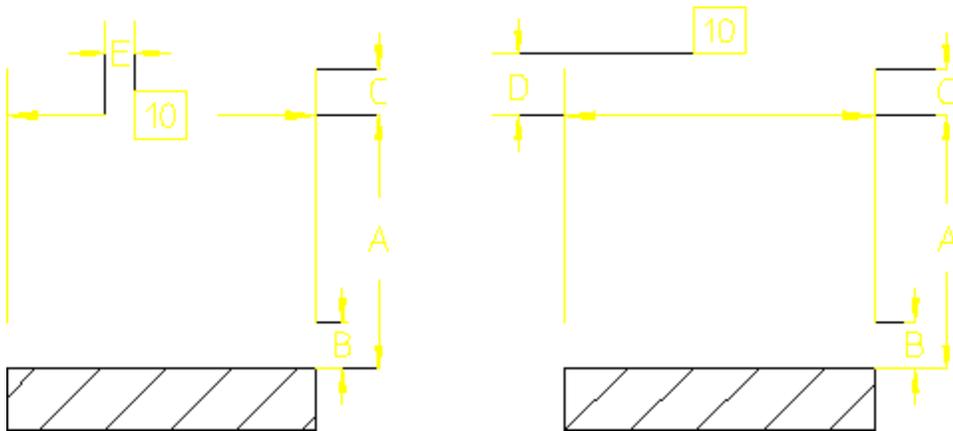


Table 1 on page gives the distance, the corresponding Creo Elements/Direct Drafting function that sets the distance, and a typical value.

Table 1. Setting Dimension Distances

Distance	Description
A	The minimum distance between the dimension line and the geometry feature. Define with the Minimum Space box (in the Line settings of the Dimension Settings dialog box). The default minimum distance is 0 mm.
B	The distance between the extension line and the geometry feature. Define with the Offset Point box (in the Line settings of the Dimension Settings dialog box). The default distance is 0 mm.
C	The distance between the dimension line and the top of the extension line. Define with the Offset Line box (in the Line settings of the Dimension Settings dialog box). The default distance is 2 mm.

Distance	Description
D	The vertical distance between the dimension line and the dimension text. This only works if the location of the dimension text is above or below the dimension line. Define with the Offset Line box (in the Text settings of the Dimension Settings dialog box). The default distance is 2 mm.
E	The gap between the dimension line and the dimension text. This only works if the location of the dimension text is on the dimension line. Define with the Space box (in the Line settings of the Dimension Settings dialog box). The default gap is 2 mm.

Dimensioning With Creo Elements/Direct Drafting

The dimensioning module has the following features to make dimensioning your drawing quick and easy:

- A full-featured user interface that includes:
 - An efficient screen menu system.
 - Combination of dimensioning commands and functions.
 - Integration with Creo Elements/Direct Drafting common and selection commands.
- Powerful dimension assignment features:
 - A range of dimensioning assignment methods.
 - Multiple elements can be dimensioned at once.
 - Selection aids speed the assignment process.
 - Dimensioning between parts and on layers.
- Tools for dimension style setting and modification:
 - Standards-based dimensioning styles.
 - Style saving and re-assignment via the **Style** table.
 - Simultaneous modification of dimensioning geometry and text.

The dimensioning module is sometimes referred to as the Dimension Advisor (DA).

Dimensioning Interface Features

Dimensioning commands and functions can be executed using the ribbon user interface as well as from the command line.

Dimensioning Commands and Functions

The dimensioning interface provides both commands and functions. Commands and functions can be combined to perform complex operations in a short amount of time.

Dimensioning Commands

Generally, a command remains active until:

- You click  **OK** to end the command.
- You end the command by starting another command.

This means you can repeat the command without having to click the command again.

For example:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Linear**.
2. Click  **Single**.
3. Enclose more than one geometry feature in a box and specify the position in the drawing.
4. When you have finished using  **Single**, click  **OK** to end the command.

Dimensioning Functions

Use functions to interrupt commands. Once you have completed the function, Creo Elements/Direct Drafting will return to the active command. You can cancel the function and leave the dimensioning command active by clicking  **Cancel**.

Combining the Dimensioning Commands and Functions

A powerful feature of the Creo Elements/Direct Drafting dimensioning module is the ability to combine dimensioning commands and functions to perform complex operations efficiently.

The following procedure illustrates the importance of this feature.

Figure 91. Combining Dimensioning Commands and Functions



To create the dimension shown in [Figure 91 on page 260](#):

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Circular**.
2. Click  **Diameter**.
3. In the **Orient** gallery, select  **Center Line**.
4. Click **File** ► **Settings** ►  **Dimension**.
The **Dimension Settings** dialog box opens.
5. Click **Text Ext**.
6. In the **Set** box, select **Prefix**.
7. Type DIA in the **Ext Text** box.
8. Type an arbitrary name for the prefix (such as p1) in the entry field above the (originally empty) prefix list and click **Add**.
9. In the **Set** box, select **Postfix**.
10. Type GRIND in the **Ext Text** box.
11. In the **Set** box, select **Tolerance**.
12. In the **Display** box, select **Upper/Lower**.
13. In the **Upper** box, type +0 . 004 as the upper tolerance value.
14. In the **Lower** box, type –0 . 000 as the lower tolerance value.
Use the same procedure as above to add the tolerance values to your dimension text extensions.
15. Click the circle.
16. Click the desired location for the dimension text.

Remember that  **Diameter** and the tolerancing values remain active. This means that you can continue to apply diameter dimensions to other circular geometry features simply by clicking them.

Suppose that you do not want the GRIND postfix to appear in subsequent dimensions. You can remove this postfix without ending the current command by using the **Clear** function:

1. Click **Clear**.
2. Click another circular element (circle, arc, fillet).
3. Specify the desired location for the dimension text.

As you can see, combining dimensioning commands and functions reduces the amount of time needed to dimension your drawing.

 **Note**

All the examples in this chapter assume that the current linear units are

millimeters and the angular units are degrees. Check the **Format** tab for 

mm and  **Degree**.

Using the Common and Selection Commands

You can use Creo Elements/Direct Drafting common and selection commands with the dimensioning commands. For more information, refer to [Commonly Used Commands on page 72](#).

Dimension Assignment Features

Multiple Assignment Methods

The dimensioning module provides several methods for assigning dimensions to drawing elements.

- With direct assignment, you manually select the type of dimension you wish to assign (single, chain, datum, angle, etc.) and then identify one or more drawing elements to receive the selected dimension. For example:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Linear**.
2. Click  **Single**.
3. Click the dimension points. To dimension a single geometric feature, click two dimension points to identify the feature. To dimension several drawing elements at the same time, enclose them in a box.

-
4. Move the dimension trace around your drawing until the extension lines and dimension text are at the desired location.
 5. Specify the point where you want to place the dimension text and Creo Elements/Direct Drafting draws the dimension or dimensions.

Direct dimensioning gives you the greatest control over the dimensioning process, but is likely to take the most time.

- Geometry-sensing assignment is provided by the commands `DA_DIM_GEO_SENSE` and `DA_DIM_PD_SCAN`. `DA_DIM_GEO_SENSE` prompts you to identify a group of drawing elements and then generates a basic set of dimensions for those elements based on their geometry type and position. `DA_DIM_PD_SCAN` also generates dimensions for a selected set of elements, but it scans the parametric design constraints on the selected elements to determine what type of dimensions to generate.

Geometry-sensing assignment is not intended to provide "automatic" dimensioning, but it does provide a basic dimensioning framework that you can modify to suit your requirements.

To quickly understand the operation of geometry-sensing assignment, try the following:

1. Create a drawing with several different drawing elements.
 2. Type `DA_DIM_GEO_SENSE ALL` in the user input line and press ENTER.
Creo Elements/Direct Drafting creates dimensioning for all elements in the drawing.
 3. Type `DA_MOVE_DIMENSION` in the user input line and press ENTER.
You can now manually adjust the positions of the newly-created dimensions.
- Database-matching assignment transfers dimensioning from a "dimensioning database" of pre-dimensioned parts to similar drawing elements in your drawing. Once you create this database, you can use the entries in it as dimensioning templates to quickly generate complete dimensioning for similar parts or features in you drawings.

Database-matching assignment can greatly improve dimensioning speed and accuracy for commonly used drawing details.

Dimension Single or Multiple Elements

You can assign dimensions to one or several elements at a time. You assign single dimensions by indicating the individual dimension points you wish to dimension.

To assign multiple dimensions, you select whole elements, either with  **Select** or by enclosing them in a box.

Selection Aids

Several selection aids are available to help you assign dimensions:

- When dimensioning single elements, you do not have to precisely identify the end points of each element. You can click two points on the element that are close to each end point and the cursor will catch to the end points automatically.
- When selecting multiple elements for dimensioning, a filter can be applied to the selection process to selectively exclude items from being considered. Geometry can be filtered based on geometry type, line slope, linetype, pensize, color, and/or linesize. You set up the filter using the **Filter** options in the **Dimension Settings** dialog box. The current filter can be enabled or disabled by clicking or clearing the **On** check box in the **Filter** settings in the **Dimension Settings** dialog box.
- A dimension trace appears after you have identified the dimension points of the geometry features you want to dimension. Creo Elements/Direct Drafting then displays a movable dimension trace which shows:
 - The orientation and location of the dimension(s).
 - The location of the dimension line terminators.
 - The length of extension lines.
 - The dimension text represented by a box.

When multiple elements are selected, the dimensioning trace shows the proposed dimensioning for all the selected elements, allowing you to place the dimensioning text for all of them with a single click.

Dimensioning Between Parts

To dimension between parts, ensure that the parent part is active. You can then dimension between the parent part and one of its subparts, or between pairs of subparts.

Dimensioning Detail Parts

If you are dimensioning a detail part, you must make it the active part. Otherwise, Creo Elements/Direct Drafting produces incorrect dimensions.

If you attempt to dimension a detail part that is not also the active part, Creo Elements/Direct Drafting will display the following message in the prompt line:
Warning; Detail part is not active. Dimension value reflects the drawing scale.

To make a detail part active:

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.
2. Click  **Part**.
3. Click the detail part.

Dimensioning on Layers

Dimensioning on layers is the same as normal dimensioning.

For more information about layers, refer to [Using Layers and Infos on page 510](#).

Style Features

Creo Elements/Direct Drafting provides several methods for setting and changing dimensioning styles. In addition to specific commands that allow you to set and change dimension text attributes (size, font, orientation, spacing, etc.) and line attributes (color, width, spacing, offsets, terminators, etc.), Creo Elements/Direct Drafting also provides tools for emulating dimensioning standards and for storing and recovering often-used styles.

Standards

Dimensioning standards allow you to quickly set the Creo Elements/Direct Drafting dimension style features to emulate popular dimensioning standards, such as ANSI, DIN, JIS, and ISO. A generic "company" standard may also be

defined. Standards are easily selected using  **Dimension Standards** in the **Format** group on the **Drafting** tab. Creo Elements/Direct Drafting displays the current dimensioning standard in the **Format** group.

Style Table

The **Dimension Style** table allows you to store the current dimensioning style attributes into a table entry. These attributes can later be recovered simply by clicking the table entry. Once recovered, these settings become the current dimensioning style and will be used for subsequent dimensions.

Modification Features

Dimension modification features allow you to change the layout of existing dimensioning lines and text. Dimension modification buttons are provided in the **Annotate** group on the **Modify** tab.

Preparing to Dimension

Before you assign dimensions to your drawing, you may want to set up the basic dimensioning style that will be used for your dimensions.

You can manually set dimensioning attributes using the **Dimension Settings** dialog box. In addition, you can set up dimensioning to emulate a particular standard.

Selecting a Dimensioning Standard

Creo Elements/Direct Drafting can emulate the dimensioning styles specified in various drawing standards as well as allow you to define your own company or site-wide standard. The standards available are:

-  **ANSI**
-  **ISO**
-  **DIN**
-  **JIS**
-  **Company** (Creo Elements/Direct Drafting default)

To select a dimensioning standard:

1. Click **Drafting** and then, in the **Format** group, click  **Dimension Standards**.
2. Click the dimensioning standard you want to use.

Note

To emulate a given dimensioning standard, Creo Elements/Direct Drafting simply executes a macro that calls dimensioning style commands with the appropriate options. If a given dimensioning standard has features that are not supported by Creo Elements/Direct Drafting dimensioning commands, that feature will not be supported.

The dimensioning standards are all defined in the macro file `da_std.mac`. This macro file can be edited by hand in order to set the Company dimensioning style, but the other standards defined in this file should generally not be changed.

Please ensure automatic dimension placement by clicking the **Automatic** check box in the **Options** settings of the **Dimension Settings** dialog box. Clear the **Automatic** check box for manual placement.

You may also want to select or specify your own dimension settings before you begin to dimension. For details see:

- [Dimension Line Settings on page 308](#)
- [Dimension Text Settings on page 318](#)
- [Predefining Prefixes, Postfixes, Superfixes, Subfixes, and Tolerances on page 324](#)
- [Dimension Unit Settings on page 329](#)
- [Selection Filtering on page 332](#)
- [Dimension Options on page 337](#)

Single Dimensioning

This section describes how to dimension single linear geometry features.

Creo Elements/Direct Drafting allows you to define the orientation of the single dimension relative to a geometry feature using the buttons of the **Orient** gallery.

All buttons enable you to apply dimensioning to a single linear drawing element or to a group of elements depending on your selection. You can:

- Click any two dimension points. The line between the two points is dimensioned.
- Click a single linear drawing element. The starting point of the selected element becomes the first dimension point. Select a second dimension point on this or any other linear drawing element to complete the dimension.

- Select multiple linear drawing elements with  **Select** or by enclosing them in a box. The end points of each selected element become dimension points, and each element is dimensioned.

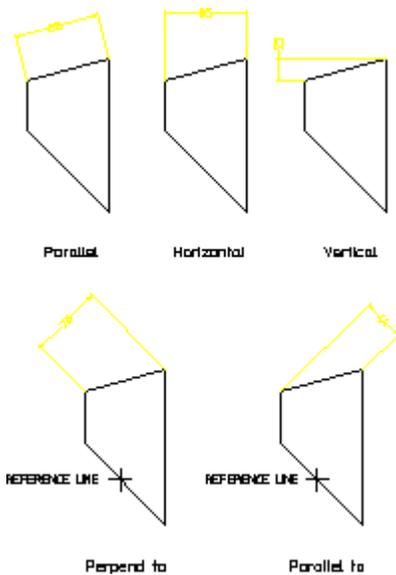
If you clicked the **Automatic** check box (automatic dimension displacement) in the **Options** settings of the **Dimension Settings** dialog box, Creo Elements/Direct Drafting positions the new dimensions automatically. **Automatic** placement only applies when you have selected multiple elements for dimensioning.

If you cleared the **Automatic** check box (automatic dimension displacement) in the **Options** settings of the **Dimension Settings** dialog box, position the dimensions on your drawing manually. A dimension trace appears when you select elements. Move the trace around your drawing until the dimension text is in the desired location. Click the desired text location and Creo Elements/Direct Drafting draws the dimension or dimensions. When multiple elements are selected, the trace shows the proposed dimensions for all of them at the same time.

Example

Figure 92 on page 267 shows the result of using each single dimensioning option.

Figure 92. Single Dimensioning – Options



To dimension single linear geometry features:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Linear**.
2. Click  **Single** and then, click the required **Orient** button in the gallery and perform the associated procedure:

Orient Button	Purpose	Procedure
 Parallel	Draw a dimension parallel to a line drawn between two dimension points (along an inclined plane).	<ol style="list-style-type: none"> Click the dimension points or elements to be dimensioned. Specify the location of the dimension. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, position the dimension on your drawing manually. Click  OK.
 Horizontal	Draw a dimension in the horizontal plane.	<ol style="list-style-type: none"> Click the dimension points or elements to be dimensioned. Specify the location of the dimension. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, position the dimension on your drawing manually. Click  OK.
 Vertical	Draw a dimension in the vertical plane.	<ol style="list-style-type: none"> Click the dimension points or elements to be dimensioned.

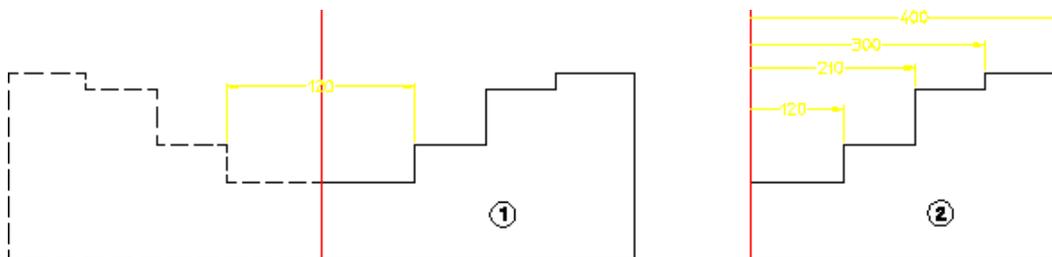
Orient Button	Purpose	Procedure
		<ul style="list-style-type: none"> b. Specify the location of the dimension. c. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, position the dimension on your drawing manually. d. Click <input checked="" type="checkbox"/> OK.
 Perpendicular to Line	Draw a dimension perpendicular to a reference line.	<ul style="list-style-type: none"> a. Click a reference line. b. Click the dimension points or elements to be dimensioned. c. Specify the location of the dimension. d. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, position the dimension on your drawing manually. e. Click <input checked="" type="checkbox"/> OK.

Orient Button	Purpose	Procedure
 Parallel to Line	Draw a dimension parallel to a reference line.	<ol style="list-style-type: none"> Click a reference line. Click the dimension points or elements to be dimensioned. Specify the location of the dimension. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, position the dimension on your drawing manually. Click  OK.

Symmetry-Style Dimensions

Creo Elements/Direct Drafting allows you to specify symmetry-style dimensions using  **Symmetric Single** and  **Symmetric Long**. [Figure 93 on page 270](#) shows an example of symmetric dimensioning. In drawing 2, the first dimension point is assumed to be on a symmetry line. No extension line or line terminator is drawn for that element, and the dimension text value is doubled to indicate that the dimension represents both sides of the symmetry line. Drawing 1 shows the unseen half of the geometry.

Figure 93. Symmetry-Style Dimensioning



To create a symmetry-style dimension:

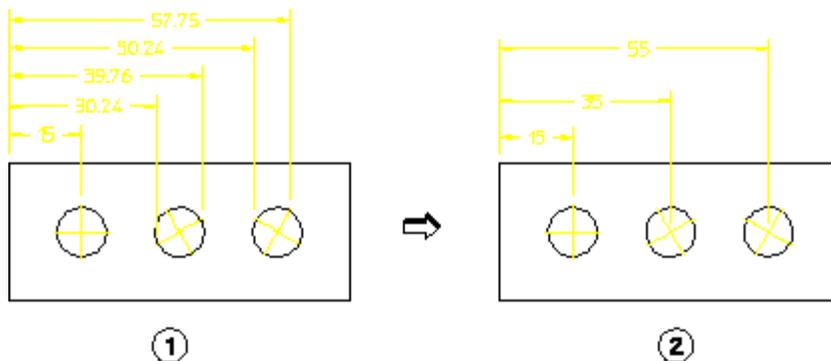
1. Click **Drafting** and then in the **Annotate** group, click the arrow next to  **Linear**.
2. Click the required **Linear** button and perform the associated procedure.

Linear Button	Purpose	Procedure
 Symmetric Single	Draw a (single) symmetric dimension.	<ol style="list-style-type: none"> a. Click the first dimension point. b. Click the second dimension point.
 Symmetric Long	Draw a (datum) symmetric dimension.	<ol style="list-style-type: none"> a. Click the first dimension point. b. Click the second dimension point. c. Continue clicking dimension points as required.

Centerline Dimensions

When dimensioning geometry contains centerlines, the dimension extension line is attached to the model point (center point) of the centerline. This is true even if the centerlines are not perpendicular to the measured direction of the dimensions.

Figure 94. Dimensions and Centerlines



Modify a symmetry or center line

Use the handles of a symmetry or center line to modify the line.

Extend the symmetry/center line	Click the handle and drag.
Rotate the center line	Click the center line (not on the handles) and drag.

To change the **Handles** settings:

1. Click **File** ► **Settings** ►  **System**.
The **System** dialog box opens.
2. Click **Handles**.

Chain Dimensioning

This section describes how to dimension groups of linear geometry features.

Creo Elements/Direct Drafting draws the chain dimensions parallel to the first dimension and chained to the same axis. Locate your first chain dimension so that subsequent chain dimensions do not overwrite the drawing or dimensions. You

can correct mistakes by repeatedly clicking  **Undo**, which removes preceding dimensions. You can then click the correct points.

Creo Elements/Direct Drafting allows you to define the orientation of chain dimensions relative to geometry features using the buttons of the **Orient** gallery.

All buttons enable you to apply dimensioning to a single linear drawing element or to a group of elements depending on your selection. You can:

- Click any two dimension points. The line between the two points is dimensioned.
- Click a single linear drawing element. The starting point of the selected element becomes the first dimension point. Select a second dimension point on this or any other linear drawing element to complete the dimension.
- Select multiple linear drawing elements with  **Select** or by enclosing them in a box. The end points of each selected element become dimension points, and each element is dimensioned.

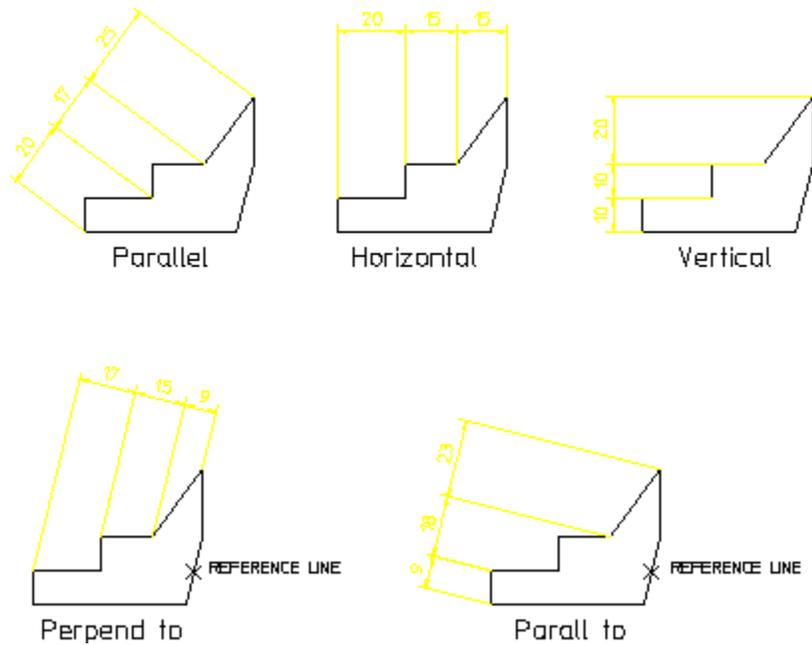
If you clicked the **Automatic** check box (automatic dimension displacement) in the **Options** settings of the **Dimension Settings** dialog box, Creo Elements/Direct Drafting positions the new dimensions automatically. **Automatic** placement only applies when you have selected multiple elements for dimensioning.

If you cleared the **Automatic** check box (automatic dimension displacement) in the **Options** settings of the **Dimension Settings** dialog box, position the dimensions on your drawing manually. A dimension trace appears when you select elements. Move the trace around your drawing until the dimension text is in the desired

location. Click the desired text location and Creo Elements/Direct Drafting draws the dimension or dimensions. When multiple elements are selected, the trace shows the proposed dimensions for all of them at the same time.

Figure 95 on page 273 shows examples of chain dimensioning using each option.

Figure 95. Chain Dimensioning



To draw chained dimensions:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Linear**.
2. Click  **Chain** and then, click the required **Orient** button in the gallery and perform the associated procedure.

Orient Button	Purpose	Procedure
 Parallel	Draw chain dimensions along an inclined plane.	<ol style="list-style-type: none"> a. Click the first two dimension points. These define the angle for this and subsequent chain dimensions. b. If your initial selection contains multiple drawing elements, you are prompted to identify one of them to provide the reference angle for the chain. c. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance. d. To continue the chain along the inclined plane, select additional linear elements. e. Click  OK.
 Horizontal	Draw chain dimensions in the horizontal plane.	<ol style="list-style-type: none"> a. Click the dimension points or elements to be dimensioned. If your selection contains multiple

Orient Button	Purpose	Procedure
		<p>drawing elements, they become the first members in the chain.</p> <p>b. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.</p> <p>c. To continue the chain, select additional linear elements.</p> <p>d. Click  OK.</p>
 Vertical	Draw chain dimensions in the vertical plane.	<p>a. Click the dimension points or elements to be dimensioned. If your selection contains multiple drawing elements, they become the first members in the chain.</p> <p>b. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box,</p>

Orient Button	Purpose	Procedure
		<p>click the desired location for the dimension text or texts, using the dimension trace for guidance.</p> <p>c. To continue the chain, select additional linear elements.</p> <p>d. Click  OK.</p>
 Perpendicular to Line	<p>Draw a dimension chain perpendicular to a given line.</p>	<p>a. Click a reference line. You can use any line on the drawing as the reference line.</p> <p>b. Click the dimension points or elements to be dimensioned. If your selection contains multiple drawing elements, they become the first members in the chain.</p> <p>c. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.</p>

Orient Button	Purpose	Procedure
		<p>d. To continue the chain, select additional linear elements.</p> <p>e. Click  OK.</p>
 Parallel to Line	<p>Draw a dimension chain parallel to a reference line.</p>	<p>a. Select a reference line. You can identify any line on the drawing as the reference line.</p> <p>b. Click the dimension points or elements to be dimensioned. If your selection contains multiple drawing elements, they become the first members in the chain.</p> <p>c. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.</p> <p>d. To continue the chain, select additional linear elements.</p> <p>e. Click  OK.</p>

Datum Dimensioning

Datum dimensioning is useful for marking holes and other features on a part. The dimensions are parallel to each other (like chain dimensioning), and have a common datum point. Normally, design engineers use this type of dimensioning for parts that will be NC machined.

Creo Elements/Direct Drafting allows you to define the orientation of datum dimensions relative to geometry features using the buttons of the **Orient** gallery.

All buttons enable you to apply dimensioning to a single linear drawing element or to a group of elements.

If you clicked the **Automatic** check box (automatic dimension displacement) in the **Options** settings of the **Dimension Settings** dialog box, Creo Elements/Direct Drafting positions the new dimensions automatically. **Automatic** placement only applies when you have selected multiple elements for dimensioning.

If you cleared the **Automatic** check box (automatic dimension displacement) in the **Options** settings of the **Dimension Settings** dialog box, position the dimensions on your drawing manually. A dimension trace appears when you select elements. Move the trace around your drawing until the dimension text is in the desired location. Click the desired text location and Creo Elements/Direct Drafting draws the dimension or dimensions. When multiple elements are selected, the trace shows the proposed dimensions for all of them at the same time.

Creating Datum Dimensions

To create a datum dimension:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Linear**.
2. Click the required **Linear** button:
 -  **Datum Long** to draw dimension using long base lines.
 -  **Datum Short** to draw dimensions using short base lines.
 -  **Coordinate** to draw dimensions without dimension lines. This is useful if the geometry of the part will cause the dimension lines to be too close together.
3. Click the required **Orient** button in the gallery and perform the associated procedure.

 Parallel	<p>Draw datum dimension along an inclined plane.</p>	<ol style="list-style-type: none"> Click a single dimension point. This becomes the common datum point. Click one or more additional dimensioning points. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance. To add more dimensions relative to the common datum point, select additional drawing elements. Click  OK.
 Horizontal	<p>Dimension in the horizontal plane. You can dimension in the horizontal plane irrespective of the position of the dimension points.</p>	<ol style="list-style-type: none"> Click a single dimension point. This becomes the common datum point. Click one or more additional dimensioning points. If you have cleared the Automatic check box (automatic

		<p>dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.</p> <p>d. To add more dimensions relative to the common datum point, select additional drawing elements.</p> <p>e. Click  OK.</p>
 Vertical	<p>Dimension in the vertical plane.</p>	<p>a. Click a single dimension point. This becomes the common datum point.</p> <p>b. Click one or more additional dimensioning points.</p> <p>c. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.</p> <p>d. To add more</p>

		<p>dimensions relative to the common datum point, select additional drawing elements.</p> <p>e. Click <input checked="" type="checkbox"/> OK.</p>
<p> Perpendicular to Line</p>	<p>Draw a perpendicular datum dimension.</p>	<p>a. Click a reference line. You can use any line on the drawing as the reference line.</p> <p>b. Click a single dimension point. This becomes the common datum point.</p> <p>c. Select one or more additional dimensioning points.</p> <p>d. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.</p> <p>e. To add more dimensions relative to the common datum point, select additional drawing elements.</p> <p>f. Click <input checked="" type="checkbox"/> OK.</p>

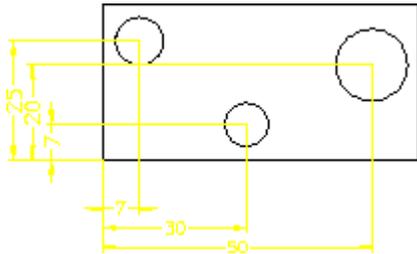
 Parallel to Line	<p>Draw a datum dimension parallel to a reference line.</p>	<ol style="list-style-type: none"> a. Click a reference line. You can use any line on the drawing as the reference line. b. Click a single dimension point. This becomes the common datum point. c. Select one or more additional dimensioning points. d. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance. e. To add more dimensions relative to the common datum point, select additional drawing elements. f. Click  OK.
--	---	--

Datum Dimension Examples

Example 1

Figure 96 on page 283 shows a rectangular plate dimensioned using the  **Datum Long** and the  Parallel to Line orientation button.

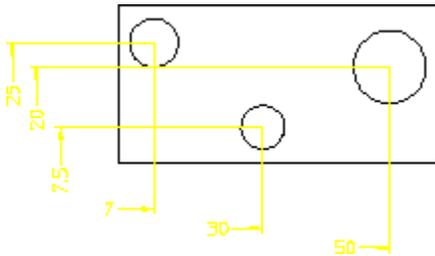
Figure 96. Rectangular Plate — Dimensioned with Datum Long



Example 2

Figure 97 on page 283 shows a rectangular plate dimensioned using  **Datum Short** with the  Parallel and  Vertical orientation button.

Figure 97. Rectangular Plate — Dimensioned with Datum Short

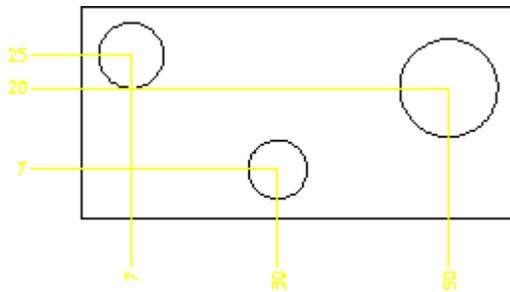


Example 3

The following example shows how to use  **Coordinate** to dimension a simple rectangular plate with three holes. Assume that the common datum point is the lower left hand corner of the plate.

Figure 98 on page 284 shows the rectangular plate:

Figure 98. Rectangular Plate — Dimensioned with Coordinate



To add the dimensions measured along the vertical (from datum point):

1. Ensure that the **Vertex** check box is clicked in the **Show** dialog box.
2. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Linear**.
3. Click  **Coordinate**.
4. In the **Orient** gallery, click  Vertical.
5. Click the lower left hand corner of the plate.
6. Click the center of one of the circles.
7. Click the position for the dimension text.
8. Repeat step 6 for the other two circles.

To add the dimensions measured along the horizontal (from datum point):

9. Click  **Coordinate**.
10. In the **Orient** gallery, click  Horizontal.
11. Click the lower left hand corner of the plate.
12. Click the center of one of the circles.
13. Click the position for the dimension text.
14. Repeat step 12 for the other two circles.

 **Note**

The dimension text may be preceded by a leading zero. For details see [Coordinate Dimension Leading Zero on page 323](#) in this chapter.

Radius, Diameter, Tangential and Arc Dimensioning

Dimensioning Radii

This section shows how to dimension the radius of an arc, circle, BSpline or fillet feature.

- Radius dimension text includes the prefix R (radius).
- You can locate the dimension text either inside or outside the feature.
- When the radius dimension is placed at a position where the dimension line does not hit the dimensioned geometry then the system automatically creates an arc extension for the dimension.

To dimension a radius:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Circular**.
2. Click  **Radius** and then click the required **Orient** button in the gallery:
 -  **Center Line** to dimension the radius of a feature with the dimension line drawn from the feature center.
 -  **No Center Line** to dimension the radius of a feature with the dimension line drawn towards the center of the feature but not connecting with the center.

Figure 99. Dimensioning Radii with and without a Centerline



3. Select one or more arc, circle, or fillet features.
4. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.
5. Click  **OK**.

 **Note**

To dimension a BSpline, you must select a point on the BSpline. The radius dimension value is valid at the selected point; the dimension value may be different at different points.

Dimensioning using Tangents

Tangential dimensions are useful for dimensioning geometry where vertices are not available. This section shows how to create tangential dimensions on the following elements:

- Circle
- Arc
- Fillet
- Ellipse
- B-spline (closed, both interpolation and control)

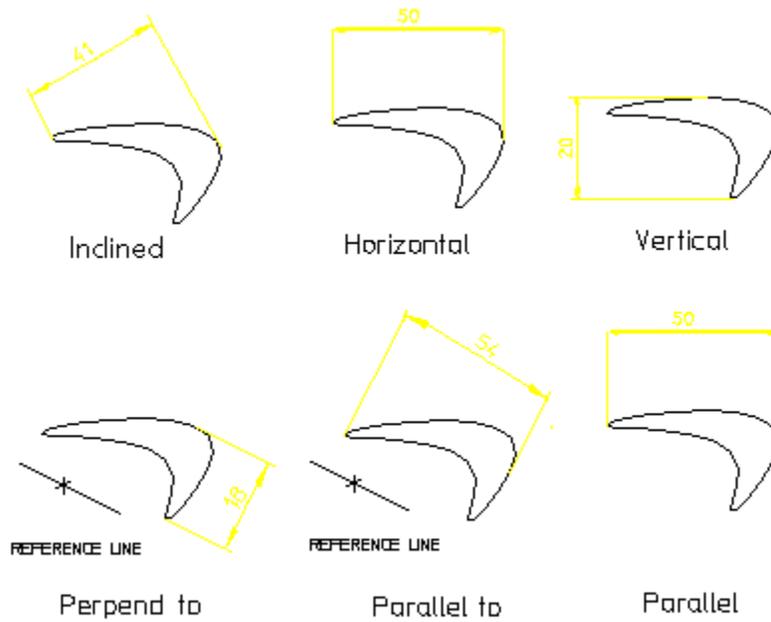
 **Note**

Tangential dimensions are not possible for open B-splines.

Creo Elements/Direct Drafting always chooses the extreme tangent points of a closed B-spline for dimensioning, for the B-spline with more than two tangent points at the given angle. If you select the points on a closed B-spline and a circle, arc, line, fillet or an ellipse, Creo Elements/Direct Drafting automatically chooses the tangent point of the B-spline which is closest to the selected point on that bspline.

The following figure shows examples of tangential dimensioning on a closed B-spline using each option.

Figure 100. Tangential Dimensioning



To create tangential dimensions:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Circular**.
2. Click  **Tangential** and then, click the required **Orient** button in the gallery and perform the associated procedure.

Orient Button	Purpose	Procedure
 Inclined	Draw tangential dimensions in an inclined plane.	<ol style="list-style-type: none"> a. Select the two elements to be dimensioned. The dimension appears at the cursor. b. Type the inclination angle of the dimension in the user input line and press ENTER or move the cursor to define the angle. c. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance. d. Click  OK. <p> Note You can specify any value of the inclination angle, Creo Elements/ Direct Drafting converts the angle between 360 and -360 degrees.</p>
 Parallel	Draw a tangential dimension that is parallel to a line through	<ol style="list-style-type: none"> a. Select the two elements to be dimensioned. The

Orient Button	Purpose	Procedure
	the tangent points.	<p>dimension appears at the cursor.</p> <p>b. Click <input checked="" type="checkbox"/> OK.</p>
 Horizontal	Draw tangential dimensions in the horizontal plane.	<p>a. Select the two elements to be dimensioned. The dimension appears at the cursor.</p> <p>b. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.</p> <p>c. Click <input checked="" type="checkbox"/> OK.</p>
 Vertical	Draw tangential dimensions in the vertical plane.	<p>a. Select the two elements to be dimensioned. The dimension appears at the cursor.</p> <p>b. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or</p>

Orient Button	Purpose	Procedure
		<p>texts, using the dimension trace for guidance.</p> <p>c. Click <input checked="" type="checkbox"/> OK.</p>
 Perpendicular to Line	<p>Draw a tangential dimension perpendicular to a given line.</p>	<p>a. Select a reference line in the viewport. You can select any line on the drawing as the reference line.</p> <p>b. Select the two elements to be dimensioned. The dimension appears at the cursor.</p> <p>c. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.</p> <p>d. Click <input checked="" type="checkbox"/> OK.</p>

Orient Button	Purpose	Procedure
 Parallel to Line	Draw a tangential dimension parallel to a reference line.	<ol style="list-style-type: none"> a. Select a reference line in the viewport. You can select any line on the drawing as the reference line. b. Select the two elements to be dimensioned. The dimension appears at the cursor. c. If you have cleared the Automatic check box (automatic dimension placement) in the Options settings of the Dimension Settings dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance. d. Click  OK.

Dimensioning a Diameter

This section shows how to dimension the diameter of a:

- Circle
- Arc
- Fillet feature

You can locate the dimension text either inside or outside the diameter.

To dimension a circle, arc or fillet:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Circular**.

2. Click  **Diameter** and then click the required **Orient** button in the gallery:

- Click  **Center Line** to dimension the diameter of a feature with the dimension line drawn from the feature center.
- Click  **No Center Line** to dimension the diameter of a feature with the dimension line drawn towards the center of the feature but not connecting with the center.

Figure 101. Dimensioning Circles with and without a Centerline



3. Select one or more circle, arc, or fillet features.
4. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.
5. Click  **OK**.

Dimensioning Arcs

Creo Elements/Direct Drafting can measure the length or angle of the major or minor arc. When dimensioning arcs singly, you determine which measurement is used by selecting either the start or end point of the arc first. When dimensioning multiple arcs at once, the length or angle of the major arc will be measured. Creo Elements/Direct Drafting measures arcs in a counterclockwise direction. Dimensioning the length or angle between two arcs is also possible, but only for arcs with the same radius and center.

You can locate the dimension either inside or outside the arc.

To dimension an arc using  **Arc Length**:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Circular**.
2. Click  **Arc Length**.
3. Select the arc features to be dimensioned. You can either:

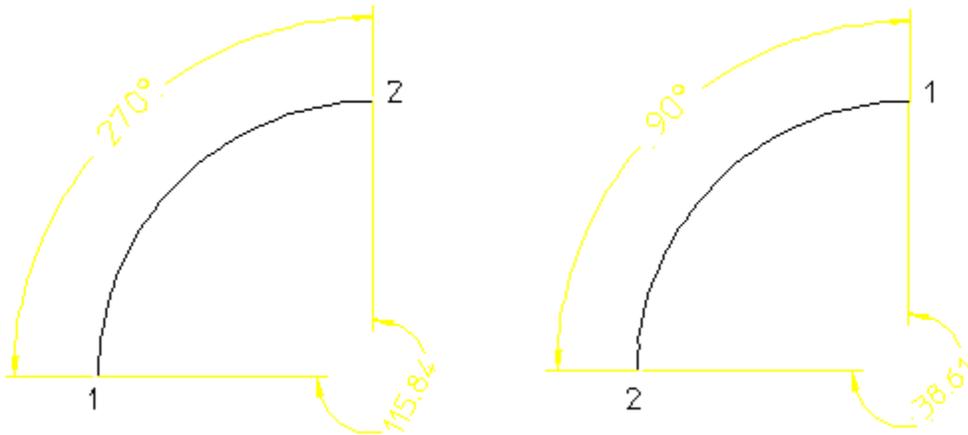
-
- Identify the first and second end points on individual arcs. The order in which you select the endpoints determines whether the major or minor arc is dimensioned.
 - Identify one or more arc elements using the **Select** window or by enclosing them in a box. The major arc will be dimensioned for all selected arcs.
4. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.
 5. Click  **OK**.

To dimension an arc using  **Arc Angle**:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Angular**.
2. Click  **Arc Angle**.
3. Select the arc features to be dimensioned. You can either:
 - Identify the first and second end points on individual arcs. The order in which you select the endpoints determines whether the major or minor arc is dimensioned.
 - Identify one or more arc elements using the **Select** window or by enclosing them in a box. The major arc will be dimensioned for all selected arcs.
4. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.
5. Click  **OK**.

Figure 102 on page 294 shows two possible results of length and angle dimensions obtained using **Arc Dimension** UI. Each arc end point is marked in the order in which it has been identified.

Figure 102. Dimensioning Arcs — Two Results



Angle and Chamfer Dimensioning

Dimensioning Angles

During your work you will need to dimension angles between lines. Creo Elements/Direct Drafting measures an angle depending on the order in which you identify the lines making up the angle. Therefore, there are two possible results for each **Angle** button. Creo Elements/Direct Drafting measures angles in a counterclockwise direction.

To dimension an angle:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Angular**.
2. Click  **Angle**.
3. In the **Orient** gallery, select the required button.
4. Click the two lines that make up the angle.
5. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.
6. Click  **OK**.

The buttons in the **Orient** gallery are:

-  Direct
-  Opposite
-  Adjacent Negative
-  Adjacent Positive

For each of the above button you can achieve two results.

Figure 103 on page 295 shows how you identify the angle lines in the opposite order to achieve a different result.

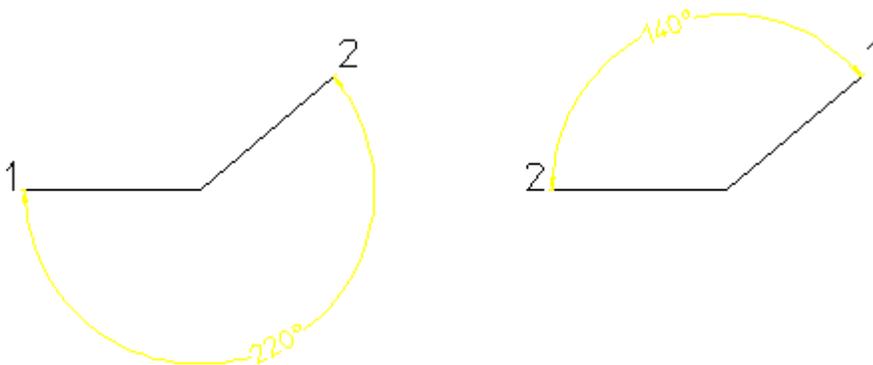
Figure 103. Identifying Angle Lines



Between Two Lines

Figure 104 on page 295 shows the two possible results.

Figure 104. Angle Between Two Lines —Two Results



To dimension the angle between two lines:

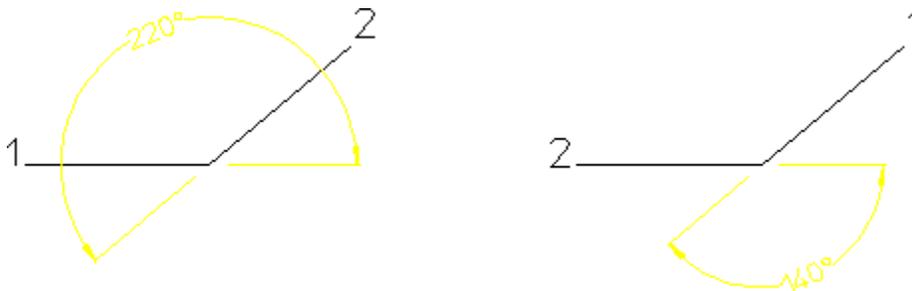
1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Angular**.
2. Click  **Angle**.
3. In the **Orient** gallery, select  **Direct**.
4. Select the two lines making up the angle (the angle dimensioned depends upon which of the two lines you identify first).

5. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.

Between Intersections

Figure 105 on page 296 shows the two possible results.

Figure 105. Angle Between Intersections — Two Results



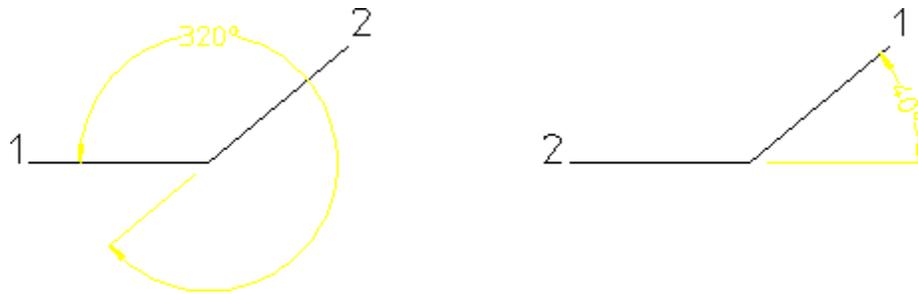
To dimension the angle between the projected intersection of two lines:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Angular**.
2. Click  **Angle**.
3. In the **Orient** gallery, select  **Opposite**.
4. Click the two lines making up the angle (the angle dimensioned will depend upon which of the two lines you identify first).
5. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.

Between First and Second Line

Figure 106 on page 297 shows the two possible results.

Figure 106. Angle Between First and Second Line — Two Results



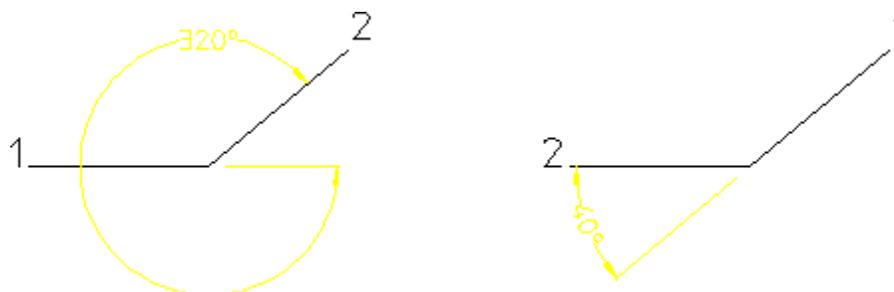
To dimension the angle between the first line you clicked and the projection of the second line:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Angular**.
2. Click  **Angle**.
3. In the **Orient** gallery, select  **Adjacent Negative**.
4. Click the two lines making up the angle (the angle dimensioned depends upon which of the two lines you identify first).
5. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.

Between Second and First Line

Figure 107 on page 297 shows the two possible results.

Figure 107. Angle Between Second and First Line — Two Results



To dimension the angle between the second line you clicked and the projection of the first line:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Angular**.

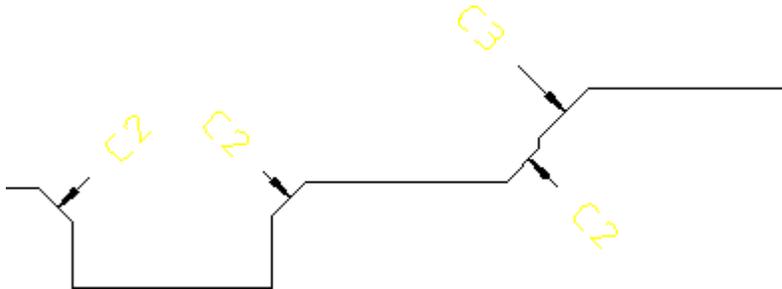
2. Click  **Angle**.
3. In the **Orient** gallery, select  **Adjacent Positive**.
4. Click the two lines making up the angle (the angle dimensioned depends upon which of the two lines you identify first).
5. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.

Dimensioning a Chamfer

Chamfered surfaces can be dimensioned using  **Chamfer** under  **Line** in the **Annotate** group on the **Drafting** tab. (This feature is required in certain countries to meet recognized design standards.)

By default, the chamfer dimension is preceded by the character C. Only chamfered surfaces created with the chamfer option **Vertex** or set at an angle of 45° can be dimensioned.

Figure 108. Dimensioning a Chamfer



To dimension a chamfer:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Linear**.
2. Click  **Chamfer**.
3. Click the chamfer you want to dimension.
4. If you have cleared the **Automatic** check box (automatic dimension placement) in the **Options** settings of the **Dimension Settings** dialog box, click the desired location for the dimension text or texts, using the dimension trace for guidance.
5. Click  **OK**.

 **Note**

- You can use DIM_CHAMFER_TEXT_CONTROL and DIM_CHAMFER_PREFIX_STRING to change the default chamfer prefix string.
 - If you stretch a chamfer or its reference geometry, the chamfer reference geometry is updated. However, the chamfer dimensions are not updated. The chamfer dimensions are not visible after you stretch the chamfer.
-

Geometry-Sensing Assignment

The  **Geo Sense** and  **Scan** buttons in the **Annotate** group on the **Drafting** tab quickly generate basic dimensioning for a selected set of drawing elements based on their geometry types, positions, and parametric constraints, if any.

Geometry-sensing assignment is not intended to provide "automatic" dimensioning. It is best used to provide a basic dimensioning framework that you can then modify as needed to generate the final dimensioning.

Dimensioning with Geo Sense

 **Geo Sense** assigns dimensions to preselected types (see [Geo Sense Settings on page 339](#)) of geometric elements. The type and location of the dimensions is determined automatically by the system, based on the geometry type and the location of the selected elements. The following dimension types can be generated:

DIM_LINE	for lines
DIM_RADIUS	for arcs, fillets and circles (by default)
DIM_DIAMETER	for arcs, fillets and circles (optional)
DIM_CHAMFER	for 45-degree chamfers (optional)
DIM_ARC	for arcs and fillets (optional)

To assign dimensions with **Geo Sense**:

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Automatic Dimensioning**, click  **Geo Sense**.
3. Click the required line, arc, fillet, chamfers or circle.

4. Select a group of drawing elements, either with the  **Select** buttons or by enclosing them in a box.

Creo Elements/Direct Drafting generates and positions dimensions for the selected elements.

5. Click  **OK**.

Dimensioning with Scan

 **Scan** assigns dimensions to preselected types (see [Scan Settings on page 340](#)) of geometric elements. The type and location of the dimensions are determined from parametric constraints currently assigned to the selected elements. Generally, the constraint information allows Creo Elements/Direct Drafting to generate more accurate dimensioning than is possible with  **Geo Sense**. If no constraints are present in the selected geometry, however, dimensions will not be generated.

 **Scan** generates the following dimension types for selected elements:

DIM_LINE	for lines and splines
DIM_ANGLE	for angle elements
DIM_RADIUS	for arcs, fillets and circles (by default)
DIM_DIAMETER	for arcs, fillets and circles (optional)

To assign dimensions with **Scan**:

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Automatic Dimensioning**, click  **Scan**.
3. Click the required line, arc, fillet, chamfers or circle.
4. Select a group of drawing elements, which must have been previously given parametric constraints, either with the  **Select** buttons or by enclosing them in a box.

Creo Elements/Direct Drafting generates dimensions for the selected elements.

5. Click  **OK**.

Database-Matching Assignment

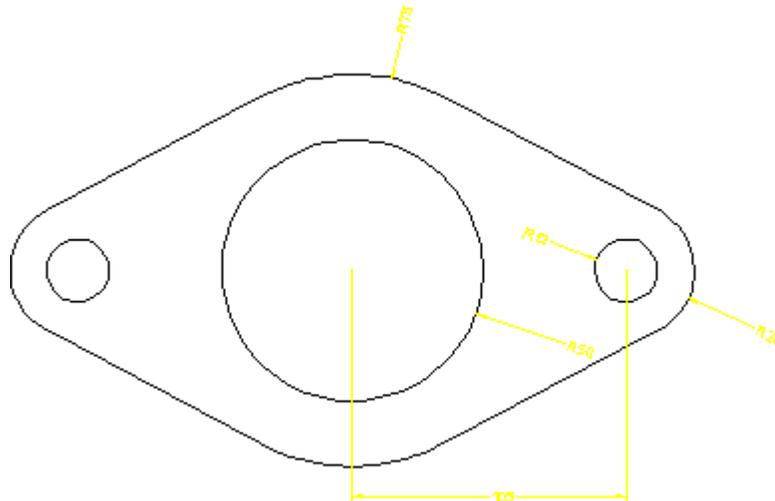
Database-matching commands allow you to create a database of common drawings, parts, or features that have already been dimensioned to company standards. You can then use the entries in this database as templates for assigning dimensions to similar (but not necessarily identical) features and parts in the future, ensuring consistent dimensioning.

All database-matching commands are found in the **Dimension Match** dialog box which you can access using the  **Match** button in the **Annotate** group on the **Drafting** tab.

Database Matching Example

1. Draw and dimension the flange shown in [Figure 109 on page 301](#).

Figure 109. Initial Part for Database Matching

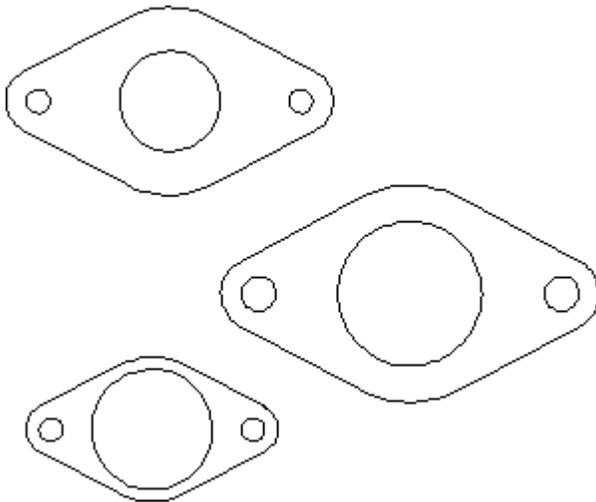


2. Add the flange to the dimension database:
 - a. Delete the construction lines from the drawing.
 - b. Click **Drafting** and then, in the **Annotate** group, click **More**.
 - c. Under **Automatic Dimensioning**, click  **Match**.
The **Dimension Match** dialog box opens.
 - d. Click **Add**.
 - e. Type the name 'Flange 001' when prompted.
 - f. Select the entire drawing by boxing it.

Creo Elements/Direct Drafting adds your drawing to the dimension database.

3. Delete all dimensions from your drawing, and using the buttons on the **Modify** tab, make two or three copies of it. You can further modify each copy by scaling the whole part or resizing the holes ([Figure 110 on page 302](#)). Do not make modifications that alter the basic topology of the part.

Figure 110. New Parts Needing Dimensions



4. Return to the **Dimension Match** dialog box and click the Flange 001 entry from the database table.

Creo Elements/Direct Drafting shows the Flange 001 entry in the style display window.

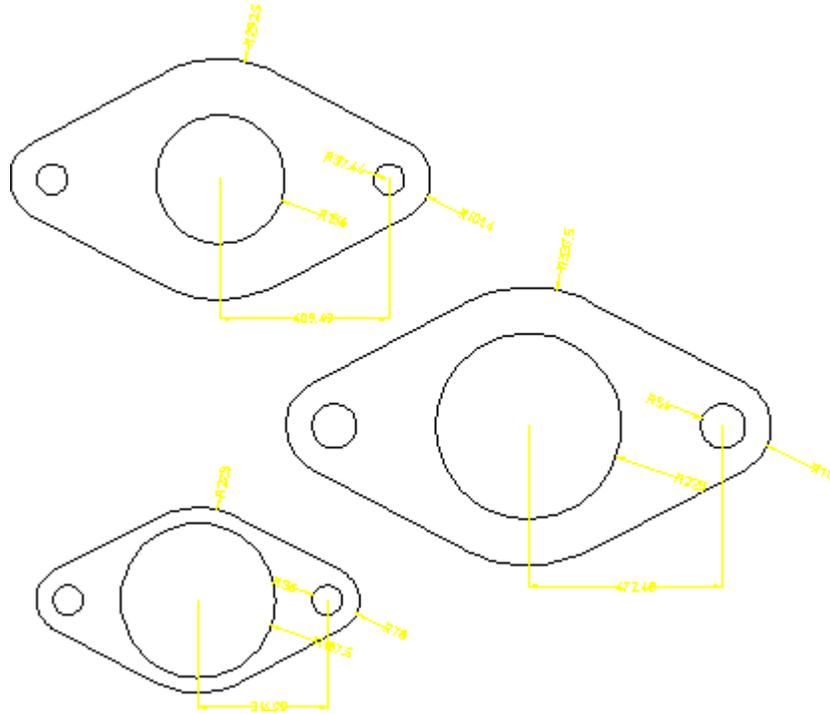
5. Identify a reference element on the Flange 001 entry by clicking directly in the style display window. For this example, start by clicking the line element on the lower left edge of the flange.
6. Identify the corresponding target element on one of the flanges in your drawing.

Creo Elements/Direct Drafting transfers the dimensioning from the database entry to the target drawing ([Figure 111 on page 303](#)). You may need to repeat the previous two steps in order to transfer the dimensioning for the flange holes.

Note that each time you transfer dimensions from the database to the target, a portion of the database entry is highlighted. Once a database element is highlighted, it cannot be used for further matching during this iteration of the command. You can start a new iteration of the command by clicking the name of the entry from the database table.

7. Repeat the previous three steps as needed until all the flanges in your drawing are dimensioned.

Figure 111. Assigning Dimensions from the Database



Adding Entries to the Dimension Database

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Automatic Dimensioning**, click  **Match**.
The **Dimension Match** dialog box opens.
3. Click **Add**.
4. Type the name that will identify this entry ('flange 002', for example).
5. Select the geometry to be added to the database by boxing.

Creo Elements/Direct Drafting creates a new entry in the database and adds the selected geometry under that entry.

Deleting Entries from the Dimension Database

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Automatic Dimensioning**, click  **Match**.

The **Dimension Match** dialog box opens.

3. Click **Delete**.
4. Enter the name of the entry to be deleted, either by clicking its entry in the dimension database table or by typing it in the user input line.

Creo Elements/Direct Drafting deletes the selected entry from the dimension database.

Assigning Dimensions from the Database

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Automatic Dimensioning**, click  **Match**.

The **Dimension Match** dialog box opens.

3. Select a database entry by clicking its entry in the database table.
Creo Elements/Direct Drafting draws the database entry in the style display window.
4. Click one end of a reference element on the database entry. Do this by clicking in the style display window.
5. Click the corresponding end of a target element in your drawing.

Creo Elements/Direct Drafting dimensions your drawing to match the database entry.

6. Click  **OK**.

Note

If you find that a template image in the style display window is too small or not detailed enough for you to easily select a reference element, you can use the **View** buttons in this window to make your job easier. To make the style display window active so that these commands will work on it, click on the mark at the top right corner of the window.

Working Methods and Limitations

When assigning dimensions from the database, you should follow the guidelines below for the best results.

- Make sure that the geometry of the dimensioned part does not contain any duplicate or stacked contours. Optionally, use the **Parametric Design** module to remove such contours.
- Make sure that the dimensioned geometry does not contain any unwanted gaps. Click the **Vertex** check box in the **Show** dialog box to check for green vertices (which indicate gaps).
- To identify an element, it is best to click it near its center. Then click a point on the element (such as a vertex) away from its center.
- The topology of the database entry and your drawing must be the same. You cannot, for example, use a circular template to assign dimensions to a rectangle. The assignment tool can compensate for minor topological differences such as line splits, but major differences will lead to unsatisfactory results.

The topological differences between a database entry and your drawing may be subtle. If dimension assignments don't appear to be working correctly, you can click the **Vertex** check box in the **Show** dialog box in your drawing window to try to track down the differences.

- The part structure (if any) of the database entry and your drawing should be similar. Results can be unsatisfactory if you try to match a multi-part database entry to a single-part drawing.
- The reference element on the database entry and the target element on your drawing must be in the same relative location, or the resulting dimensioning will be unpredictable.
- A reference element can only be matched once in a single iteration of  **Match**. Once you have matched one or more reference elements to your drawing, they are highlighted in the style display to indicate that they cannot be used for further matching.

Exporting a Database Entry

The **Export** option copies a selected entry from the dimensioning database into the current drawing. The copied entry becomes a new part in the current drawing.

A common application for **Export** is to edit an entry in the dimensioning database. To edit a database entry, export it into the current drawing, edit the exported geometry, and then **Add** it back into the database.

Saving and Loading Dimension Databases

You can use the **Store** and **Load** options to save the current dimension database to a file and later reload it into Creo Elements/Direct Drafting. Note that when you load a database into Creo Elements/Direct Drafting, its entries are appended to the current database.

To save the current database:

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Automatic Dimensioning**, click  **Match**.

The **Dimension Match** dialog box opens.

3. Click **Store**.
4. Type a filename (Flange_DB, for example).

Creo Elements/Direct Drafting stores the current database under the given filename.

To load a dimension database from a file, appending to the current database:

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Automatic Dimensioning**, click  **Match**.

The **Dimension Match** dialog box opens.

3. Click **Load**.
4. Type or select a filename (for example, Flange2_DB).

Creo Elements/Direct Drafting loads the entries from the named database file at the end of the current database.

Unloading the Dimension Database From Memory

To unload the current dimension database from memory and clear the current database table, use the **Unload** command:

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Automatic Dimensioning**, click  **Match**.

The **Dimension Match** dialog box opens.

3. Click **Unload**.

Creo Elements/Direct Drafting unloads the current database from memory.

Common Interrupt Functions

You can interrupt any direct assignment command to change the prefix, postfix, and/or tolerance values for the dimension text. To find out how to predefine these values, see [Predefining Prefixes, Postfixes, Superfixes, Subfixes, and Tolerances](#) on page 324 in this chapter.

Adding a New Prefix

You can interrupt a dimensioning command to add a new prefix to the dimension text. Examples of prefixes are:

- DIA (diameter)
- R (radius)
- CHAMFER
- BC (base circle)

After entering the prefix, complete the dimensioning command and Creo Elements/Direct Drafting will draw the dimension including the prefix.

The new prefix will also be included in all subsequently assigned dimensions. This avoids having to enter the same prefix every time you are dimensioning a similar geometry feature.

To add a new prefix:

1. Click **Modify** and then, in the **Annotate** group, click  **Style**.

2. Click  **Text Extension**.

The **Set Text Extension** dialog box opens.

3. Click the button next to **Text**.

The **Edit Text** dialog box opens.

4. Define the prefix.
5. Complete the dimensioning command.

Creo Elements/Direct Drafting draws the dimension including the prefix text.

To cancel the function and leave the dimensioning command active, click **Clear**.

Adding Special Characters

You can include special characters such as ° and Phi in your prefix:

1. Select and copy the special character from any character table or from an existing text document in your Windows environment.
2. Click the **Edit Text** box.

3. Right-click and select **Paste** to add the special character to the prefix.

Also, you can add any special character defined in the `hp_symbols` and `hp_symbols2` fonts. See [Special Characters in Dimension Text Extensions](#) on page 328 for instructions.

Dimension Line Settings

You can set the attributes of dimension lines using the **Dimension Settings** dialog box.

To open the **Dimension Settings** dialog box:

- Click **File** ► **Settings** ►  **Dimension**.

The **Line** settings of the **Dimension Settings** dialog box enable you to set attributes of dimension lines:

- Line Settings
- Terminator (Arrow) Settings

These settings are applied to all subsequent dimensioning that you assign using the **Annotate** group.

Line Settings

In the **Line** area of the **Line** settings in the **Dimension Settings** dialog box, you can set the following dimension line attributes:

Width	Sets the width for all subsequent dimension lines.
Offset Line	Sets the distance between the top end of the extension line and the dimension line.
Space	Sets the distance between the dimension text and the dimension line.
Datum Step	Sets the vertical distance (or step) that is automatically placed between datum dimension lines drawn with the dimension placement set to Automatic .
Color	Sets the color for extension lines, dimension lines, and text frames.
Offset Point	Sets the offset distance between a geometry feature and the dimensioning extension lines.
Minimum Space	Sets the minimum distance allowed between a geometry feature and its dimension line.
Catch Line	Sets whether or not a dimension may be selected by clicking its dimension line, extension line, or text.

Broken	Sets whether or not a line is drawn between the extension lines of a dimension when the dimension text is placed outside of the extension lines.
Catch Range	Sets the size of the catch range used to automatically center dimensioning text between extension lines.

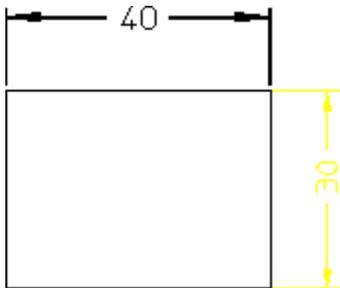
Dimension Line Width

To set the width of the dimension line:

1. Click **Line** in the **Dimension Settings** dialog box.
2. In the **Width** box, type the required value (such as .25).

Dimension a drawing element to see the effect of the changed dimension line width.

Figure 112. Dimension Line Width



Offset Line

Various national standards are used for dimensioning (American Standards, British Standards, and so on). You can set the distance (or offset) between the top end of the extension line and the dimension line to conform to the standards you use. The Creo Elements/Direct Drafting default for this distance is 2 mm.

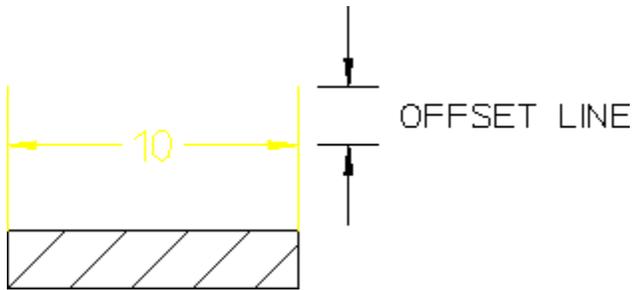
To set the dimension offset:

1. Click **Line** in the **Dimension Settings** dialog box.
2. In the **Offset Line** box, type a positive value (for example 3).

Creo Elements/Direct Drafting sets this distance for all subsequent dimensions.

[Figure 113. Dimension Offset Line Distance on page 310](#) shows the dimension offset line distance.

Figure 113. Dimension Offset Line Distance



Space

The **Space** box enables you to specify a gap between the dimension text and its associated dimension line.

Note

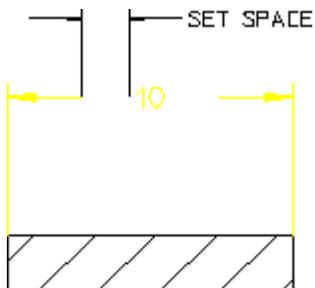
This gap is shown only if the dimension text location is set to On. See [Dimension Text Location on page 321](#).

To set this space:

1. Click **Line** in the **Dimension Settings** dialog box.
2. In the **Space** box, type a positive value (for example 2 . 3).

[Figure 114. Distance between Dimension Lines and Text on page 310](#) shows the distance between the inner ends of the dimension lines and the dimension text. The Creo Elements/Direct Drafting default for this distance is 2 mm.

Figure 114. Distance between Dimension Lines and Text



Datum Step

You may find that your datum dimensions are positioned too close together. This section shows how to specify a vertical distance (or step) between each subsequent datum dimension line.

The Creo Elements/Direct Drafting default for this distance is 8 mm. The distance is active for the  **Datum Long** and  **Datum Short** buttons.

To specify a datum step:

1. Click **Line** in the **Dimension Settings** dialog box.
2. In the **Datum Step** box, type a new distance value.

Creo Elements/Direct Drafting sets the vertical distance to the new value.

3. Complete, or start the dimensioning command.

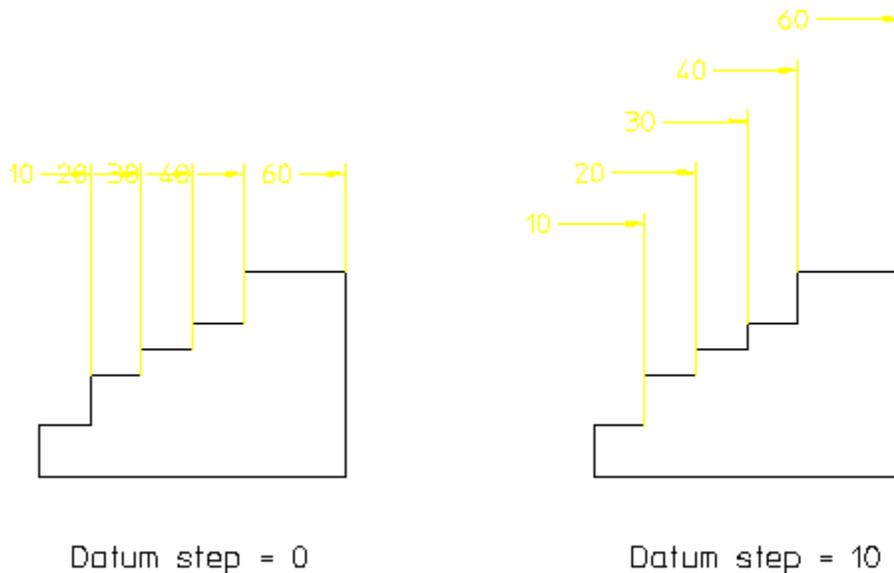
Creo Elements/Direct Drafting steps the dimension line for each dimension.

Example

Figure 115 on page 311 shows the difference between:

- Datum dimensioning without using **Datum Step** (datum step distance is 0). This produces a poor quality dimension where the dimension text has been overwritten by the dimension lines.
- Datum dimensioning using **Datum Step** (datum step distance is 10).

Figure 115. Datum Stepping



Dimension Line Color

To set the color of dimension lines and dimension extension lines:

1. Click **Line** in the **Dimension Settings** dialog box.

2. Select a color from the **Color** box. Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

 **Tip**

To set a single color for dimension lines and dimension texts while defining a dimension (when dimension buttons are active), use  in the **Format** group on the **Drafting** tab.

Dimension Offset Point

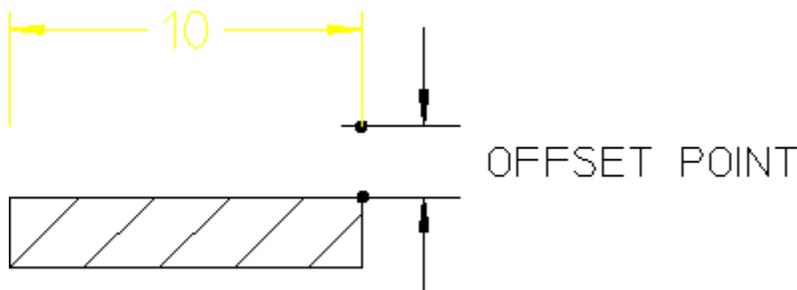
Various national standards are used for dimensioning (American Standards, British Standards, and so on). You can set the distance (or offset) between the geometry feature and the dimension extension lines to conform to the standards you use. The Creo Elements/Direct Drafting default for this distance is 0 mm.

To set the dimension offset point distance:

1. Click **Line** in the **Dimension Settings** dialog box.
2. In the **Offset Point** box, type a positive value (such as 1 . 5).

[Figure 116 on page 312](#) shows the dimension offset point.

Figure 116. Dimension Offset Point



Minimum Space

This section shows how to avoid crowding dimensions by setting a minimum distance (or space) between the geometry feature you are dimensioning and the dimension line. If you try to locate the dimension trace closer than the minimum distance, the trace will skip to the opposite side of the geometry feature. The Creo Elements/Direct Drafting default for this distance is 0 mm.

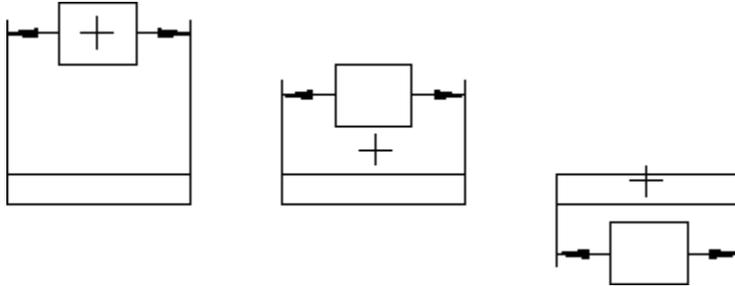
To set the minimum space:

1. Click **Line** in the **Dimension Settings** dialog box.

2. In the **Minimum Space** box, type a positive value (such as 10).

Refer to [Figure 117 on page 313](#). If you try to locate the dimension text closer than the minimum distance (10 mm), the dimension skips to the opposite side of the geometry feature.

Figure 117. Locating a Dimension – Closer Than the Minimum Distance



Catch Line

When you want to select a dimension, you can specify which components of the dimension can be selected. This is done by clicking or clearing the **Catch Line** check box in **Line** settings of the **Dimension Settings** dialog box.

Click	A dimension may be selected by clicking one of the dimension components: <ul style="list-style-type: none"> • Dimension line • Extension line • Dimension text This is the default.
Clear	A dimension may be selected by clicking the dimension text only.

Broken or Unbroken Dimension Lines

Drawing Dimension Lines Inside Extension Lines

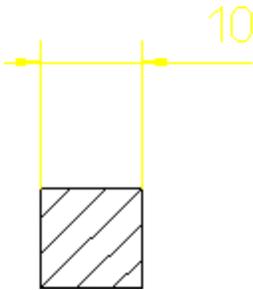
Because of space limitations when dimensioning, you may want to place dimension text outside the extension lines, but still want the dimension lines to be drawn inside the extension lines.

To draw an unbroken dimension line inside the extension lines:

1. Click **Line** in the **Dimension Settings** dialog box.
2. Clear the **Broken** check box.

Figure 118. [Dimension Lines Inside Extension Lines on page 314](#) shows dimension lines drawn inside the extension lines and dimension text outside.

Figure 118. Dimension Lines Inside Extension Lines



Drawing Dimension Lines Outside Extension Lines

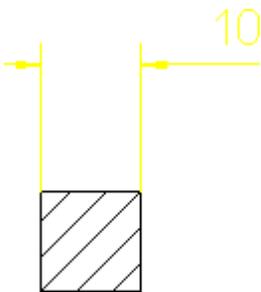
Because of space limitations you might want to place dimension lines and text outside the extension lines.

To draw an unbroken dimension line outside the extension lines:

1. Click **Line** in the **Dimension Settings** dialog box.
2. Click the **Broken** check box.

Figure 119. [Dimension Lines and Text Outside Extension Lines on page 314](#) shows how dimension lines and text are drawn outside the extension lines.

Figure 119. Dimension Lines and Text Outside Extension Lines



Catch Range

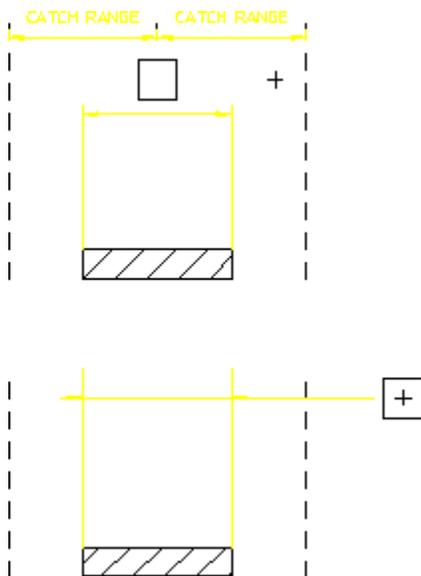
This section shows how to center dimension text whenever you indicate a point within the catch range. The catch range is governed by two catch lines parallel to the extension lines. The origin is the bisection of the distance between the two extension lines and the catch range is the horizontal distance from the origin to the catch lines. If you indicate any point within the catch range, Creo Elements/Direct Drafting will center the dimension text exactly between the two extension lines.

Figure 120 on page 315 shows the catch range. Any point you indicate within the catch range results in Creo Elements/Direct Drafting centering the dimension text.

To specify the catch range:

1. Click **Line** in the **Dimension Settings** dialog box.
2. In the **Catch Range** box, type the catch range value.

Figure 120. Dimension Catch Range



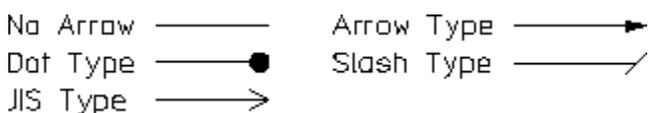
Terminator Settings

Creo Elements/Direct Drafting allows you to set the type, size, and fill attributes for all subsequent dimension line terminators in the **Arrow** area of the **Line** settings.

Normally, the terminator is an arrowhead.

[Figure 121 on page 315](#) shows the five possible dimension line terminators:

Figure 121. Dimension Line Terminators



To set the dimension line terminator:

1. Click **Line** in the **Dimension Settings** dialog box.
2. Select the required terminator settings in the **Arrow** area.

Creo Elements/Direct Drafting uses these settings for all subsequent dimensions.

You can set the first, the second or both terminators using the following options:

- **Both Identical**
- **First Arrow**
- **Second Arrow**

Both

Click the **Both Identical** check box and then select the required settings. The settings you select applies to both the dimension line terminators.

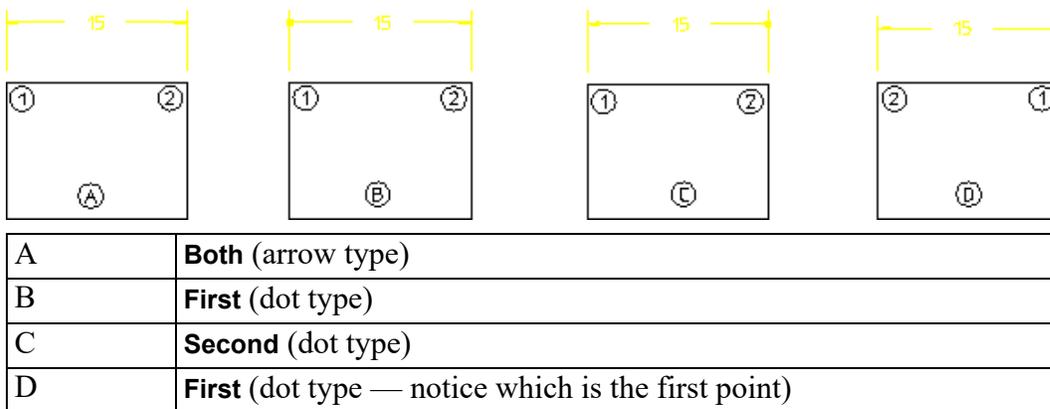
First Arrow

In the **First Arrow** area, define the settings for applying the chosen terminator settings to the first point you click when identifying the dimension start-point.

Second Arrow

In the **Second Arrow** area, define the settings for applying the chosen terminator settings to the second point you click when identifying the dimension end-point.

Figure 122. Dimension Terminators — First and Second



Because you specify a first point and a second point, it makes no difference if the dimensions are horizontal or vertical.

Terminator Position

By default, the position of terminators with respect to the extension lines is determined automatically. If there is enough room between the dimension's extension lines for the terminators and dimension text, the terminators are placed between the extension lines. Otherwise, terminators are placed outside of the extension lines. The terminator position can also be set manually.

To set the terminator position:

1. Click **Line** in the **Dimension Settings** dialog box.

-
2. In the **Position** box, select **Auto** or **Inside** or **Outside**.

 **Note**

You cannot change the direction of arrows in [Broken on page 308](#) dimension lines.

Absolute Terminator Size

The Creo Elements/Direct Drafting default terminator size is 3.5 mm. To set the absolute size of the terminator:

1. Click **Line** in the **Dimension Settings** dialog box.
2. In the **Size** box, type a value for the terminator size.

Relative Terminator Size

You can set the size of the dimension line terminator by entering a scale factor relative to the main dimension text size.

The Creo Elements/Direct Drafting default scale factor is 1 and the default main dimension text size is 3.5 mm.

To set the relative size of the dimension line terminator:

1. Click **Line** in the **Dimension Settings** dialog box.
2. Click the **Relative** check box.

Terminator Fill On or Off

By default, dimension line terminators are filled.

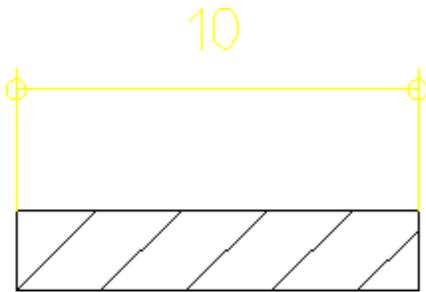
To enable or disable the filling of dimension line terminators:

1. Click **Line** in the **Dimension Settings** dialog box.
2. Click or clear the **Fill** check box to enable or disable the fill mode.

Example

[Figure 123 on page 318](#) shows unfilled dot type terminators.

Figure 123. Unfilled Dot Type Terminators



Dimension Text Settings

The **Text** settings on the **Dimension Settings** dialog box enable you to set the attributes of all dimension text or of specific dimension text components. These settings will be applied to all subsequent dimensioning that you assign via the **Annotate** tab.

Set	Enables you to select the dimension text components to which the dimension text settings will apply.
Font	Sets the font in which these dimension text components will be displayed.
Color	Sets the color for the selected dimension text components.
Orientation	Sets the orientation for the selected dimension text components.
Frame	Determines what kind of box (if any) is drawn around subsequent dimensioning text.
Location	Sets the location of the dimension text relative to the dimension line.
Size	Sets the size of dimension text.
Offset Line	Sets the offset distance between dimension text and the dimension line.
Slant	Sets the dimension text slant.
Ratio	Sets the character height-to-width ratio of dimension text. Sets text appearance attributes such as size, font, slant, and aspect ratio.
Brackets	Encloses primary and secondary dimension text in brackets.
Fill	Fills characters of block fonts.

Dimension Text Components

Dimension text is comprised of several components:

- Prefix
- Dimension value
- Tolerance
- Postfix

To select the dimension text component(s) to which you want to apply the dimension text settings:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **Set** box, select a dimension text component from the following:
 - **Upper**
 - **Lower**

By default, the dimension text settings apply to **all** dimension text components.

3. Click the **All** check box to apply the dimension settings to dimension text components.
4. Select or specify the required dimension text settings.

Creo Elements/Direct Drafting uses these settings for all subsequent dimensions.

Dimension Text Font

To select a dimension text font:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **Font** box, select a font. The font names in this selection list include:
 - Fonts supplied with Creo Elements/Direct Drafting
 - Fonts already installed on your PC.

To load additional fonts for selection, follow the instructions in [Importing a Font on page 543](#).

Dimension Text Color

To set the color of dimension text:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **Color** box, select a color. Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

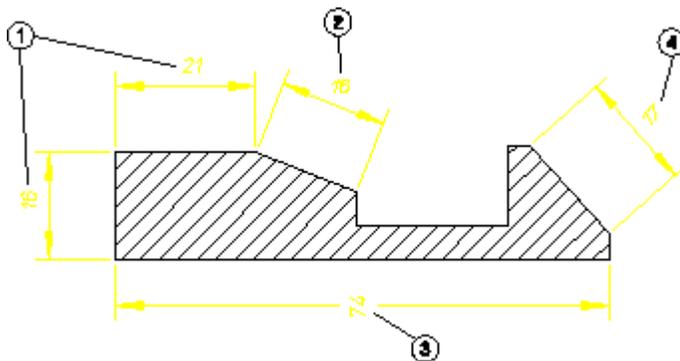
Tip

To set a single color for dimension texts and dimension lines while defining a dimension (when dimension buttons are active), use  in the **Format** group on the **Drafting** tab.

Dimension Text Orientation

The orientation of the dimension text can be one of the following:

Figure 124. Dimension Orientation



- 1 Parallel to the dimension line. This is the default.
- 2 Horizontal (2).
- 3 Vertical (3).
- 4 Perpendicular to the dimension line.

To set the orientation of dimension text:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **All Texts** area, select one of the following in the **Orientation** box:
 - **Parallel**
 - **Perpendicular**
 - **Horizontal**
 - **Vertical**

Dimension Text Frame

To set the dimension text frame:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **All Texts** area, select one of the following in the **Frame** box:

- **None**
- **Box**
- **Balloon**
- **Flag**
- **Basic**

The basic dimension text frame allows you to have a frame just around the dimension main text; by that you can mark all basic dimensions in your drawing. In addition to that Creo Elements/Direct Drafting enables you to include a user-defined portion of the prefix and postfix text into the basic frame in addition to the normal prefix and postfix text outside the frame. You can add and edit both, the basic and normal prefix and postfix text. You can also convert an existing dimension text frame to a basic dimension text frame.

Figure 125 on page 321 shows the dimension text frame options.

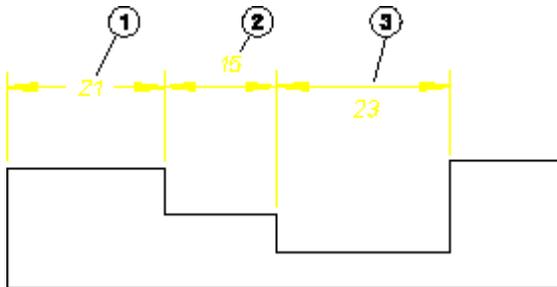
Figure 125. Dimension Text Frame – Options



Dimension Text Location

The location of dimension can be in one of three positions:

Figure 126. Dimension Location



- 1 On the dimension line.
- 2 Above the dimension line. This is the default.
- 3 Below the dimension line.

To set the location of dimension text:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **All Texts** area, select one of the following in the **Location** box:
 - **On**
 - **Above**

- **Below**

Absolute Dimension Text Size

The Creo Elements/Direct Drafting default dimension text size (character height) is 3.5 mm. To set the absolute size of dimension text:

1. Click **Text** in the **Dimension Settings** dialog box.
2. Ensure that you clear the **Relative** check box.
3. In the **Size** box, type the absolute dimension text size.

Relative Dimension Text Size

Creo Elements/Direct Drafting multiplies the relative factor by the current main dimension text size. This gives a new text size for all selected dimension text.

To set a relative text size:

1. Click **Text** in the **Dimension Settings** dialog box.
2. Ensure that you click the **Relative** check box.
3. In the **Size** box, type the relative dimension text size.

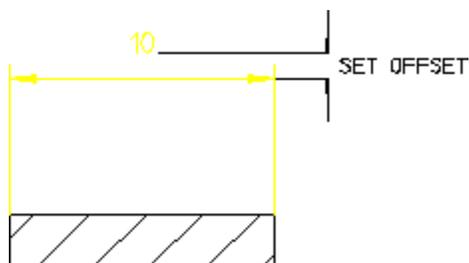
Dimension Text Offset

Note

This function only works if you have previously selected either **Above** or **Below** in the **Location** box.

Figure 127 on page 322 shows the distance (or offset) between the dimension line and dimension text. The Creo Elements/Direct Drafting default for this distance is 2 mm.

Figure 127. Dimension Text Offset



To set the dimension text offset:

-
1. Click **Text** in the **Dimension Settings** dialog box.
 2. In the **Offset Line** box, type a positive offset.

Dimension Text Slant

The Creo Elements/Direct Drafting default slant of dimension text characters is 0 degrees.

To set the dimension text slant:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **Slant** box, type a new slant angle.

Dimension Text Ratio

The Creo Elements/Direct Drafting default for the width-to-height ratio of dimension text characters is 1. A number smaller than one narrows dimension text characters, a number greater than one widens dimension characters.

To change the dimension text character width-to-height ratio:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **Ratio** box, type the new value.

Dimension Text Brackets

Primary and secondary dimension text can be enclosed in brackets. By default, brackets are not enabled for enclosing the primary and secondary dimension text.

To enable or disable dimension text brackets:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **Set** box, select **Upper** or **Lower**.
3. Click or clear the **Brackets** check box to enable or disable brackets, respectively.

Coordinate Dimension Leading Zero

The leading zero in coordinate dimensions can be shown or hidden. By default, the leading zero is shown.

To show or hide the leading zero:

1. Click **Text** in the **Dimension Settings** dialog box.
2. In the **Specific Texts** area, click or clear the **Coordinate Dimension - Leading Zero** check box to show or hide the leading zero, respectively.

Predefining Prefixes, Postfixes, Superfixes, Subfixes, and Tolerances

If you frequently add dimension text extensions (such as postfixes) to a dimension, it is useful to first predefine these via the **Dimension Settings** dialog box. Once defined, they can easily be selected from a table. Apart from making the process quicker, this method also improves consistency.

The **Text Ext** settings of the **Dimension Settings** dialog box enable you to add, delete, save, open and clear the following:

- **Prefix**
- **Postfix**
- **Superfix**
- **Subfix**

See [What is Dimension Text?](#) on page 253 for a definition of these terms.

Adding a Dimension Text Extension

Although the following example is for a prefix, the procedure is the same for a postfix, a superfix, a subfix, and a tolerance.

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension** area, select **Prefix** in the **Set** box.
3. Type a name for the prefix (such as p1) into the entry field above the (originally empty) prefix list.
4. Type the prefix text (such as the following example) in the **Ext Text** box.
Examples of prefixes are:
 - DIA (diameter)
 - R (radius)
 - CHAMFER
 - BC (base circle)
5. Optionally, you can select a special character to serve as the dimension text extension:

Click **Spec Char** to display a graphical list of special text characters. The **Special Characters** window opens.

 **Note**

The special characters are based on the `hp_symbols` and `hp_symbols2` fonts.

6. Select the special character you want to use as a dimension text extension. A graphical representation of each character is provided in the **Meta Name** column.
7. Click **Insert** to copy the selected special character to the dimension text extension. Note that the meta name of this special character is added to the **Ext Text** box.
8. In the **Extension** area, type the basic dimension prefix text string in the **Ext Text** box.
9. Click **Add** to add this prefix to your list of prefixes. Each time you click a dimension, you can select the desired prefix from the list.

Which Dimension Text Extension or Extensions Will be Used in the Drawing?

When the prefix list contains multiple prefix names, the prefix named in the name entry field (above the prefix list) will be used in any dimension you add to your drawing. This rule also applies to the other dimension text extension types (postfix, superfix, subfix, and tolerance). The dimension text extension specified in the name entry field is active and will be included in subsequent dimensions.

To add a dimension plus prefix to a drawing:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension** area, select **Prefix** in the **Set** box.
3. Click the prefix you want from the list.
4. Click the begin and end points of the dimension.

Deleting a Predefined Dimension Text Extension

Although the following example is for a prefix, the procedure is the same for a postfix, a superfix, a subfix, and a tolerance.

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension** area, select **Prefix** in the **Set** box.
3. Select the name of the prefix to be deleted from the prefix list.
4. Click **Delete** to delete this prefix.

Saving Predefined Dimension Text Extensions

Although the following example is for a prefix, the procedure is the same for a postfix, a superfix, a subfix, and a tolerance.

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension** area, select **Prefix** in the **Set** box.
3. Click **Store** to display the **Store** dialog box.
4. Specify a filename and select a file type.
5. Click **Save** to save the dimension text extension or extensions.

Opening Predefined Dimension Text Extensions

Although the following example is for a prefix, the procedure is the same for a postfix, a superfix, a subfix, and a tolerance.

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension** area, select **Prefix** in the **Set** box.
3. Click **Load** to display the **Load** dialog box.
4. Select or specify a filename under which you saved predefined dimension text extensions.

Clearing a Predefined Dimension Text Extension

Although the following example is for a prefix, the procedure is the same for a postfix, a superfix, a subfix, and a tolerance.

1. Click the **Text Ext** tab in **Dimension Settings**.
2. In the **Extension** area, select **Prefix** in the **Set** box.
3. Click **Clear** to remove the prefix name (if any) from the name entry field above the prefix list. Note that the **Clear** command merely deactivates the prefix. It remains in the prefix list for further use unless you **Delete** it.

Dimension Text Extension Settings

The **Extension Text** area of **Text Ext** settings of the **Dimension Settings** dialog box enables you to define the following attributes of prefixes, postfixes, superfixes, subfixes and tolerances:

- Font
- Color
- Ratio
- Size

-
- Slant
 - Special Characters
 - Basic Dimension Text Extensions

Dimension Text Extension Font

To select a dimension text extension font:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension Text** area, select a font in the **Font** box. The font names in this selection list include:
 - Fonts supplied with Creo Elements/Direct Drafting.
 - Fonts already installed on your PC.

To load additional fonts for selection, follow the instructions in [Importing a Font on page 543](#).

Dimension Text Extension Color

To set the color of dimension text extensions:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension Text** area, select a color in the **Color** box. Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

Dimension Text Extension Ratio

The Creo Elements/Direct Drafting default for the width-to-height ratio of dimension text characters is 1. A number smaller than one narrows dimension text characters, a number greater than one widens dimension characters.

To change the dimension text character width-to-height ratio:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension Text** area, type the new value in the **Ratio** box.

Absolute Dimension Text Extension Size

The Creo Elements/Direct Drafting default dimension text extension size (character height) is 3.5 mm. To set the absolute size of dimension text extensions:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension Text** area, ensure that you clear the **Relative** check box.
3. Type the absolute dimension text extension size in the **Scale/Size** box.

Relative Dimension Text Extension Size

Creo Elements/Direct Drafting multiplies the relative factor by the current main dimension text size. This gives a new text size for dimension text extensions.

To set a relative text extension size:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension Text** area, ensure that you click the **Relative** check box.
3. Type the relative dimension text extension size in the **Scale/Size** box.

Dimension Text Extension Slant

The Creo Elements/Direct Drafting default slant of dimension text extension characters is 0 degrees.

To set the dimension text extension slant:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension Text** area, type a new slant angle in the **Slant** box.

Special Characters in Dimension Text Extensions

To add a special character to a dimension text extension:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension Text** area, select the type of extension to which you want to add a special character.
3. Click **Spec Char** to display a graphical list of special text characters.

These special characters are included in the `hp_symbols` and `hp_symbols2` fonts. You can create additional special characters by editing these fonts.

4. Select the special character you want to add to the dimension text extension. A graphical representation of each character is provided in the **Meta Name** column.
5. Click **Insert** to copy the selected special character to the dimension text extension.

Basic Dimension Text Extensions

To add a basic dimension text extension to a normal dimension text extension:

1. Click **Text Ext** in the **Dimension Settings** dialog box.
2. In the **Extension Text** area, click the button next to the **Ext Text** box.

The **Edit Text** dialog box opens.

-
3. Type the basic dimension extension text in the **Edit Text** box.
 4. Click **OK** to save the basic dimension text extension.

All the dimensions created after setting the basic dimension text extension inherit the text extension.

Dimension Unit Settings

The **Units** settings of the **Dimension Settings** dialog box enable you to set the units of all dimension text or of specific dimension text components. These settings will be applied to all subsequent dimensioning that you assign using the dimension buttons in the **Annotate** group on the **Drafting** and **Modify** tabs.

To select the dimension text component or components to which you want to apply the unit settings:

1. Click **Units** in the **Dimension Settings** dialog box.
2. In the **Units** area, select a dimension text component in the **Set** box from the following:
 - Select **Upper** to set the units for the main dimension text.
This is the default setting.
 - Select **Lower** to set the units for the lower dimension text (if applicable).

Linear Units

After you select the **Upper** or **Lower** dimension text component in the **Set** box, select the linear units for that component from the following:

- **Micrometers**
- **Millimeters**
- **Centimeters**
- **Meters**
- **Kilometers**
- **Inches**
- **Fractional Inches**
- **Feet and Fractional Inches**
- **Feet and Fractional Inches with Sign**
- **Feet and Fractional Inches with Text**

Micrometers

Select **Micrometers** (um) to set the dimension units to micrometers.

Millimeters

Select **Millimeters** to set the dimension units to millimeters. This is the default option.

Centimeters

Select **Centimeters** to set the dimension units to centimeters.

Meters

Select **Meters** to set the dimension units to meters.

Kilometers

Select **Kilometers** to set the dimension units to kilometers.

Inch

Select **Inches** to set the dimension units to inches.

Fractional Inches

Select **Fractional Inches** to set the dimension units to fractional inches. For example, 12" and 6 1/2 ".

Feet and Fractional Inches

Select **Feet and Fractional Inches** to set the dimension units to feet and fractional inches. For example, 1 - 0 and 0 - 6 1/2.

Feet and Fractional Inches with Sign

Select **Feet and Fractional Inches with Sign** to set the dimension units to feet and fractional inches. Creo Elements/Direct Drafting displays the dimension value with feet and inches signs. For example, 1' - 0" and 0' - 6 1/2".

Feet and Fractional Inches with Text

Select **Feet and Fractional Inches with Text** to set the dimension units to feet and fractional inches. Creo Elements/Direct Drafting displays the dimension value with feet and inches indicated by text. For example, 1 FT 0 IN and 0 FT 6 1/2 IN.

Angular Units

After you select the **Upper** or **Lower** dimension text component in the **Set** box, select the angular units for that component from the following:

- **Degrees**
- **Degrees Minutes Seconds**
- **Radians**
- **Gradian**

Degrees

Select **Degrees** to set the angle dimension units to degrees.

Degrees Minutes Seconds

Select **Degrees Minutes Seconds** to set the angle dimension units to degrees, minutes, and seconds.

Radians

Select **Radians** to set the angle dimension units to radians.

Gradian

Select **Gradian** to set the angle dimension units to gradian.

Decimal Places

After you select a dimension unit in the **Set** box of the **Display** area, type an associated number format in the **Decimal Places** box to specify the number of decimal places for that unit.

For example, to specify that dimensions displayed in mm are shown with two decimal places, type 0.01:

The format string in the **Decimal Places** box defines:

- The type of decimal period: a comma (,) or point (.).
- The suppression of zeroes before the decimal period (left zero suppression).
- The suppression of zeroes after the decimal period (right zero suppression).
- The number of decimal places.

Creo Elements/Direct Drafting has a default format string which defines:

- The decimal period as a point (.).
- No left zero suppression. For example, 0.21 appears as 0.21.

- Right zero suppression. For example, 2.100 appears as 2.1.
- The dimension value to be displayed to two decimal places.

For example, 2.2354 appears as 2.24 and 12.299 appears as 12.3 (notice the right zero suppression).

This format is general and may not suit certain types of dimensioning. For example: large scale dimensions, precision dimensions, and so on. You can use the **Decimal Places** field to specify your own format string.

[Table 2 on page](#) explains the format string using six examples of how Creo Elements/Direct Drafting would display 10.000, 3.142, and 0.090.

Table 2. Format Strings

Format String	Represents
1,	Dimension values with a decimal comma, left zeroes not suppressed, right zeroes suppressed, and no decimal places quoted. Creo Elements/Direct Drafting displays the values 10, 3, and 0.
0.1	Dimension values with a decimal point, left zeroes not suppressed, right zeroes suppressed, and one decimal place quoted. Creo Elements/Direct Drafting displays the values 10, 3.1, and 0.1.
1,1	Dimension values with a decimal comma, both left and right zeroes suppressed, and one decimal place quoted. Creo Elements/Direct Drafting displays the values 10, 3,1, and ,1.
1.000	Dimension values with a decimal point, with left zero suppression, right zeroes not suppressed, three decimal places quoted. Creo Elements/Direct Drafting displays the values 10.000, 3.142, and .090.
0,001	Dimension values with a decimal comma, left zeroes not suppressed, right zeroes suppressed, three decimal places quoted. Creo Elements/Direct Drafting displays the values 10, 3,142, and 0,09.
+1,001	Dimension values with a decimal comma, with left and right zero suppression, three decimal places quoted, plus included in value. Creo Elements/Direct Drafting displays the values +10, +3,142, and +,09.

Selection Filtering

When you assign dimensions to multiple drawing elements at one time, you can also apply a selection filter. The selection filter prevents elements that meet certain criteria from receiving dimensions. The **Filter** settings in the **Dimension Settings** dialog box enable you to apply selection filtering.

Drawing elements can be filtered on the basis of any combination of the following types of criteria:

- Geometry Type
- Line Slope
- Linetype
- Pensize
- Color
- Linesize

Enabling and Disabling Selection Filtering

The selection filter can be enabled or disabled at any time by clicking or clearing the **Filter On** check box, respectively, in the **Filter** settings of the **Dimension Settings** dialog box.

Setting up the Selection Filter

Before you can use the selection filter, you must specify the filter criteria. You can combine options from any of the categories below to come up with a filter that meets your needs.

1. Click **Filter** in the **Dimension Settings** dialog box.
2. In the **Filter** area, click the **On** check box.
3. In the **Attributes** area, select a filter type from the following in the **Type** box:
 - **Geometry Type**
 - **Line Slope**
 - **Linetype**
 - **Pensize**
 - **Color**
 - **Linesize**
4. Select the required options for this filter type.
5. If needed, select other filter type and options until the selection filter is complete.

Geometry-Type Filter

This filter excludes elements based on their geometry type. For example, if the **Arc** and **Circle** options are set, all circular elements are excluded by this filter.

To set the geometry-type filter:

-
1. Click **Filter** in the **Dimension Settings** dialog box.
 2. In the **Filter** area, click the **On** check box.
 3. In the **Attributes** area, select the **Geometry Type** filter type in the **Type** box.
 4. Select one or more geometry types to add these to the filter. The **Geometry Type** options are:
 - **Point**
 - **Line**
 - **Arc**
 - **Fillet**
 - **Circle**
 - **Text**
 - **Spline**
 - **BSpline**

You can click **Reset** to quickly remove all geometry types from the filter.

The Line Slope Filter

The line-slope filter is a list of angle ranges. Each entry in the table has a lower bound and an upper bound, given in the current angular units. Linear elements with orientations that fall within the angle range of any entry in the table are excluded by this filter.

For example, if the orientation filter contains one entry with a lower bound of 30 degrees and an upper bound of 40 degrees, all lines with 33 degree elevations will be filtered, but lines with 45 degree elevations will remain.

Adding To The Line-Slope Filter

1. Click **Filter** in the **Dimension Settings** dialog box.
2. In the **Filter** area, click the **On** check box.
3. In the **Attributes** area, select the **Line Slope** filter type in the **Type** box.
4. Click the **Horizontal** check box related to **Exclude**.
5. In the **Lower** box, type the lower bound (in current angular units).
6. In the **Upper** box, type the upper bound (in current angular units).
7. Click **Add** to create a new entry in the filter table.

To quickly add an entry for horizontal or for vertical lines, you can click the **Horizontal** check box or the **Vertical** check box of **Exclude**.

Deleting from The Line-Slope Filter

1. Click **Filter** in the **Dimension Settings** dialog box.
2. In the **Attributes** area, select the **Line Slope** filter type in the **Type** box.
3. Select the line slope filter row you want to delete from the list.
4. Click **Delete**.

Creo Elements/Direct Drafting deletes the entry from the filter table.

The Linetype Filter

This filter excludes elements based on their linetype. For example, if the **Dashed** option is set, all elements with a dashed linetype will be excluded by this filter.

To set the linetype filter:

1. Click **Filter** in the **Dimension Settings** dialog box.
2. In the **Attributes** area, select the **Linetype** filter type in the **Type** box.
3. Click the check box of one or more linetypes to add them to the filter. Types currently in the filter are highlighted. Click **Reset** to quickly remove all linetypes from the filter.

The **Linetype** options are:

- **Solid**
- **Dashed**
- **Long Dashed**
- **Dot Center**
- **Dash Center**
- **Phantom**
- **Center Dash Dash**
- **Dotted**

The Pensize and Linesize Filter

The pensize and linesize filters are tables of size ranges. Each entry in the table has a lower bound and an upper bound, given in the current distance units. Elements with sizes that fall within the range of any entry in the table are excluded by this filter.

For example, if the pensize filter contains one entry with a lower bound of .5 mm and an upper bound of 1.0 mm, and a second entry with bounds of 1.1 mm and 3.0 mm, all lines with sizes between .5 mm and 3.0 mm will be filtered except for lines with sizes between 1.0 mm and 1.1 mm.

Adding to the Pensize or Linesize Filter

1. Click **Filter** in the **Dimension Settings** dialog box.
2. In the **Attributes** area, select the **Pensize** filter type in the **Type** box.
3. In the **Lower** box, type the lower bound.
4. In the **Upper** box, type the upper bound.
5. Click **Add**.

Creo Elements/Direct Drafting creates a new entry in the filter table.

Deleting from the Pensize or Linesize Filter

1. Click **Filter** in the **Dimension Settings** dialog box.
2. In the **Attributes** area, select the **Pensize** filter type in the **Type** box.
3. Select the row you want to delete from the table.
4. Click **Delete**.

Creo Elements/Direct Drafting deletes the entry from the filter table.

The Color Filter

The color filter is a table of colors. Elements with colors that match any entry in the table are excluded by this filter.

Adding to the Color Filter

1. Click **Filter** in the **Dimension Settings** dialog box.
2. In the **Attributes** area, select the **Color** filter type in the **Type** box.
3. In the **Add Color** box, select a color. Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.
4. Click **Add**.

Creo Elements/Direct Drafting creates a new entry in the filter table.

5. Click **OK**.

Deleting from the Color Filter

1. Click **Filter** in the **Dimension Settings** dialog box.
2. In the **Attributes** area, select the **Color** filter type in the **Type** box.
3. Select the color you want to delete from the list.
4. Click **Delete**.

Creo Elements/Direct Drafting deletes the entry from the filter list.

Clearing the Selection Filter

To clear all filter criteria from the selection filter:

1. Click **Filter** in the **Dimension Settings** dialog box.
2. Click **Reset**.

Saving and Loading Selection Filters

You can use the **Store** and **Load** to save selection filters to files and later reload them into Creo Elements/Direct Drafting, respectively.

To save the current selection filter:

1. Click **Filter** in the **Dimension Settings** dialog box.
2. Click **Save**.
3. Type a filename (for example, `Select_filter.1`) in the **Name** box.
4. Click **Save**.

Creo Elements/Direct Drafting stores the selection filter as per the specified filename.

To load a new selection filter from a file (overwriting any current filter):

1. Click **Filter** in the **Dimension Settings** dialog box.
2. Click **Load**.
3. Select a filename (for example, `Select_filter.1`).

Creo Elements/Direct Drafting loads the specified selection filter, replacing any current filter.

Dimension Options

The **Options** settings in the **Dimension Settings** dialog box enable you to set dimension location options as well as automatic dimensioning settings. These settings will be applied to all subsequent dimensioning that you assign using the dimension buttons in the **Annotate** group on the **Drafting** and **Modify** tabs.

Dimension Location Settings

In the **Dimension Location** area of the **Options** settings, you can set the following:

- Automatic Dimension Positioning
- Auto-Placement Position
- Dimension Strategy

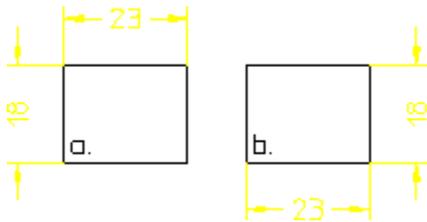
Automatic Dimension Positioning

You can click the **Automatic** check box option to automatically place dimensions rather than use the dimension trace (if the **Automatic** check box is clear).

Dimension Position Settings

By default, Creo Elements/Direct Drafting attempts to auto-place dimensions above or to the right of their associated (linear) drawing elements. Using the **Position** box in the **Options** settings of the **Dimension Settings** dialog box, you can configure Creo Elements/Direct Drafting to place dimensions either above and to the right or below and to the left (see [Figure 128 on page 338](#)).

Figure 128. Automatic Dimension Positioning — Above (a.) and Below (b.)



To set the auto-placement position:

1. Click **Options** in the **Dimension Settings** dialog box.
2. In the **Dimension Location** area, click the **Automatic** check box.
3. In the **Position** box, select **Above** or **Below**.

Dimensioning Strategy - Interference Check

By default, Creo Elements/Direct Drafting attempts to auto-place new dimensions so that the new dimension text does not intersect existing geometry or dimensioning. This process is called interference check. Full interference checking can slow down a busy system, so you are given the option of reducing or disabling this check. Doing this may improve system performance, but will also increase the likelihood that auto-placed dimensions will need to be re-located later. Note that even full interference checking cannot always prevent overlaps from occurring.

To change the level of interference checking:

1. Click **Options** in the **Dimension Settings** dialog box.
2. In the **Dimension Location** area, click the **Automatic** check box.
3. In the **Strategy** box, select **Off**, **Geo**, or **Geo+Dim**.
 - **Off** disables all interference checking.

- **Geo** checks for interference with geometry elements only.
- **Geo + Dim** checks for interference with geometry elements and with existing dimensioning.

There are two check box at the bottom of the **Options** settings in the **Dimension Settings** dialog box:

- Click the **Delete Dimension Check** check box to set the check option for deleting the dimension.
- Click the **Insert Dimension Check** check box to set the check option for **Insert**.

Automatic Dimensioning Settings

In the **Automatic Dimensioning Settings** area of the **Options** dialog box, you can set the following:

- Settings for Automatic Dimensioning using **Geo Sense**.
- Settings for Automatic Dimension using **Scan**.

Geo Sense Settings

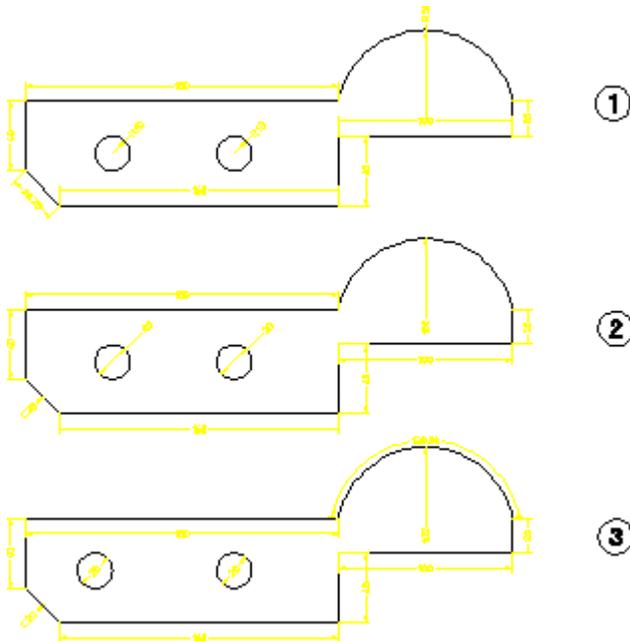
To preselect the geometry items to be included in automatic dimensioning with **Geo Sense**:

1. Click **Options** in the **Dimension Settings** dialog box.
2. In the **Automatic Dimensioning Settings** area, select the geometric element types for **Geo Sense**:

Radius	Radius dimensions are generated for all circular elements in the selection set.
Diameter	Diameter dimensions are generated for all circular elements in the selection set.
Chamfer	Chamfer dimensions are generated for any valid chamfers in the selection set.
Arc	Arc dimensions are generated for any arc elements, including fillets, found in the selection set.

[Figure 129 on page 340](#) illustrates the effects of these options on the resulting dimensions.

Figure 129. Geo Sense Options



- 1 Defaults
- 2 Diameter and Chamfer On
- 3 Diameter, Chamfer, and Arc On

Scan Settings

To preselect the geometry items to be included in automatic dimensioning with **Scan**:

1. Click **Options** in the **Dimension Settings** dialog box.
2. In the **Automatic Dimensioning Settings** area, select the geometric element types for **Scan**:

Radius	Radius dimensions are generated for all circular elements in the selection set.
Diameter	Diameter dimensions are generated for all circular elements in the selection set.

Setting Dimension Attributes in the Dimension Style Table

The **Dimension Style** table enables you to store your current dimensioning style settings (font, size, units, line styles, etc.) as table entries. You can later recover the stored style settings and make them the current dimensioning style simply by selecting the table entry. The currently active settings are always previewed in the style display window, so you always know what you will be assigning.

If you often dimension particular features or details in a given style, the **Dimension Style** table frees you from having to manually set up that style each time you need it.

By default, **Style** is clicked to indicate that you are using your own dimensioning styles. Optionally, you can click **Standard** to select one of the preconfigured industrial standards (DIN, ANSI, JIS, ISO) for dimensioning.

Adding a Style Table Entry

Before you can take advantage of the **Dimension Style** table, you need to add one or more entries to hold the style settings you want to apply later. To create an entry in this table:

1. Set up the dimensioning style that you want to save to the table. You can use the **Dimension Settings** dialog box to do this. The style display window shows the current settings.
2. Click **Style** in the **Dimension Settings** dialog box.
3. In the **Style** area, type a name for the new table entry in the name entry field above the (originally empty) style table.
4. Click **Add**.

Creo Elements/Direct Drafting adds the new entry to the **Dimension Style** table.

Setting the Current Dimension Style with the Dimension Style Table

1. Click **Style** in the **Dimension Settings** dialog box.
2. Click an entry in the **Dimension Style** table to move this entry to the name entry field above the (originally empty) style list.

The dimensioning style settings associated with this entry become the current settings. These settings will be used for subsequent dimensioning.

Assigning Dimension Styles

You can assign a new dimensioning style to an existing dimension using  **Style**.

1. Set up the desired dimensioning style, using either the **Dimension Settings** dialog box or the **Dimension Style** table.
2. Click **Modify** and then, in the **Annotate** group, click  **Style**.
3. Click  **Apply**.
4. Select dimensions in your drawing to receive the new settings.

Creo Elements/Direct Drafting updates the style of the selected dimensions to match the current style.

Setting the Current Dimension Style from An Existing Dimension

 **Grab** queries the style settings of a selected dimension and then makes them the current settings. You can use these settings for new dimensions or store them as an entry in the **Style** table.

1. Click **Modify** and then, in the **Annotate** group, click  **Style**.
2. Click  **Grab**.
3. Select the text of a dimension in your drawing.

Creo Elements/Direct Drafting makes the style settings of the selected dimension the current settings.

4. Click  **OK**.

Editing a Dimension Style Table Entry

Each entry in the **Dimension Style** table is a macro that contains the dimension style commands needed to define the entry's style. The entry can be modified by editing this macro.

1. Click **Style** in the **Dimension Settings** dialog box.
2. Select the table entry you wish to change.
3. Click **Edit**. Creo Elements/Direct Drafting displays the entry's macro in a screen editor window.

-
4. Edit the text, then save and exit the file.

Deleting an Entry

1. Click **Style** in the **Dimension Settings** dialog box.
2. Click the table entry you wish to delete.
3. Click **Delete**.

Saving the Dimension Style Table to a File

The **Dimension Style** table can be stored in a file for later use.

1. Click **Style** in the **Dimension Settings** dialog box.
2. Select a dimension style.
3. Click **Store**.
4. In the **Name** box, specify a filename and select a file type.
5. Click **OK** to save the dimension text extension or extensions.

Creo Elements/Direct Drafting stores the table macros in the given file.

Loading Table Entries from a File

New table entries can be appended to the **Dimension Style** table by loading them from a file.

1. In the **Dimension Settings** dialog box, click **Style**.
2. Click **Load**.
3. In the **Name** box, select or specify a filename under which you saved predefined dimension text extensions.

Creo Elements/Direct Drafting loads the table macros from the given file into the **STYLE** table. These entries are appended to the end of the table.

Changing Dimension Settings

To change the settings of existing dimensions:

1. Click **Modify** and then, in the **Annotate** group, click  **Properties**.
2. Select the dimension or dimensions to be changed.

You can click a single dimension or enclose multiple dimensions in a box or use the **Select** window.

The **Annotation Properties** dialog box opens.

Alternatively, you can preselect single or multiple dimensions to open the **Annotation Properties** dialog box.

 **Note**

In the preselection mode, the **Annotation Properties** dialog box opens after you click  **Properties**.

3. Click the following settings of the dimension items you want to change:

- **Line**
- **Text**
- **Units**

 **Note**

The settings in the **Dimension Settings** dialog box apply to all dimensions to be added to the drawing, while the settings in the **Annotation Properties** dialog box apply to existing dimensions you selected.

Changing Dimension Line Attributes

The **Annotation Properties** dialog box enables you to change the following attributes of the dimension lines:

- Line Settings
- Terminator (Arrow) Settings

Changing Dimension Line Settings

In the **Line** area on the **Line** settings of the **Annotation Properties** dialog box, you can change the following dimension line attributes:

Width	Changes the width of dimension lines.
Offset Line	Changes the distance between the top end of the extension line and the dimension line.

Color	<p>Changes the color of extension lines, dimension lines, and text frames.</p> <p> Note</p> <p>To use the same dimension line color for dimension texts, click the Connected To Text check box (default). If you change the color of dimension lines subsequently, similar color is also used for dimension texts.</p> <p>To use separate colors for dimension lines and dimension texts, clear the Connected To Text check box.</p>
Offset Point	<p>Changes the offset distance between a geometry feature and the dimensioning extension lines.</p>

See [Dimension Line Settings on page 308](#) for an explanation of these dimension line attributes.

Changing Dimension Line Terminator Settings

In the **Arrow** area on the **Line** settings of the **Annotation Properties** dialog box, you can change the type, size, and fill attributes of dimension line terminators. See [Terminator Settings on page 315](#) for an explanation of these terminator attributes.

Changing Dimension Text Settings

In the **Text** settings of the **Annotation Properties** dialog box, you can change the attributes of dimension text.

1. Begin by selecting the dimension text component or components whose attributes you want to change:
 - If you click the **All** check box under **Set**, the attribute changes apply to all dimension text components.
 - Alternatively, you can select any of the following in the **Set** list:
 - **Upper**
 - **Lower**
 - **Prefix**
 - **Postfix**
 - **Superfix**
 - **Subfix**

-
- Then select **Value** or **Tolerance** in the **Subset** box or click the **All Subsets** check box to select both **Value** and **Tolerance** (if applicable).
2. Change the attributes for the selected dimension text components. See [Dimension Text Settings on page 318](#) for an explanation of these attributes.

 **Note**

To use the same dimension text color for dimension lines, click the **Connected To Line** check box (default). If you change the color of dimension texts subsequently, similar color is also used for dimension lines.

To use separate colors for dimension texts and dimension lines, clear the **Connected To Line** check box.

Changing Dimension Unit Settings

In the **Units** settings of the **Annotation Properties** dialog box, you can change the settings of dimension units. See [Dimension Unit Settings on page 329](#) for an explanation of dimension unit settings.

Example

The example below shows how to change dimension units from millimeters to fractional inches.

[Figure 130 on page 346](#) shows the dimension before and after the change:

Figure 130. Changing Millimeters to Fractional Inches



Changing Dimension Text Ext Settings

In the **Text Ext** settings of the **Annotation Properties** dialog box, you can change the text extension settings of the dimension text.

1. In the **Extensions** area, click the **Change** check box for the required text extension from the following:
 - **Prefix**
 - **Postfix**
 - **Superfix**

- **Subfix**
2. In the **Tolerance** area, click the **Change** check box, and specify the revised settings for tolerance.

Modifying Dimensions

The following sections describe how to modify the layout of existing dimensions.

Modifying Dimension Lines

If dimension and extension lines are crowded or overlapped, you can:

- Break an extension or dimension line.
- Slant an extension line.
- Stagger an extension line.
- Reset an extension or dimension line.

Any breaks and staggers you make to a dimension line are retained until either:

- The dimension line is reset using  **Line Reset** under  **Modify** in the **Annotate** group on the **Modify** tab.
- The location of the dimension is moved to within the range of the break or stagger.

Use the handles of a dimension to quickly modify it:

Move the dimension and text	Click anywhere on the dimension and drag.
Move the dimension text	Hold SHIFT and drag the text.
Reattach the dimension	Click the square-shaped handle where the dimension is attached to the element and drag.
Change the direction of the arrows	Click the diamond-shaped handle.
Modify the stagger line	Click the square-shaped handle and drag.

To change the **Handles** settings:

1. Click **File** ► **Settings** ►  **System**.

The **System** dialog box opens.

2. Click **Handles**.

To modify a dimension line:

- Click **Modify** and then, in the **Annotate** group, click one of the following:

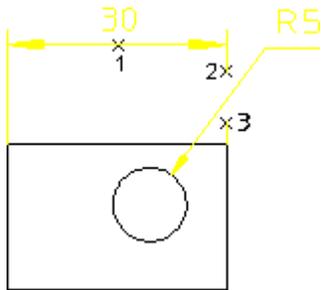
-  **Break**
-  **Stagger**
-  **Line Keep Text** under  **Modify**
-  **Line Move Text** under  **Modify**
-  **Line Reset** under  **Modify**

Breaking Dimension Lines

To break a dimension extension line around obstructing features:

1. Click **Modify** and then, in the **Annotate** group, click  **Break**.
Refer to [Figure 131](#) on page 348.

Figure 131. Broken Extension Line



2. Click the dimension text for the dimension (1).
Alternatively, you can preselect the dimension text.
3. Specify the start point for the break (2).
4. Specify the end point for the break (3).

 **Note**

- In the preselection mode, you can directly specify the start point of the break after you click  **Break**.
 - If preselection is active for the dimension, you can select another dimension to break.
 - If you specify a different option in the user input line after clicking  **Break** (for example, STAGGERED, BREAK, RESET, CHG_STAGGER), the preselected dimension is modified accordingly.
-

 **Note**

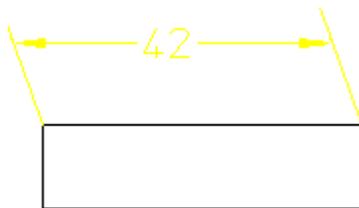
The break is lost after any modification of the geometry to which the dimension is attached.

Slanting Dimension Extension Lines

To slant dimension extension lines:

1. Click **Modify** and then, in the **Annotate** group, click  **Slant**. See the following figure:

Figure 132. Slanted Extension Lines



2. Type the slant angle in the user input line and press ENTER.
3. Select the dimension to slant.

Alternatively, you can preselect the dimension.

The dimension extension lines are slanted by the specified angle.

Note

- In the preselection mode, Creo Elements/Direct slants the dimension extension lines after you specify the slant angle.
 - If preselection is active for the dimension, you can select another dimension to slant.
 - If you specify a different option in the user input line after clicking  **Slant** (for example, STAGGERED, BREAK, RESET, CHG_STAGGER), the preselected dimension is modified accordingly.
-

Note

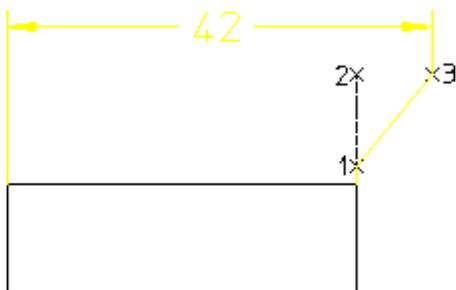
Slant angles must be between 0 and 60 degrees, or 0 and -60 degrees. You can only slant the extension lines of single dimensions with a parallel dimension orientation. Broken or staggered dimensions can be slanted and slanted dimensions can be broken or staggered. A slanted dimension cannot be further slanted.

Staggering Dimension Extension Lines

To stagger dimension extension lines:

1. Click **Modify** and then, in the **Annotate** group, click  **Stagger**.
Refer to [Figure 133 on page 350](#).

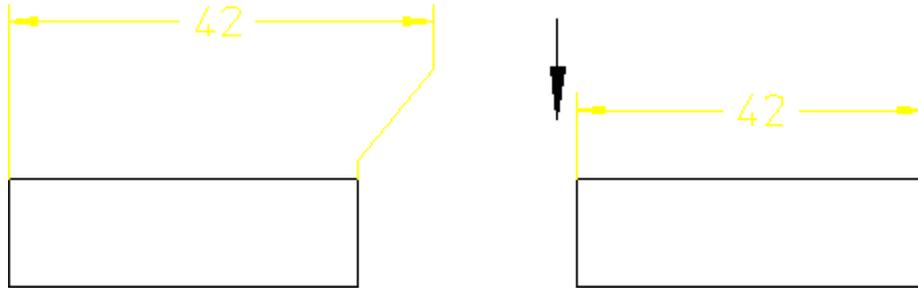
Figure 133. Staggered Extension Lines



2. Select the dimension.
Alternatively, you can preselect the dimension.
3. Specify the start point for the stagger on the extension line (1).

- Specify the end point for the stagger on the extension line (2).
- Specify the point where you want the stagger to go (3).

Figure 134. Effect of Moving Staggered Extension Lines



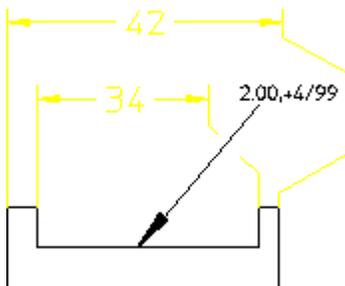
Note

- In the preselection mode, Creo Elements/Direct prompts you to specify the start point for the stagger after you click  **Stagger**.
- If preselection is active for the dimension, you can select another dimension to stagger.
- If you specify a different option in the user input line after clicking  **Stagger** (for example, BREAK, SLANT, RESET), the preselected dimension is modified accordingly.

Note

If a dimension contains several staggers, only the first is retained following a move. The staggers is lost after any modification of the geometry to which the dimension is attached.

Figure 135. Multiple Staggered Extension Lines



Modifying Dimension Orientation

You can modify the orientation of dimensions in a drawing.

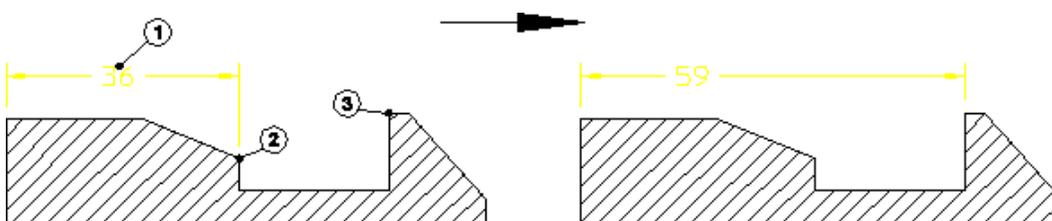
To modify the orientation of a dimension:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click one of the following:
 -  **Orientation Parallel** to orient the dimension parallel to an element.
 -  **Orientation Horizontal** to orient the dimension in the horizontal direction.
 -  **Orientation Vertical** to orient the dimension in the vertical direction.
 -  **Orientation Inclined** to orient the dimension at the specified inclination angle.
 -  **Orientation Perp to line** to orient the dimension perpendicular to a selected reference line.
 -  **Orientation Par to line** to orient the dimension parallel to a selected reference line.
3. Select the dimension.

Modifying a Dimension Vertex

The dimension vertex can be relocated to a new position without the need to click the first point again.

Figure 136. Relocating a Dimension Vertex



To modify a dimension vertex:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Line Keep Text**.

-
- Click the dimension (point 1).

Alternatively, you can preselect the dimension.

- Click the old reference element (point 2).
- Click the new reference element (point 3).

Creo Elements/Direct Drafting relocates the dimension but leaves the dimension text in its original position.

 **Note**

- If the dimension is preselected, you only need to specify the old and new reference points after you click  **Line Keep Text**.
- You can use the preselected dimension only once. At the next prompt, you must select another dimension or click  **Cancel** to end the command.

If you click  **Line Move Text** under  **Modify**, you can also relocate the dimension text:

- Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
- Click  **Line Move Text**.
- Click the dimension (point 1).

Alternatively, you can preselect the dimension.

- Click the old reference element (point 2).
- Click the new reference element (point 3).

Creo Elements/Direct Drafting relocates the dimension with the dimension text.

 **Note**

- If the dimension is preselected, you only need to specify the old and new reference points after you click  **Line Move Text**.
- You can use the preselected dimension only once. At the next prompt, you must select another dimension or click  **Cancel** to end the command.

Resetting Dimension Lines

To reset a broken or staggered dimension line:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Line Reset**.
3. Select the dimension for the dimension line reset.

Alternatively, you can preselect the dimension.

Creo Elements/Direct Drafting resets the broken or staggered dimension line.

Note

In the preselection mode, Creo Elements/Direct Drafting resets the broken or staggered dimension line after you click  **Line Reset**.

Note

If you reset a dimension with broken, staggered, or slanted dimension extension lines, Creo Elements/Direct Drafting straightens the extension lines and removes breaks or staggers, if any.

Modifying Dimension Text

This section shows how to move or rotate existing dimension text, with or without the associated dimension lines.

Moving Dimension Text

To move dimension text without moving the dimension or extension lines:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Move Text**.
3. Select the dimension text items you want to move.

Alternatively, you can preselect the dimension text.

Creo Elements/Direct Drafting replaces each selected dimension text item with a trace box.

4. Click the desired location for the dimension text.

Creo Elements/Direct Drafting relocates the selected dimension text items.

 **Note**

In the preselection mode, Creo Elements/Direct Drafting replaces each selected dimension text item with a trace box after you click  **Move Text**.

Moving Complete Dimensions

To move one or more complete dimensions:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Move Text & Line**.
3. Click the dimension text of the dimension or dimensions you want to move.
Creo Elements/Direct Drafting draws a trace of the dimensions.
Alternatively, you can preselect the text of the dimension that you want to move.
4. Click the desired location for the dimension text.
Creo Elements/Direct Drafting relocates the selected dimensions.

 **Note**

In the preselection mode, Creo Elements/Direct Drafting draws a trace of the dimension after you click  **Move Text & Line**.

Moving Dimension Text and Leader Line

To move a dimension text and its associated leader line:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.

-
2. Click  **Move Text & Leader**.
 3. Click the dimension whose text and leader lines you want to move. Creo Elements/Direct Drafting draws a trace of the dimension text and its leader line.

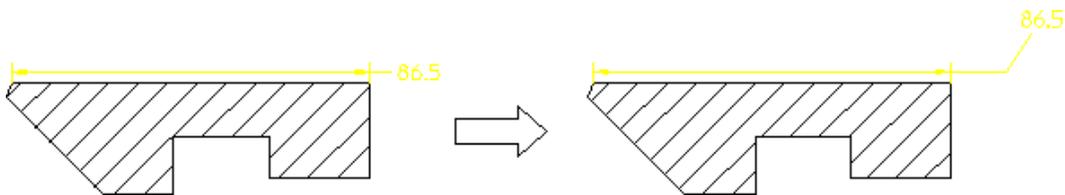
Alternatively, you can preselect the dimension text.

4. Click the desired location for the dimension text.
Creo Elements/Direct Drafting relocates the selected dimension text and its leader line.

Note

In the preselection mode, Creo Elements/Direct Drafting draws a trace of the dimension text and its leader line after you click  **Move Text & Leader**.

Figure 137. Moving a Dimension Text and Its Leader Line



Rotating Dimension Text

Horizontal is the zero angle and positive rotation is counterclockwise.

To rotate dimension text:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Rotate Text**.
3. Type an absolute angle of rotation in the user input line and press ENTER.
A positive angle causes a counterclockwise rotation.
4. Select the dimension text or texts you want to rotate.
Alternatively, you can preselect the dimension text.
Creo Elements/Direct Drafting rotates the selected dimension text.

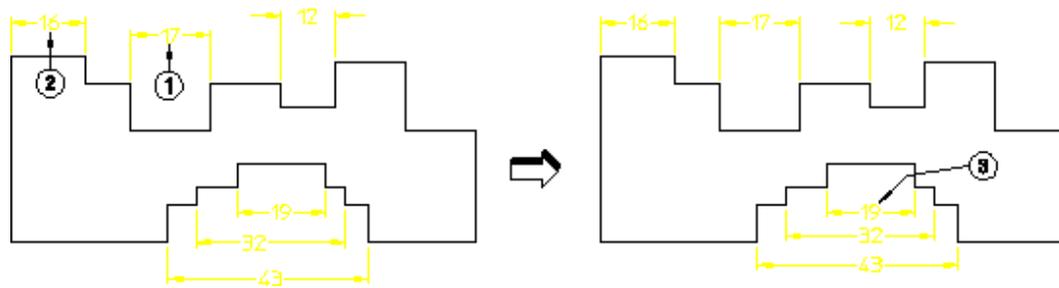
 **Note**

In the preselection mode, Creo Elements/Direct Drafting rotates the dimension text after you specify the rotation angle.

Aligning Dimensions

Dimensions can be aligned with other dimensions. First, you identify an existing dimension and align the new dimension above, below, or adjacent to it.

Figure 138. Aligning Dimensions to Other Dimensions



To align a dimension:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Move Text & Line**.
3. Click the dimension you want to move (point 1).
4. Click the reference dimension (point 2).
5. Drag the cursor to the new location.
6. Release the mouse button.
7. Repeat for other dimensions (point 3).

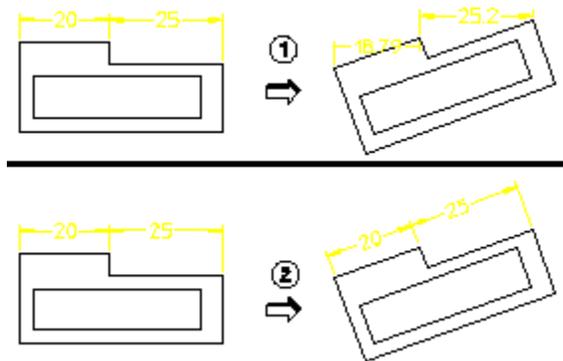
Aligning of dimensions applies to single dimensions and not group dimensions (a group of dimensions cannot be boxed because each dimension needs a specific shift-vector). Additionally, aligning dimensions does not apply to non-linear dimensions such as:

- Diameters
- Radii
- Arcs

Inclining Dimensions

The dimension attributes  Horizontal and  Vertical can be overridden such that when an element is inclined, the dimension is also inclined. What you are doing is freezing the dimension to the describing element. This helps speed up the modification process for dimensioned geometry. Dimension attributes such as text size and font are not changed.

Figure 139. Effects of Freezing a Dimension



- 1 Dimension created by  **Single**,  Horizontal. When the part is rotated 20°, the dimensions remain horizontal giving a correct but meaningless result.
- 2 Same as item 1 but the dimensions were frozen with  **Freeze** before the part was modified.

To freeze a dimension:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Freeze**.
3. Click the dimension you want to incline.

Alternatively, you can preselect a single dimension. You can use  **Freeze** for a single preselected dimension.

Note

- If you select the dimensions by enclosing them in a box, only those dimensions with the attribute  Horizontal or  Vertical are affected.
 - There is no  **Undo** for this command.
-

Editing Dimension Text

This section shows how to edit existing dimension text. Identified dimension text is displayed in the user input line. After editing the text, make sure that quotes (' ' or " ") surround the text, and then press ENTER.

Creo Elements/Direct Drafting redraws the edited dimension text.

To edit existing dimension text:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Edit Dim Text**.
3. Click the dimension text to edit.

Alternatively, you can preselect the dimension text.

Creo Elements/Direct Drafting displays the dimension text in the user input line.

4. Edit the text and press ENTER.
5. Click  **OK**.

Note

- In the preselection mode, Creo Elements/Direct Drafting displays the dimension text in the user input line after you click  **Edit Dim Text**.
 - You can preselect only a single dimension.
-

Dimension Text Underlining

Creo Elements/Direct Drafting provides the possibility to underline dimension text. This is particularly useful when you want to flag a dimension that has been manually changed (or edited). This underlining feature is a requirement of DIN dimensioning standards. The default setting depends on the selected dimension standard:

DIN Standard	Default is on.
ANSI, ISO, JIS, or COMPANY Standard	Default is off.

If the selected dimension standard is DIN, dimension text that is edited is automatically underlined. Otherwise, edited dimension text is not underlined but can be underlined selectively. Dimension underlines have the following properties:

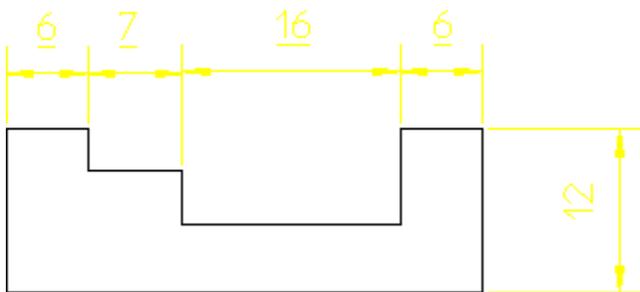
- The length is the same length as the dimension text.
- Prefixes, postfixes, and tolerances are not underlined.
- The color is the same as dimension lines.
- The space between the dimension text and underline is fixed.
- The pensize is the same as that of the dimension lines.

To underline a dimension:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Edit Dim Text**.
3. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
4. Click  **Underline**.
5. Click the dimension to be underlined.
6. Press ENTER to accept the current value displayed in the user input line.
Optionally, you can edit this value.

Creo Elements/Direct Drafting displays the edited dimension value with an underline.

Figure 140. Dimension Underlining



To remove an underline:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Cancel Edit**.
3. Click the underlined dimension you want to reset.

Creo Elements/Direct Drafting displays the edited dimension value without an underline.

Canceling a Dimension Text Edit

This section shows how to cancel an edit and display the dimension text in its original form. This means that if you edit the dimension text twice and then use  **Cancel Edit**, the original dimension text will appear and not your first edited version of the text.

To cancel an edit:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Cancel Edit**.
3. Click the dimension text you want to reset to its original version.

Creo Elements/Direct Drafting displays the original dimension text.

Alternatively, you can preselect the edited dimension text.

Note

In the preselection mode, Creo Elements/Direct Drafting displays the original dimension text after you click  **Cancel Edit**.

4. Click  **OK**.

Modifying the Color of Dimension Lines and Dimension Texts

Creo Elements/Direct Drafting allows you to change the color of dimension lines and dimension texts in a single operation. You can also separately change the color of the dimension lines and dimension texts.

To change the color of dimension lines and dimension texts in a single operation:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Dim Text and Line Color**.
The **Select Color** dialog box opens.
3. In the **Color** box, select the required color for dimension texts and dimension lines.
4. Click **OK**.

-
5. Select the dimensions to apply the color.

Alternatively, you can use the **Select** dialog box to select the elements that contain dimensions.

Creo Elements/Direct Drafting applies the specified color to the dimension lines and dimension texts of the selected dimensions.

 **Note**

After you select the elements, you can change the color of the dimension lines and dimension texts again, using  **Dim Text and Line Color**.

6. Click  **OK**.

To change the color of dimension lines only:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Dim Line Color**.

The **Select Color** dialog box opens.

3. In the **Color** box, select the required dimension color.
4. Click **OK**.
5. Select the dimension lines to apply the dimension line color.

Creo Elements/Direct Drafting applies the specified color to the selected dimension.

Alternatively, you can use the **Select** dialog box to select the elements that contain dimension lines.

Creo Elements/Direct Drafting applies the specified color to the selected dimension lines.

 **Note**

After you select the elements, you can change the dimension line color again using  **Dim Line Color**.

6. Click  **OK**.

To change the color of dimension texts only:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Dim Text Color**.
The **Select Color** dialog box opens.
3. In the **Color** box, select the required dimension text color.
4. Click **OK**.
5. Select the dimension to apply the color to the dimension text.

Alternatively, you can use the **Select** dialog box to select the elements that contain dimension texts.

Creo Elements/Direct Drafting applies the new color to the selected dimension texts.

Note

After you select the elements, you can change the dimension text color again using  **Dim Text Color**.

6. Click  **OK**.

Modifying Chain and Datum Dimensions

Individual segments can be inserted or deleted within chain and datum dimension groups. Creo Elements/Direct Drafting automatically adjusts other segments within the chain or datum group to account for the new/deleted segment. Optionally, Creo Elements/Direct Drafting can check to ensure that the modified chain or datum group will not intersect existing geometry.

Inserting a New Chain or Datum Dimension Segment

To insert a new segment into an existing chain or datum dimension group:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Insert**.

-
3. Click the chain or datum group you want to modify by clicking on the dimension text of any segment within the group.
 4. Click the new dimension point.
 5. When you have finished inserting segments, click  **OK**.

Deleting a Segment from a Chain or Datum Dimension Group

To delete a segment from an existing chain or datum dimension group:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Modify**.
2. Click  **Delete Member**.
3. Click the dimension text of the segment which you want to delete.
4. If the dimension text segment is a part of a dimension chain, click the neighboring dimension text in the chain.

This segment grows to fill the space created by the deleted segment.

5. Click  **OK** after you finish deleting dimensions.

Grouping

You can group annotations together. [Groups on page 241](#) may consist of a single dimension, multiple text elements and multiple symbols.

Further Information

For further technical information about dimensions, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- `help da_dimension`
- `help da_dim_line`
- `help da_dim_line_sym`
- `help da_dim_datum_long`
- `help da_dim_datum_short`
- `help da_dim_chain`

-
- `help da_dim_coord`
 - `help da_dim_chamfer`
 - `help da_dim_radius`
 - `help da_dim_diameter`
 - `help da_dim_dtan`
 - `help da_dim_arc`
 - `help da_dim_angle`
 - `help wui_dim_set_text_ext`
 - `help da_style_get`
 - `help da_style_apply`
 - `help da_dim_geo_sense`
 - `help da_dim_pd_scan`
 - `help wui_da_db_match`
 - `help wui_change_dim`
 - `help modify_dim_lines`
 - `help change_dim_vertex`
 - `help da_move_dimension`
 - `help rotate_dim_text`
 - `help da_dim_insert`
 - `help da_dim_incline`
 - `help edit_dim_text`
 - `help cancel_edit_dim_text`
 - `help wui_dimension_settings`
 - `help da_dim_delete`

Creo Elements/Direct Drafting displays further information about dimensioning.

11

Hatching a Drawing

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This chapter describes how to hatch your drawing and how to set and change the hatch parameters.

If you wish to cross-hatch an area or build up a custom hatch pattern, simply use any number of  **Hatch** buttons on the same area but with different hatch angles, distances, hatch points, colors, and linetypes.

Alternatively, you can define your own hatch patterns for use in multiple areas. See [Creating a Hatch Pattern on page 372](#) for details.

Hatching a Drawing Area

You have a choice of two hatch methods:

- [Automatic Hatching on page 367](#)
- [Manual Hatching on page 368](#)

Automatic Hatching

You can hatch any area fully enclosed by drawing lines or already hatched.

To automatically hatch a drawing area:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Automatic**.
3. Specify a point inside the area to be hatched.

The cursor automatically tracks around the enclosing polygon and then searches for any inner holes. Having established the inner and outer contours, Creo Elements/Direct Drafting hatches the bounded area. If Creo Elements/Direct Drafting finds any gaps or incorrectly split lines in the enclosing polygon, the command is aborted and Creo Elements/Direct Drafting marks the gap or split line.

Match Point

You can set a match point to be used for all subsequent hatching. One hatch line will be aligned to pass through the match point. The match point need not necessarily lie inside an area to be hatched; in this case an imaginary extension of the hatch lines would pass through the hatch point.

[Figure 141. Hatch Pattern Aligned with Match Point on page 367](#) shows the hatch pattern aligned with an external match point.

Figure 141. Hatch Pattern Aligned with Match Point



To set the hatch match point:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Hatch**.

-
2. Click  **Match Point**.
 3. Specify a point for the hatch line to pass through.

One hatch line of all subsequent hatching passes through the new match point.

Manual Hatching

Manual hatching is used to hatch any area not fully enclosed by drawing lines.

To manually hatch a drawing area:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Manual**.
3. Click the start point on the existing outer contour of the area to be hatched. See [Automatic Splitting on page 369](#) for further contour creation options.
4. Continue by clicking successive points around the outer contour.
5. Click the start point again to close the contour.

When you have closed the contour, use the same method to indicate any inner holes that are not to be hatched.

6. Click  **OK**.

The area is hatched.

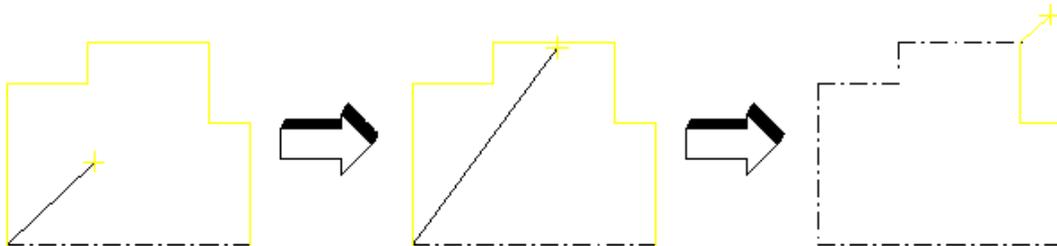
To abort the command click  **Cancel**. Use  **Undo** to undo erroneous points.

If a side consists of a complete drawing line, simply indicate a mid-point on the line using the element (**ELEM**) catching mode. Drafting will accept the line. If you wish to identify only part of a line, mark the desired end point with a construction line, use the intersection (**INTERSECTION**) catching mode, and indicate the desired end point. Drafting will then accept only part of the line.

A series of connected drawing lines can be indicated by the end of one line and the mid-point of the last line. Drafting will not accept a series of successive lines if there is a branch at one of the vertices.

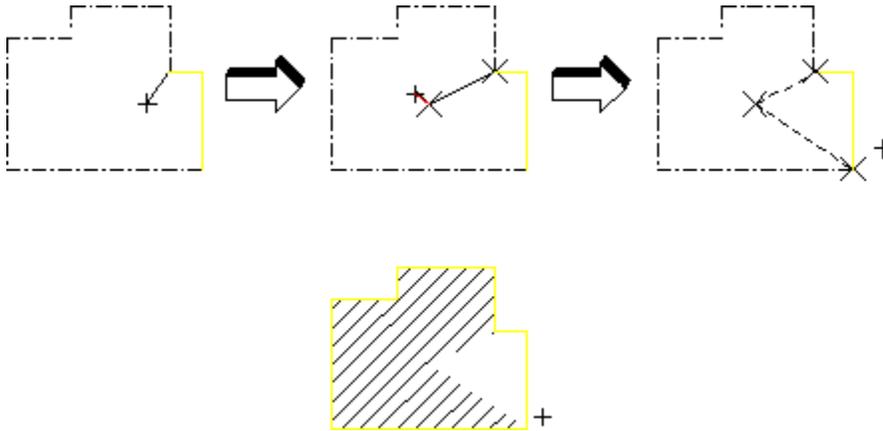
[Figure 142. Indicating the Area for Hatch on page 369](#) illustrates the above techniques.

Figure 142. Indicating the Area for Hatch



Not all the sides have to be drawing lines. An imaginary line can be used to separate free space or existing points. See [Figure 143. Hatching — Leaving an Unhatched Area](#) on page 369.

Figure 143. Hatching — Leaving an Unhatched Area



Automatic Splitting

You can use automatic splitting options for extended contour creation functionality.

To enable automatic splitting options:

1. Click **File** ► **Settings** ►  **Hatch**.

The **Hatch Settings** dialog box opens.

2. Click the **Automatic Split for Hatch Manual** check box.

Use these options to automatically detect element intersections and add only parts of elements to contours.

- Use `ANY_POINT` to automatically create a split point at the pick point on a selected element and use that element as the contour element if it is connected to previously selected element. All elements connecting currently picked and

previously picked elements are included in the contour. This option remains active until you select another option (INTERSECTION, ANY_POINT).

- Use INTERSECTION to automatically find the next intersection of elements on the contour and splits them both at that point. Drafting starts searching for an intersection along the contour in the same direction as between the previous and current pick points. All elements connecting the previously picked element and the intersection are included in the hatch contour. If Drafting doesn't find an intersection, this option behaves the same as the NO_SPLITTING option. This option remains active until you select another option (INTERSECTION, ANY_POINT).
- Use NO_SPLITTING to restore default behavior. The command behaves as if the **Automatic Split for Hatch Manual** setting is off. This option remains active until you select another option (INTERSECTION, ANY_POINT).
- Use VERTEX to automatically split element at the pick point, and directly connect it to your previous pick point without following the contour. This option remains active only for one pick.

Hatch Settings

The **Hatch Settings** dialog box enables you to define:

- [Simple Hatch Settings on page 370](#)
- [Setting Hatch Patterns on page 372](#)
- [Standard Hatch Patterns on page 378](#)

To open the **Hatch Settings** dialog box:

- Click **File** ► **Settings** ►  **Hatch**.

After you define your hatch settings, either click **Hatch** to apply these settings to the next face or faces you click, or click **OK** to apply these settings to subsequent hatching (while using the  **Hatch** buttons).

Simple Hatch Settings

Click **Hatch** in the **Hatch Settings** dialog box to enable the **Hatch** area for editing. In the **Hatch** area, you can select or change the following hatch settings:

- **Distance**
- **Angle**
- **Linetype**
- **Color**

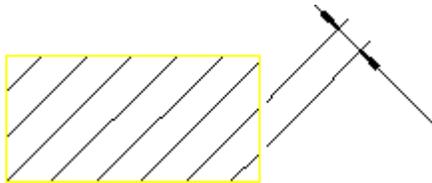
- **Ref. Point**

The settings you specify or select in the **Hatch Settings** dialog box are applied to all subsequent hatching. These settings remain active until you change them.

Hatch Distance

To set the distance between hatch lines, type the required value in the **Distance** box in the **Hatch** area. This value is interpreted in terms of the length unit (such as mm) currently set up for your Creo Elements/Direct Drafting environment. [Figure 144. Hatch Line Distance on page 371](#) shows an example of a hatch line distance.

Figure 144. Hatch Line Distance



Hatch Angle

To set the hatch angle, type the required value in the **Angle** box in the **Hatch** area. The angular units are those currently defined for your Creo Elements/Direct Drafting environment. A zero value implies horizontal hatch lines and a positive angle results in counterclockwise rotation.

[Figure 145. Positive Hatch Angle on page 371](#) shows a positive hatch angle.

Figure 145. Positive Hatch Angle



Hatch Linetype

To set the hatch linetype, select a linetype in the **Linetype** box in the **Hatch** area.

Hatch Color

To set the hatch color, select a color in the **Color** box in the **Hatch** area. Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

Setting Hatch Patterns

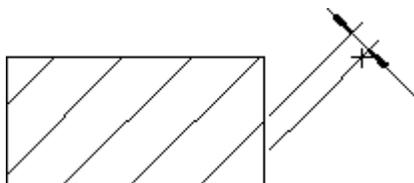
Patterns consist of any number of simple hatch lines built up in layers. After you create a pattern, it is active every time you click the  **Hatch** buttons in the **Annotate** group on the **Drafting** tab. The pattern is used to hatch all subsequent areas.

The five hatch pattern settings are described in [Table 3 on page](#) .

Table 3. Hatch Pattern Parameters

Name	Allowed Value	Description
Offset	0 to 1	The offset is used to align the hatch lines of the layer to the match point (set using Match Point). The perpendicular offset is the entered value multiplied by the distance (see Figure 146. Hatch Perpendicular Offset on page 372).
Distance	Any Number	The distance between each of the layer's hatch lines will be the entered value multiplied by the distance value specified in the Distance box in the Hatch area.
Angle	0 to ± 360	The angle is added to the angle specified in Angle box in the Hatch area to determine the actual angle at which the layer's hatch lines will be drawn.
Color	-	Any of predefined colors in the drop-down list box or a custom color. Refer to Defining Custom Colors on page 100 for instructions on defining custom colors.
Linetype	-	Any of the eight Drafting standard linetypes can be selected from a drop-down list box.

Figure 146. Hatch Perpendicular Offset



Creating a Hatch Pattern

To create a hatch pattern:

-
1. Click **Pattern** in the **Hatch Settings** dialog box to enable the **Pattern** area for editing.
 2. In the **Pattern** area, define the first layer's values for **Offset**, **Distance**, **Angle**, **Color** and **Linetype**.
 3. Define the next layer's values or finish the pattern by clicking **Add To**.

There is no limit to the number of layers in a pattern. For examples of patterns, refer to the section [Standard Hatch Patterns on page 378](#). The exact values required for each standard pattern are listed there.

Canceling a Hatch Pattern

You can cancel any previously created hatch pattern. After canceling the hatch pattern, all future  **Hatch** buttons in the **Annotate** group on the **Drafting** tab revert to the hatch parameters set using the linetype and color specified in the **Hatch** area (see [Simple Hatch Settings on page 370](#)).

To cancel a hatch pattern:

1. Click **Pattern** in the **Hatch Settings** dialog box to enable the **Pattern** area for editing.
2. Click **Cancel**.

The previously created hatch pattern is canceled.

Changing Hatch Settings

Creo Elements/Direct Drafting allows you to change existing hatching in your drawing using the  **Hatch** buttons in the **Annotate** group on the **Modify** tab:

- [Changing Individual Hatch Settings on page 373](#)
- [Changing a Hatch Pattern on page 376](#)

Changing Individual Hatch Settings

Use the  **Hatch** buttons in the **Annotate** group on the **Modify** tab to change individual hatch settings.

- [Changing the Hatch Angle on page 374](#)
- [Changing the Hatch Line Distance on page 374](#)
- [Changing the Hatch Color on page 374](#)
- [Changing the Match Point on page 375](#)

- [Changing a Hatch Pattern on page 376](#)
- [Automatic Splitting on page 369](#)

Changing the Hatch Angle

To change the hatch angle:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Angle**.
3. Type the required hatch angle in the user input line and press ENTER.
4. Click the hatch you want to change.

Creo Elements/Direct Drafting changes the selected hatching.

You can click individual areas of hatch or box several hatch areas. To change all the hatches, type ALL in the user input line and press ENTER.

Alternatively, use the  **Selection** buttons (as you would for a  **Delete** operation), and finish by clicking  **Confirm**.

Changing the Hatch Line Distance

To change the line distance of an existing hatch:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Distance**.
3. Type the required line distance in the user input line and press ENTER.
4. Select the hatching to be changed.

Creo Elements/Direct Drafting changes the selected hatching.

You can click individual areas of hatch or box several hatch areas. To change all the hatches, type ALL in the user input line and press ENTER.

Alternatively, use the  **Selection** buttons (as you would for a  **Delete** operation), and finish by clicking  **Confirm**.

Changing the Hatch Color

To change the color of an existing hatch:

-
1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
 2. Click  **Color**.

The **Select Color** dialog box opens.
 3. In the **Color** box, select a color.

Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.
 4. Select the hatching to be changed.

You can only select individual areas of hatch. When there are two hatch patterns superimposed on each other, the pattern nearest to the cursor is changed.

Alternatively, use the  **Selection** buttons (as you would for a  **Delete** operation), and finish by clicking  **Confirm**.

Changing the Match Point

To change the hatch match point:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Match Point**.
3. Type the required point for the hatch lines to match in the user input line and press ENTER.
4. Select the hatching to be changed.

Creo Elements/Direct Drafting changes the selected hatching.

You can click individual areas of hatch or box several hatch areas. To change all the hatches, type ALL in the user input line and press ENTER.

Alternatively, use the  **Selection** buttons as you would for a  **Delete** command, and finish by clicking  **Confirm**.

Automatic Splitting

See [Automatic splitting in the Manual Hatching on page 369](#) section.

Changing Multiple Hatch Settings

Instead of changing individual settings of a hatch, you can change multiple hatch settings at the same time.

To change multiple settings of a hatch:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Multiple**.
The **Change Hatch** dialog box opens.
3. Change the required hatch settings. (See [Simple Hatch Settings on page 370](#).)
4. Select the hatch.

Alternatively, use the **Select** buttons to select the hatch and click  **Confirm** to finish.

Changing a Hatch Pattern

To change a hatched area to a new pattern:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Pattern**.
3. Specify or select the settings for the first layer of the pattern (see [Setting Hatch Patterns on page 372](#) for details).
4. If there are further layers, continue by defining the next layer or layers in the same way.
5. Click **OK** in the **Hatch Settings** dialog box to finish defining the pattern.
6. Select the required hatched areas.

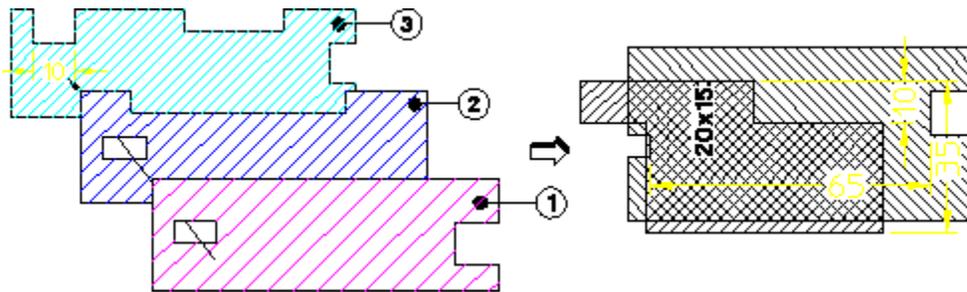
Creo Elements/Direct Drafting changes them to the new pattern.

Hatching Around Text and Dimension Text

When text lies in a hatched area, Creo Elements/Direct Drafting automatically creates a window around the text. The result is the same whether you hatch first and then add text, or add text first. If you modify the drawing by moving the text, the hatching is automatically updated.

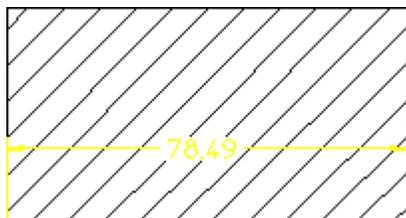
 **Text Window** creates a hole around text and dimension text when the text runs through a hatched area (as in overlapping parts). The following figure shows three parts in a parts structure (part 1 is the top part and part 3 is the lowest part). In a real assembly, the parts overlap and the hatching on parts 1 and 2 appears to run through the dimension text on part 1. Use  **Text Window** to punch a window through the overlapping hatching.

Figure 147. Text and Dimension Text: Hatched Area



1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Hatch the required area using  **Automatic** or  **Manual**.
3. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
4. Click  **Text Window**.
5. Click the dimension text around which you want to hatch.

Figure 148. Hatching Around Dimension Text



To move dimension text after using  **Text Window** to create the hole around the text:

1. Move the dimension text to its new position.
2. Delete the old hole in the hatching (Creo Elements/Direct Drafting deletes the hole and regenerates the hatch).

-
- Use  **Text Window** to create a new hole around the dimension text.

Standard Hatch Patterns

This section describes how to set up standard hatch patterns which are used by all subsequent  **Hatch** buttons in the **Annotate** group on the **Drafting** tab.

To set up a standard hatch pattern:

- Click **File** ► **Settings** ►  **Hatch**.
The **Hatch Settings** dialog box opens.
- Click **Standard** to enable the **Standard** area for editing.
- Click the required standard hatch pattern from the following:
 - **Iron**
 - **Steel**
 - **Copper**

The selected standard hatch pattern becomes active.

The three standard options are displayed below along with the values of **Offset**, **Distance**, **Angle**, **Color**, and **Linetype** used (refer to [Setting Hatch Patterns on page 372](#)). All the examples shown are drawn with the following hatch parameters:

- **Hatch Angle:** 45
- **Hatch Distance:** 5
- **Match Point:** 0,0

Standard Hatching — Iron

This option sets the following Hatch Pattern values:

Offset: 0

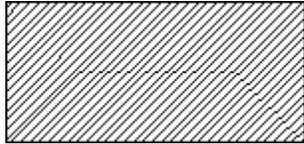
Distance: 1

Angle: 0

Color: Cyan

Linetype: Solid

Figure 149. Iron



Standard Hatching — Steel

This option sets the following Hatch Pattern values:

Offset: 0.33 & 0

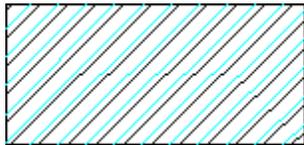
Distance: 1 & 1

Angle: 0 & 0

Color: Cyan & Cyan

Linetype: Solid & Solid

Figure 150. Steel



Standard Hatching — Copper

This option sets the following Hatch Pattern values:

Offset: 0.5 & 0

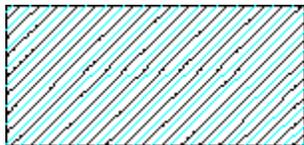
Distance: 1 & 1

Angle: 0 & 0

Color: Cyan & Cyan

Linetype: Dashed & Solid

Figure 151. Copper



Converting Hatch to Geometry

Hatches are handled differently by different applications. In some cases, hatch that is passed from Creo Elements/Direct Drafting to AutoCAD can change appearance. To ensure your hatch does not change appearance, convert it to geometry.

To convert a hatch to geometry:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Text To Geo**.
3. Select the required hatch to convert it to geometry.

Click in your viewport or use the **Select** window to select hatches in the current part. **Global** selection is not available.

4. Click  **Confirm**.
5. Click  **OK**.

The appearance of your hatching does not change. You can see, however, that it is now geometry. For example, you can now delete a single line of the hatch.

Note

Completely filled hatches are not converted. In this case, you will see the error message, "Completely filled hatches will not be converted to geometry" or "Some hatches will not be converted to geometry".

You can use  **Undo** or  **Redo** in combination with  **Text To Geo**.

Deleting Hatching

To delete an area of hatching:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Hatch**.
2. Click  **Delete**.
3. Click individual areas of hatching to be deleted, (or enclose a group of hatched areas in a box). Alternatively:

-
- Use the  **Selection** buttons as you would for a normal deletion.
 - Finish by clicking  **Confirm** to delete the selected groups of hatching.
4. If you delete an area by mistake, click  **Undo** to undo the last delete operation.
 5. Click  **OK**.

Delete hatch that has been converted to geometry the same way you would delete geometry by using  **Delete**.

Further Information

For further technical information about hatching, refer to the Creo Elements/Direct Drafting help system.

Type the following in the user input line and press ENTER:

- help hatch
- help hatch_ref_pt
- help change_hatch_angle
- help change_hatch_dist
- help change_hatch_ref_pt
- help change_hatch_pattern
- help wui_hatch_settings
- help hatch_to_geo
- help delete_hatch

Creo Elements/Direct Drafting displays further information about hatching.

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Adding Text to a Drawing

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Entering Text

Creo Elements/Direct Drafting provides the following options for entering text:

- [Entering Text Directly in the Drawing on page 383](#)
- [Entering Text Using a Text Box on page 384](#)

Note

If Creo Elements/Direct Drafting draws boxes instead of the text, you must input a text font. Refer to [Text Font on page 391](#). To load additional fonts for selection, follow the instructions in [Importing a Font on page 543](#).

Entering Text Directly in the Drawing

To enter text directly in your drawing:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **In Place**.
3. Click the position for the text in the viewport.
4. Type the required text.

Creo Elements/Direct Drafting adds the text directly at the cursor position. By default, the newly added text reflects the text settings defined using the [Text Settings on page 388](#) dialog box.
5. Press CTRL+D.
6. Insert further text in the drawing, if required.
7. Click  **OK**.

When you have entered text in your Creo Elements/Direct Drafting drawing, you can:

- Edit and/or delete this text (see [Editing and Deleting Text on page 386](#))
- Change the settings of this text (see [Changing Text Settings on page 393](#))
- Modify this text just like any geometric element (see [Modifying a Drawing on page 205](#))
- Add a reference line to this text (see [Creating a Reference Line on page 394](#))
- Convert the text to normal geometry (see [Converting Text to Geometry \(DXF/DWG and IGES 2D Translation\) on page 394](#))

Entering Text Using a Text Box

To add text to your drawing via a text box:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **Editor**.
The **Text** dialog box opens.
3. Type the required text in the **Text** box.
4. Click **OK** to add the text to the drawing.
5. Click the position for the text.

Creo Elements/Direct Drafting draws the text. The newly added text reflects the text settings defined via [Text Settings on page 388](#).

If you require the same text at several locations on the drawing, simply continue to click locations and the text will be repeated.

In addition to entering text, you can use the **Text** box for:

- [Entering Special Characters on page 384](#)
- [Text Settings on page 388](#)
- [Loading an Existing Text File on page 385](#)
- [Storing Text in a Text File on page 385](#)

Entering Special Characters

To add a special character to your Creo Elements/Direct Drafting drawing:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **Editor**.
The **Text** dialog box opens.
3. Click **Special Characters**.

The **Special Characters** dialog box opens. It displays a graphical list of special text characters.

These special characters are included in the `hp_symbols` and `hp_symbols2` fonts. You can create additional special characters by editing these fonts.

-
4. Select the special character you want to add as text to your drawing. A graphical representation of each character is provided in the **Meta Name** column.
 5. Click **Insert** to copy the selected special character to the **Text** box.

 **Note**

You can insert the special character or characters to the existing text in the **Text** box.

6. When the **Text** box contains the required special character or characters, click **OK**.
7. Click the position for the text in the drawing.

Loading an Existing Text File

To load an existing text file for use in your Creo Elements/Direct Drafting drawing:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **Editor**
The **Text** dialog box opens.
3. Click **Load**.
The **Load** dialog box opens
4. Use the Load browser to browse to the path of the required text file and select it.
5. In the **Text** box, edit the text as required.
6. Click **OK**.
7. Click the position for the text in the drawing.

Storing Text in a Text File

To store text in a text file:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **Editor**

The **Text** dialog box opens.

3. Type the text in the **Text** box or load an existing text file (see [Loading an Existing Text File on page 385](#)) and edit it.
4. Click **Store**.

The **Store** dialog box opens.

5. Using the Store browser, browse to the required directory and type the name of the text file in which you want to store the text.
6. Click **OK** to store your text in this file.

The text is saved in the specified file

Editing and Deleting Text

Creo Elements/Direct Drafting provides the following options for editing and deleting text in a drawing:

- [Editing Text Using a Text Box on page 387](#)
- [Editing Text Directly in the Drawing on page 386](#)
- [Moving Text on page 387](#)
- [Deleting Text on page 388](#)

Note

If Creo Elements/Direct Drafting draws boxes instead of the text, you must input a text font. Refer to [Text Font on page 391](#). To load additional fonts for selection, follow the instructions in [Importing a Font on page 543](#).

Editing Text Directly in the Drawing

To edit text directly or by using a text box in your drawing:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **In Place**.
3. Click the text that you want to edit.

Alternatively, you can preselect the text block to edit the text.

To edit the preselected text using the context menu:

- a. Right-click the preselected text.

-
- b. Click  **Edit Text**.
 4. Edit the text as required.

Any changes are performed at the cursor location. If you add text while editing, the newly added text will reflect the settings of the existing text (see [Text Settings on page 388](#)).
 5. Press CTRL+D.

Editing Text Using a Text Box

To edit existing text using a text box:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **Editor**.
3. Select the text in the drawing.

Alternatively, you can preselect the text block or blocks to edit the text.

The **Edit Text** dialog box opens. It contains the text that you have selected.
4. Make the required changes to the text and click **OK**.
5. Click  **OK**.

Note

In the preselection mode, the **Edit Text** dialog box opens after you click  **Editor**.

Moving Text

To quickly move text by moving the text box:

1. Hover the cursor over the text.

Creo Elements/Direct Drafting highlights a box around the text (if you have enabled fly-by highlighting).
2. Drag the text to a new location.

Alternatively:

1. Select the required text.

The Command Mini Toolbar appears.

-
- Click  **2 Points**.
 - Specify the reference point and then a destination point.
Creo Elements/Direct Drafting moves the text to the new position.

Deleting Text

There are several ways to delete text from your drawing.

To delete the text using the keyboard:

- Click the text.
A frame appears around the text.
- Press DELETE.

Creo Elements/Direct Drafting deletes text in the same way as normal drawing lines. To delete text without using the selection block:

- Click  **Delete**.
- Box the text to be deleted, or click a single text to be deleted. Creo Elements/Direct Drafting deletes the selected text.

If you use the **Text** option in the **Select** window, the text blocks you select for deletion are highlighted. Click **Confirm** to delete the highlighted text.

Click  **Undo** to correct mistakes and then repeat the operation or end the command.

To delete text using the  **Selection** buttons:

- Click  **Delete**.
- Select text for deletion using  **Select** buttons (similar to operation of  **Delete**). You can click  **Undo** to correct any mistakes.
- Click  **Confirm** to delete the selected text block or blocks.

Text Settings

To open the **Text Settings** dialog box:

- Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Text**.

-
2. Click  **Editor**.

The **Text** dialog box opens.

3. Click **Settings**.

The **Text Settings** dialog box opens.

Alternatively, click **File** ► **Settings** ► **Text** to open the **Text Settings** dialog box.

The settings you specify or select in the **Text Settings** dialog box are applied to all subsequent text. These settings remain active until you change them.

The **Text Settings** dialog box enables you to define:

- [Text Parameters on page 389](#)
- [Text Font on page 391](#)
- [Text Options on page 391](#)

Text Parameters

In the **Parameters** area of the **Text Settings** dialog box, set the following text parameters:

- **Size**
- **Angle**
- **Ratio**
- **Linespace**
- **Slant**
- **Fill**

Text Character Size

In the **Size** box, type the required value to set the text character size.

This value is interpreted in terms of the length unit (such as mm) currently set up for your CoCreate Drafting environment.

The Creo Elements/Direct Drafting default text size is 3.5 mm.

Text Angle

In the **Angle** box, type the required value to set the text angle.

The angular units are interpreted in terms of the angle units currently defined for your Creo Elements/Direct Drafting environment. A zero value implies horizontal text and a positive angle results in counterclockwise rotation.

[Figure 152. Setting the Text Angle on page 390](#) illustrates the text angle concept.

Figure 152. Setting the Text Angle



Text Character Ratio

In the **Ratio** box, type the required value to set the width-to-height ratio of text characters.

The default text ratio is 1.0.

Linespace

In the **Linespace** box, type the required value to set the line spacing of text lines.

The distance between lines of text in a block is computed as follows:

Actual line spacing = Linespace * Text Size

The default linespace in Creo Elements/Direct Drafting is 2.2 mm.

Text Character Slant

In the **Slant** box, type the required value to set the slant angle of text characters.

The angular units are interpreted in terms of the angle units currently defined for your Creo Elements/Direct Drafting environment. A zero value implies no slant and a positive angle results in clockwise rotation.

The default slant in Creo Elements/Direct Drafting is 0.

Figure 153. Text with a 15° Slant on page 390 shows text with a 15° slant.

Figure 153. Text with a 15° Slant



Text Character Fill On/Off

Creo Elements/Direct Drafting provides a number of text fonts (such as " hp_block_c ") that can be displayed in filled or unfilled mode.

Click or clear the **Fill** check box to enable or disable the fill mode, respectively.

[Figure 154. Unfilled and Filled Text on page 391](#) shows an example (unfilled and filled).

Figure 154. Unfilled and Filled Text



Text Font

In the **Text Settings** dialog box, select the required font in the **Name** box.

The font names in this selection list include:

- Fonts supplied with Creo Elements/Direct Drafting.
- Fonts already installed on your PC.

To load additional fonts for selection, follow the instructions in [Importing a Font on page 543](#).

OSD's different internal and external encoding schemes were [Unicode on page 782](#) incorporated for version 2007.

Note

The font selection list also includes all TrueType® fonts installed on your PC. If you select one of these fonts, it will be converted during the load process. The converted font will be stored in the fonts subdirectory below the Creo Elements/Direct Drafting installation directory.

Text Options

In the **Options** area of the **Text Settings** dialog box, set the following text options:

- **Color**
- **Frame**
- **Origin Vertex**

Text Color

To set the text color, in the **Color** box, select the required color.

Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

Text Frame

By default, text in Creo Elements/Direct Drafting is displayed without a frame.

Optionally, In the **Frame** box, select a frame from the following:

- **Box**

[Figure 155. Text with a Box Frame on page 392](#) shows text with a box frame.

Figure 155. Text with a Box Frame



- **Balloon**

[Figure 156. Text with a Balloon Frame on page 392](#) shows text with a balloon frame.

Figure 156. Text with a Balloon Frame



- **Flag**

[Figure 157. Text with a Flag Frame on page 392](#) shows text with a flag frame.

Figure 157. Text with a Flag Frame

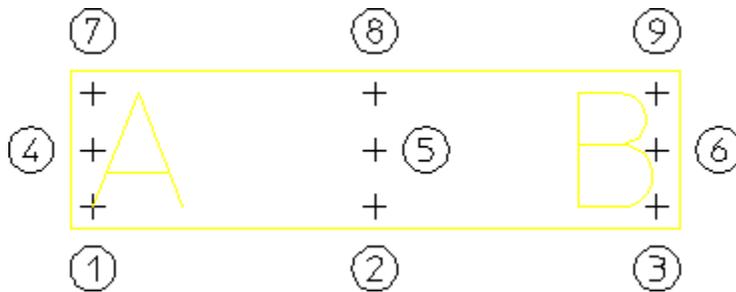


Text Origin Vertex

To set the position of the origin of any subsequent text, select the associated position number (an integer between 1 and 9) in the **Origin Vertex** box.

[Figure 158. Cursor Positions in the Text Box on page 393](#) shows you the relative position associated with each of these adjust values.

Figure 158. Cursor Positions in the Text Box



Changing Text Settings

To change the settings of existing text:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **Modify**.
3. Select the text that you want to change.

You can click a single text block or enclose multiple text blocks in a box or use the **Select** window.

The **Text Change** dialog box opens.

Alternatively, you can preselect the text to open the **Text Change** dialog box.

Note

In the preselection mode, the **Text Change** dialog box opens after you click

 **Modify**.

4. Change the settings as required.
See the [Text Settings on page 388](#) section for an explanation of these settings.

Note

The settings in the **Text Settings** dialog box apply to all text to be added to the drawing, while the settings in the **Text Change** dialog box apply to the selected existing text.

Converting Text to Geometry (DXF/DWG and IGES 2D Translation)

Creo Elements/Direct Drafting allows you to convert text entities to geometry entities using  **Text To Geo**. This is necessary for translation to DXF/DWG and IGES 2D format when customized fonts have been used (the MI file stores the font name but does not contain the customized font information).

To convert text entities to geometry entities:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Text**.
2. Click  **Text To Geo**.

Note

- Only text entities can be converted, except dimension texts.
 - Any text-windows inside hatched areas are retained.
-

Creating a Reference Line

Creo Elements/Direct Drafting allows you to create a reference line between a source element and its associated target element using the **Reference Line** buttons. The **Reference Line** buttons let you create a reference line between a source element and its associated target element. Creating reference lines helps to speed-up the modification phase as it saves you time repositioning individual elements. When you move the target element of a reference line, the reference line remains attached to the target element like a "rubber band" (see [Figure 159. Effect of Moving the Target Element on page 396](#)).

To draw a reference between two existing elements:

1. Click **Drafting** and then, in the **Annotate** group, click **More**.
2. Under **Reference Line**, click the required button.

Reference Line Button	Description
 Parallel	The reference line is drawn parallel to the orientation direction of the source element.
 Horizontal	A horizontal reference line is drawn from the source element.
 Vertical	A vertical reference line is drawn from the source element.

Reference Line Button	Description
 Perpendicular	The reference line is drawn perpendicular to the orientation direction of the source element.
 Reference Angle	This button prompts you for an angle. Type the required angle into the user input line and press ENTER. The angular units are interpreted in terms of the angle units currently defined for your Creo Elements/Direct Drafting environment. A zero value implies horizontal text and a positive angle results in counterclockwise rotation.
 Attach Source	This button enables you to attach the reference line to its source element. When you move the source element, the reference line remains attached to this element like a "rubber band".
 Underline	The source element is underlined.
 Invisible	An invisible reference line is drawn.

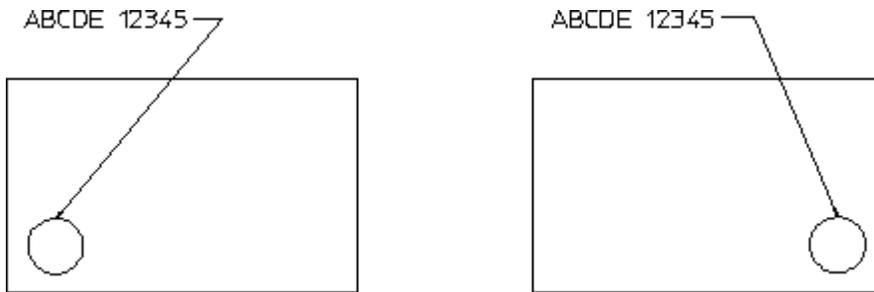
Note

When you draw a multi-segment reference line (by clicking points between the source element and its target element), note that these buttons apply to the first reference line segment (between the source element and the first reference line point you click) only.

3. Click the associated source element (such as an existing text block).
4. Optionally, click a number of points through which you want the reference line to pass.
5. Click the target element (the element to which the source element refers).

Creo Elements/Direct Drafting draws a reference line between the two elements. The newly added reference line reflects the settings defined via [Reference Line Settings on page 396](#).

Figure 159. Effect of Moving the Target Element



For deletion purposes, even a multi-segment reference line is considered as a single element and can be deleted in one operation.

Modify a Reference Line

To quickly modify the reference line, use the handles as follows:

To reattach the reference line, click the handle on the start or end of the reference line and drag.

To stretch reference line segments, click the reference line handles and drag.

To change the handle settings click **File** ► **Settings** ►  **System** and then click **Handles**.

Delete a Reference Line

To delete a reference line:

1. Click the reference line.
2. Press DELETE.

Reference Line Settings

To open the **Reference Line Settings** dialog box:

- Click **File** ► **Settings** ►  **Refline**.

The **Reference Line Settings** dialog box enables you to define:

- [Line Settings on page 396](#)
- [Reference Line Terminator on page 397](#)

Line Settings

In the **Line** area of the **Reference Line Settings** dialog box, set the following:

-
- **Type**
 - **Color**
 - **Width**

Type

To set the linetype of reference lines, select a linetype in the **Type** box in the **Line** area of the **Reference Line Settings** dialog box.

Color

To set the reference line color, select a color in the **Color** box in the **Line** area of the **Reference Line Settings** dialog box. Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

Width

To set the reference line width, type the required width in the **Width** box in the **Line** area of the **Reference Line Settings** dialog box. Refer to [Line Width vs. Line Size on page 95](#) for further information on line width.

Reference Line Terminator

In the **Terminator** area of the **Reference Line Settings** dialog box, set the following:

- **Type**
- **Size**

Type

To set the reference line terminator type, select one of the following options in the **Type** box in the **Terminator** area of the **Reference Line Settings** dialog box:

- **None**
- **Arrow**
- **Dot**
- **Slash**
- **Triangle**
- **Jis**

Size

To set the reference terminator size width, type the required value in the **Size** box in the **Terminator** area of the **Reference Line Settings** dialog box. This value is interpreted in terms of the length unit (such as mm) currently set up for your Creo Elements/Direct Drafting environment.

Reference Line Options

In the **Options** area of the **Reference Line Settings** dialog box, set the following:

- **Source Gap**
- **Destination Gap**
- **Offset**

All three values are interpreted in terms of the length unit (such as mm) currently set up for your Creo Elements/Direct Drafting environment.

Source Gap

To set a distance between the begin point of the reference line and its source element, type the required value in the **Source Gap** box in the **Options** area of the **Reference Line Settings** dialog box.

Destination Gap

To set a distance between the end point of the reference line and its target element, type the required value in the **Destination Gap** box in the **Options** area of the **Reference Line Settings** dialog box.

Offset

To set a distance between the begin point of the reference line and its source element as well as between its end point and its target element, type the required value in the **Offset** box in the **Options** section of **Reference Line Settings**.

Getting and Putting Text Properties

Any properties you assigned to a text block (see [Text Settings on page 388](#) and [Changing Text Settings on page 393](#)) can be assigned to another text block.

To assign the properties of a text block to another text block:

1. In the **Utilities** group, click  **Properties**.
2. Click  **Get**.

-
3. Click the text block whose properties you want to assign.
 4. In the **Utilities** group, click  **Properties**.
 5. Click  **Put**.
 6. Click the text block to which you want to assign these properties.

Font Editor

The Creo Elements/Direct Drafting Font Editor enables you to modify existing Creo Elements/Direct Drafting fonts and create new fonts. The Creo Elements/Direct Drafting Font Editor is particularly useful in design environments where company-specific characters or logos are required and for Japanese Kanji character-sets.

When working with the Font Editor, note the following:

- Although you can use the buttons in the **View** group to do such things as a redraw, do not change to another menu until you are finished.
- Store the current drawing (MI file).
- When you have finished creating or modifying the font, you must save it.
- All loaded fonts are listed in a selection table.
- The font you are editing becomes the current font.

Caution

Before using  **Font** in the **Font Editor** group on the **Miscellaneous** tab, save the current drawing. This is because the font editor uses the current drawing area to display the font table and deletes anything in the current workspace.

Editing a Font

To edit a font:

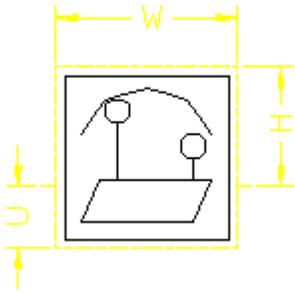
1. Click **Miscellaneous** and then, in the **Font Editor** group, click  **Font**.
The **Select Font** dialog box opens.
2. In the **List** box, select a font and click **Edit**.
Details of the selected font appear in the viewport.
3. Click **Miscellaneous** and then, in the **Font Editor** group, click  **Show Font**.

All the characters of the selected font are displayed in the Creo Elements/
Direct Drafting drawing area.

4. Click **Drafting** and then, in the **Font Editor** group, click the arrow next to  **Character**.
5. Click  **Edit**.
6. Type the character number (for example 174) which you want to edit in the user input line and press ENTER.

Creo Elements/Direct Drafting displays the character and a yellow character frame.

Character Frame



Note

- If you use this method to add special characters to the `hp_symbols` or `hp_symbols2` font, be sure to edit an empty character (indicated by a blank square).
- These special characters are available for use in text, dimensions, and symbols.

7. Modify or delete the geometry of the character.
8. Click **Miscellaneous** and then, in the **Font Editor** group, click the arrow next to  **Draw**.
9. Use the  **Draw** buttons ( **Line**,  **Rectangle**,  **Polygon**,  **Circle**,  **Arc**, or  **Point**) to create a new character.

 **Note**

The maximum number of unconcatenated elements in a character is 84.

10. Click **Miscellaneous** and then, in the **Font Editor** group, click the arrow next to  **Character**.

11. Click  **Update**.

12. Type the character number in the user input line and press ENTER.

You can retain the same character number or assign a new character number.

You can repeat the procedure for other characters, if required.

 **Note**

If you use this method to create additional special characters in the `hp_symbols` or `hp_symbols2` font, be sure to save the edited version under a unique name in a separate directory so that it can be reused following a Creo Elements/Direct Drafting update.

Changing the Character Spacing

To change the spacing between characters:

1. Click **Miscellaneous** and then, in the **Font Editor** group, click the arrow next to

 **Change**.

2. Click one of the following:

-  **Normal Distance** to change the distance between constant-spaced characters.
-  **Constant Width** to change the distance between variable-spaced characters.

After you change the character spacing, Creo Elements/Direct Drafting draws subsequent text blocks with the new character spacing.

Characters with Constant Spacing

A A A A A
↔ ↔

A A A A A
↔ ↔

Characters with Variable Spacing

A | 1 0 2
↔ ↔

A | 1 1 0 2
↔ ↔

Grouping

You can group annotations together. [Groups on page 241](#) may consist of a single dimension, multiple text elements and multiple symbols.

Further Information

For further technical information about text, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- `help text`
- `help help wui_enter_text`
- `help wui_change_text`
- `help wui_text_settings`
- `help text_to_geo`
- `help text_split`
- `help refline`
- `help wui_refline_settings`
- `help leader_line`

Creo Elements/Direct Drafting displays further information about text.

13

Adding Symbols to a Drawing

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Geometric Tolerancing Symbols

This section describes how to add geometric tolerancing symbols to your drawing. The symbols are based on the `hp_symbols2` font. When you add symbols to a drawing, Creo Elements/Direct Drafting uses this font.

Setting Symbol Size

The size of symbols and associated text is determined by the **Size** setting in the **Parameters** area of the **Text Settings** dialog box. Therefore, before you start adding symbols to your drawing, set the desired text size as described in [Adding Text to a Drawing on page 382](#).

To change the size of symbols and associated text, use the **Text Change** dialog box settings as described in [Adding Text to a Drawing on page 382](#).

Deleting and Moving Symbols

Deleting Symbols

There are two ways to delete a symbol:

1. Click the symbol.
A frame appears around the symbol.
2. Press DELETE.

Alternatively:

1. In the **Utilities** group, click  **Delete**.
2. Select the element that you want to delete.

Moving Symbols

There are two ways to move a symbol:

1. Move the symbol box.
 - Hover the cursor over the symbol. Creo Elements/Direct Drafting highlights a box around the symbol (if the **Enable Fly-By Highlighting** check box is clicked in the **System** dialog box).
 - Drag the symbol to a new location.
2. Click **Modify** and then, in the **General** group, click the required  **Move** buttons.

Move the symbol just as you would move a text, a line, or an arc (see [Moving Elements on page 209](#)).

Building a Geometric Tolerance Group

Click **Drafting** and then in the **Annotate** group, click  **Symbol**

The **Symbols** dialog box opens. The settings in this dialog box allow you to quickly build up the desired geometric tolerance group.

The main settings are:

Setting	Description
Tolerance	Sets the Normal or Composite symbol mode.
Type	Enables you to select a symbol type.
Symbol	Provides a preview of the current symbol definition in a rectangular area.
Symbol Browser Bar	Displays predefined symbols of the selected Type .
Add Text	Displays the Add Text dialog box for adding text to a symbol.
	Appends a short vertical line to the symbol.
	Adds a new line to the symbol.
	Appends the] character to the symbol.
	Undoes the last addition to the symbol.
Apply	Applies the current symbol definition. Simply position the symbol in the drawing.
Cancel	Cancels the current symbol definition.

To draw a geometric tolerance group:

1. Click **Drafting** and then in the **Annotate** group, click  **Symbol**.

The **Symbols** dialog box opens.

2. In the **Tolerance** box, select a symbol mode:
 - **Normal**
 - **Composite**
3. In the **Type** box, select a symbol type:

- **Geometric Symbols**
- **Modifying Symbols**
- **Draws Datum Target and Revision Symbols**

Symbols of the selected type are displayed on the Symbol Browser bar.

4. Click a symbol on the Symbol Browser bar.

This symbol is displayed in the **Symbol** preview area.

5. Optionally, click **Add Text** and type the text to be added to the symbol in the **Add Text** box. Use all upper case letters. Click **OK** to close the text box and append the text to the symbol.
6. Click , or , or  to append vertical and/or horizontal lines and/or] characters to the symbol.
7. Click **Apply** when the symbol displayed in the preview area is complete.
8. Position the symbol in the Creo Elements/Direct Drafting drawing.

 **Note**

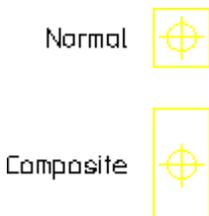
A copy of the same symbol is added to the drawing every time you click a point in the drawing.

9. Click  **OK** to terminate the positioning of the symbol.
10. Optionally, attach a reference line which connects the symbol to the associated feature in the Creo Elements/Direct Drafting drawing. See [Creating a Reference Line on page 394](#) for instructions.

Normal or Composite Symbols

Figure 160. [Position Geometric Symbol on page 406](#) shows an example of a **Position** geometric symbol. Both the **Normal** and **Composite** options are shown.

Figure 160. Position Geometric Symbol



Drawing a Geometric Symbol

In the **Symbols** dialog box, select **Geometric Symbols** in the **Type** box to draw geometric symbols.

Form Symbols

Figure 161. Symbol Type and Form Geometric Symbols on page 407 shows the four form symbols.

Figure 161. Symbol Type and Form Geometric Symbols

①	②	③	④
			
			

- 1 Straight
- 2 Flat
- 3 Round
- 4 Cylinder

Profile Symbol

Figure 162. Profile Geometric Symbols on page 407 shows the two profile symbols.

Figure 162. Profile Geometric Symbols

Line 

Surface 

Location Symbols

Figure 163. Location Geometric Symbol on page 408 shows the three location symbols.

Figure 163. Location Geometric Symbol

1	2	3
		

- 1 Position
- 2 Symmetry
- 3 Concentric

Alignment Symbols

Figure 164. Alignment Geometric Symbols on page 408 shows the three alignment symbols.

Figure 164. Alignment Geometric Symbols

1	2	3
		

- 1 Angular
- 2 Perpendicular
- 3 Parallel

Runout Symbols

Figure 165. Runout Geometric Symbols on page 408 shows the two runout symbols.

Figure 165. Runout Geometric Symbols

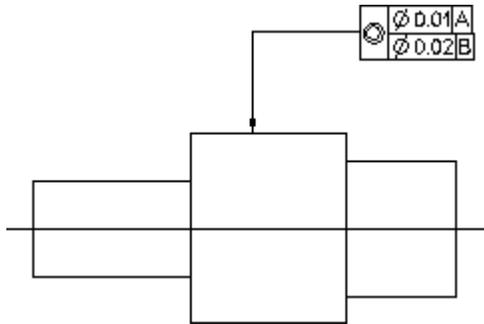
1	2
	

- 1 Circular
- 2 Total

Appending Text

Text can be added to symbols with **Add Text**. Use this command to interrupt the symbol group, add text, and then proceed with the symbols. Use all upper case letters. The two examples in [Symbol Examples on page 410](#) show you the correct sequence of steps.

Figure 166. Adding Symbol Text



Drawing Modifiers

In the **Symbols** dialog box, select **Modifying Symbols** in the **Type** box to draw any of the symbols shown in [Figure 167. Modifier Geometric Symbols on page 409](#).

Figure 167. Modifier Geometric Symbols

Diameter		
M		At Maximum Material Condition
L		At Least Material Condition
S		Regardless of Feature Size
P		Projected Tolerance Zone

Drawing Datum Target and Revision Symbols

In the **Symbols** dialog box, select **Draws Datum Target and Revision Symbols** in the **Type** box to draw any of the symbols shown in [Figure 168. Target and Revision Symbols on page 410](#).

Figure 168. Target and Revision Symbols

①	②	③
⊖	×	△

- 1 Symbol
- 2 Point
- 3 Revision

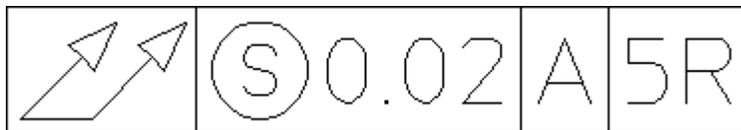
Symbol Examples

Drawing symbols is a simple task and can quickly be mastered by following specified steps.

Before starting, set the text size to 5 mm, the text adjust to 1 (refer to [Adding Text to a Drawing on page 382](#) for details).

Example 1: Single Line Symbol Group

Figure 169. Symbol Group – Example 1



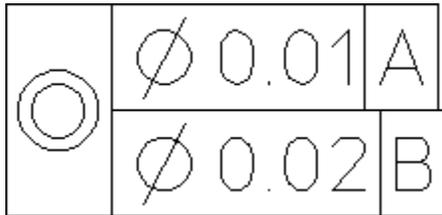
To add symbol text:

1. Click **Drafting** and then , in the **Annotate** group, click  **Symbol**.
The **Symbols** dialog box opens.
2. In the **Tolerance** box, select **Normal**.
3. In the **Type** box, select **Geometric Symbols**.
4. Click the  (Total Runout) symbol on the Symbol Browser bar.
5. In the **Type** box, select **Modifying Symbols**.
6. Click the  symbol on the Symbol Browser bar.
7. Click **Add Text** and type 0 . 02 in the box.
8. Click **OK**.
9. Click .
10. Click **Add Text** and type A in the box.
11. Click **OK**.

12. Click .
13. Click **Add Text** and type 5R in the box.
14. Click **OK**.
15. Click .
16. Click **Apply**.
17. Specify a point to position the symbol in the Creo Elements/Direct Drafting drawing.

Example 2: Multiline Symbol Group

Figure 170. Symbol Group – Example 2



To add symbol text:

1. Click **Drafting** and then in the **Annotate** group, click  **Symbol**.
The **Symbols** dialog box opens.
2. In the **Tolerance** box, select **Composite**.
3. In the **Type** box, select **Geometric Symbols**.
4. Click  (Concentric) symbol on the Symbol Browser bar.
5. In the **Type** box, select **Modifying Symbols**.
6. Click  (Diameter) symbol on the Symbol Browser bar.
7. Click **Add Text** and type 0 . 01 in the box.
8. Click **OK**.
9. Click .
10. Click **Add Text** and type A in the box.
11. Click **OK**.
12. Click .
13. Click  to initiate a second line in the symbol.

14. Click  (Diameter) symbol on the Symbol Browser bar.
15. Click **Add Text** and type 0 . 02 in the box.
16. Click **OK**.
17. Click .
18. Click **Add Text** and type B in the box.
19. Click **OK**.
20. Click .
21. Click **Apply**.
22. Specify a point to position the symbol in the Creo Elements/Direct Drafting drawing.

Example 3: Editing a Symbol Group

Creo Elements/Direct Drafting allows you to edit Symbol groups using  **Edit**. It converts the symbol into a readable text-string and displays it in a text box for editing. This text-string also contains control characters (\030) that should not be edited. If they are modified, the conversion link back to the symbols font is lost and the symbol appears as normal text.

To edit the symbol text:

1. Click **Modify** and then, in the **Annotate** group, click the arrow next to  **Symbols**.
2. Click  **Edit**.
3. Click the symbol you want to edit.

Creo Elements/Direct Drafting displays the text in a standard text editor, similar to the following:

```
\030\151e0.01|A}
\030fe0.02|B}
```

4. Edit the text and click **File ► Save** in the standard text editor to save your changes.

Surface Symbols

To open the **Surface Symbols** dialog box:

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Symbol**.

2. Click **Tolerance**.

The **TOLERANCE SYMBOLS** dialog box opens.

3. Click **Menu 2**.

The **Surface Symbols** dialog box opens.

In the **TOLERANCE SYMBOLS** dialog box, under **Functional Areas**, the following buttons are available:

Button	Description
	Load or save a symbol set.
	Load, save, or delete individual symbols in the current symbol set.
	<p>Display the positioning menu for the current symbols.</p> <p>After you click , the following Positioning buttons are available:</p> <p>Note</p> <p>The positioning point marks the foot point of the symbol.</p>
	The symbol is drawn horizontally from the positioning point to the right.
	The symbol is drawn vertically upwards from the positioning point.
	The symbol is drawn vertically downwards from the positioning point.
	The symbol is drawn from the positioning point (first point) towards the second point.
	The symbol is drawn from the positioning point (first point) at the specified angle.

Button	Description
	Remove, read or replace existing symbols in the drawing.
	Display the setup menus for surface symbols.

Setting Up Surface Symbols

To define the settings for surface symbols:

- In the **TOLERANCE SYMBOLS** dialog box, under **Functional Areas**, click



The **SETUP 1 SURFACE** dialog box opens.

You can define the following settings in the **SETUP 1 SURFACE** dialog box:

- Surface Symbols**

File	Filename of the currently loaded symbol set or, following changes, the name of the symbol set to be loaded.
Font	Select the font size in the Font Size box for surface symbols. If you prefer to use your own font size, specify the applicable number in the user input line and press ENTER. Note that italic fonts are marked by the letter <i>i</i> preceding the <code>font</code> extension. For example, the italic version of the <code>hp_symbols2</code> font is named <code>hp_symbols2_i.font</code> .
Color	Select the color in the Symbol Color box for surface symbols. Optionally, you can type RGB values in the user input line and press ENTER
Text Angle	Select a text character slant angle in the Text Slant box.

- Reference Lines**

Color	Select the color for reference lines in the RefLinColor box. Optionally, you can type RGB values in the user input line and press ENTER.
Width	Select the reference line width in the Line Size box. Optionally, you can specify a numerical value for the reference line width in the user input line and press ENTER.  Note The Width setting applies to both, reference lines and symbols.
Automatic	Toggle to enable or disable the automatic addition of reference lines.

- **Reference Lines Symbol**

Shape	Select a point, an arrow, a triangle or a dash as the terminator for reference lines in the Shape box.
Size	Select the reference line terminator size in the Size box. Optionally, you can type a numerical value for the reference line terminator size in the user input line and press ENTER.

- **Angle Correction**

Ref Line	Select the maximum deviation angle (in degrees) at which the reference line snaps to the horizontal or vertical direction in the Correction box. The default value of 45 degrees implies that the reference line is always drawn horizontally or vertically. If you want to draw reference lines at arbitrary angles, select 0.
-----------------	--

The "surf_sym" Symbol Set Supplied with Creo Elements/Direct Drafting

The `surf_sym` symbol set supplied with Creo Elements/Direct Drafting includes the following preconfigured surface symbols:

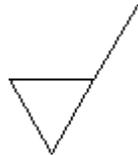
coated



milled



mat_remov



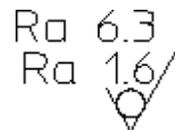
no_mat_remov



smooth



raw



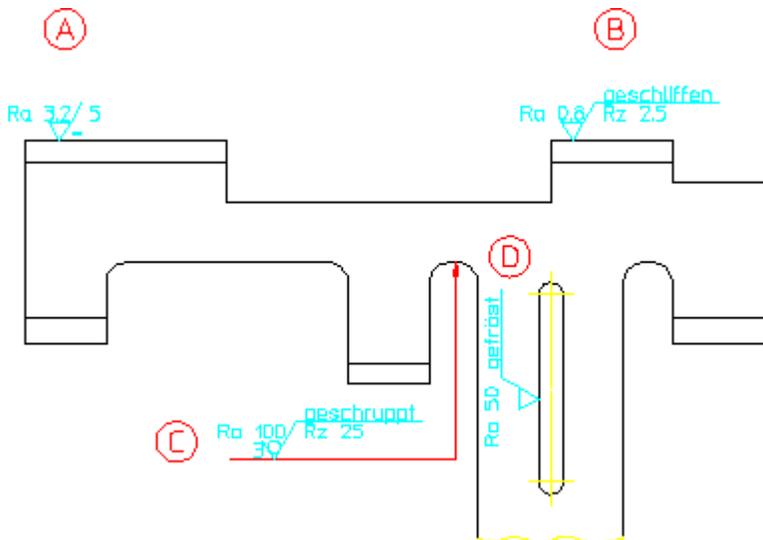
Use **Load** to load this symbol set.

Illustrated Examples of Surface Symbols

To create your own surface symbols, either modify one of the preconfigured symbols supplied with Creo Elements/Direct Drafting or create symbols "from scratch". Both methods are illustrated in the following section.

In the following example, four surface symbols are added to the drawing of a casting.

Casting plus Surface Symbols



The following exercise illustrated the procedure for creating the surface symbols ((A)), ((B)), ((C)) and ((D)).

Symbol ((A))

1. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Symbol**.
2. Click **Tolerance**.
The **TOLERANCE SYMBOLS** dialog box opens.
3. Under **Compose Menu**, click **S** to display the current symbol definitions.
The **Symbol Form** box opens.
4. Click .

Note

The extension line running to the right is drawn only if you specify additional symbol-related data.

5. Under **Compose Menu**, click **a2**.
The **Roughness** box opens.
6. Select Ra 3.2.
7. Under **Compose Menu**, click **c**.

The **Reference Distance** box opens.

8. Select 5.
9. Under **Compose Menu**, click **d**.

The **Lay Direction** box opens.

10. Select =.

11. Under **Functional Areas**, click .

You do not need to specify any positioning options because the symbol is in the default direction (from left to right).

12. Specify the positioning point (the foot point) for symbol ((A)) in your drawing.

Symbol ((B))

1. Load the smooth surface symbol.
2. Under **Compose Menu**, click **d**.

The **Lay Direction** box opens.

3. Select **EMPTY**.
4. Under **Compose Menu**, click **a2**.

The **Roughness** box opens.

5. Type 'Ra 0, 8' in the user input line and press ENTER.
6. Under **Compose Menu**, click **c**.
7. Type " (a blank space) in the user input line and press ENTER.
8. Under the **Compose Menu**, click **f1**.
9. Type 'Rz 2, 5' in the user input line and press ENTER.
10. Position symbol ((B)) in your drawing.

Symbol ((C))

1. Click .

 **Note**

When you select a different symbol type, the existing settings are preserved in the **Compose Menu**. Therefore, any settings for a different symbol type require explicit changes. To leave any symbol element blank, you either specify a space character (in apostrophes) or you can click **EMPTY**.

2. Under **Compose Menu**, click **d**.
The **Lay Direction** box opens.
3. Select **EMPTY**.
4. Under **Compose Menu**, click **a2**.
5. Type 'Ra 100' in the user input line and press ENTER.
6. Under **Compose Menu**, click **b**.
The **Manuf Process Surface Treat** box opens.
7. Select **grinded**.
8. Under **Compose Menu**, click **e**.
The **Manuf Allowance** box opens.
9. Select 3.
10. Under **Compose Menu**, click **f1**.
11. Select Rz 25.
12. Position the symbol at a suitable point to the left of and below the area to marked for grinding.
13. To select the start point of the reference line, click a point to the left of the symbol foot point.
14. To select the flexion point of the reference line, click a point close to a line which is perpendicular to the area to be marked for grinding.
15. To select the start point of the reference line arrow, click a point on the area to be marked for grinding.
16. Click  **Confirm**.

Symbol ((D))

1. Load the milled surface symbol.
2. Under **Compose Menu**, click **a1**.

The **Roughness** box opens.

3. Select Ra 50.

4. Click  to position symbol ((D)) in your drawing.

Welding Symbols

To open the **Welding Symbols** dialog box:

1. Click **Drafting** and then, in the **Annotate** tab, click the arrow next to  **Symbol**.
2. Click **Welding**.

Composing a Welding Symbol

To compose a welding symbol, click the applicable button under the **Compose Menu** or select the associated items of the symbol (indicated graphically in the menu).

Note

To familiarize yourself with the symbol composition method, load one of the welding symbols supplied with Creo Elements/Direct Drafting and check the resulting welding symbol in your drawing with its associated items in the **Compose Menu**.

Text

To view the associated **Text Menu**:

- In the **Welding Symbols** dialog box, under **Welding Symbols**, click **Text**.

You can specify the individual text strings for your welding symbols either as default strings in submenus, or you can specify text strings (in apostrophes).

Note

The following applies to fillet weld offsets:

- Specify the following if you want the Z to cross the reference line:
 \v
 - Specify the following if you do not want the Z to cross the reference line:
 \v
-

Symbol Elements

The following welding symbol elements are provided to facilitate the creation of welding symbols:

Item	Description
Sy 1	Welding symbol
Sy 2	Welding symbol
Sy 3	Welding symbol
Sy 4	Supplementary symbol
Sy 5	Supplementary symbol
R Sm	Opens the Ring Seam box.
M Sm	Opens the Mounting Seam box.
Fork	Opens the Fork box.
--	Relative position of the reference extension line. The following options are available: up: The reference extension line is located above the reference line. down: The reference extension line is located below the reference line. without: No reference extension line.

Positioning Welding Symbols

1. Click .

2. Select the applicable positioning icon.

 **Note**

Unless you explicitly select a positioning icon, the symbol is drawn from left to right. This default is associated with the  icon.

3. Click a point in your drawing to add the welding symbol.
4. Draw the reference line for your welding symbol. Follow the instructions in the following section.

Menu 2

Menu 2 for welding symbols provides a number of functions similar to those for surface symbols plus a number of welding-related functions.

Welding Seams

- : selects fillet weld symbols.
- : indicates weld surfaces in your drawing.

Text Functions

Use **Text Functions** to add text in the font size of the current drawing. Click a button, type the text (in single quotes or double quotes) in the user input line and press ENTER.



Adds text without a frame.



Adds text in a rectangular box.



Adds text in a balloon.



Enables you to move text added by one of these three text frame options.

Reference Line

The **Reference Line** function enables you to manually create a reference line for the current symbol.

The reference line always runs from the object to the symbol.

To use the **Reference Line** function:

1. On **Menu 2**, under **Reference Line**, click .
The begin point of the reference line is automatically attached to the symbol.
2. Click any points along the reference line and any bend points.
3. Click the end point of the reference line.
4. To complete the reference line, click  **Confirm**.

Note

To break (interrupt) the reference line, click **Free**.

To delete the last point on the reference line, click **Back**.

Stretch Symbols



Stretches symbols vertically.



Stretches symbols horizontally.

Construction Lines



Creates a construction line at an angle.



Creates a horizontal construction line.



Creates a vertical construction line.

Setting Up Welding Symbols

The text, color, and reference line setup options for welding symbols are similar to those for surface symbols. See [Setting Up Surface Symbols on page 414](#).

Additionally, you can also define the following settings:

- **Welding Seams**

Color	Select the color of fillet weld and weld surface symbols in the Symbol Color box. Optionally, you can type RGB values in the user input line and press ENTER.
Hatch Dist	Select the hatch distance for fillet weld symbols in the LinDist box. If you want to draw fillet weld symbols "filled", select this distance as 0.
Line Type	Select the linetype for fillet welds and weld surfaces in the Line Type box.

The "weld_sym" Symbol Set Supplied with Creo Elements/Direct Drafting

The weld_sym symbol set supplied with Creo Elements/Direct Drafting includes the following preconfigured welding symbols:

- V-U-weld



- V_back



- fillet_back



- fillet_ring



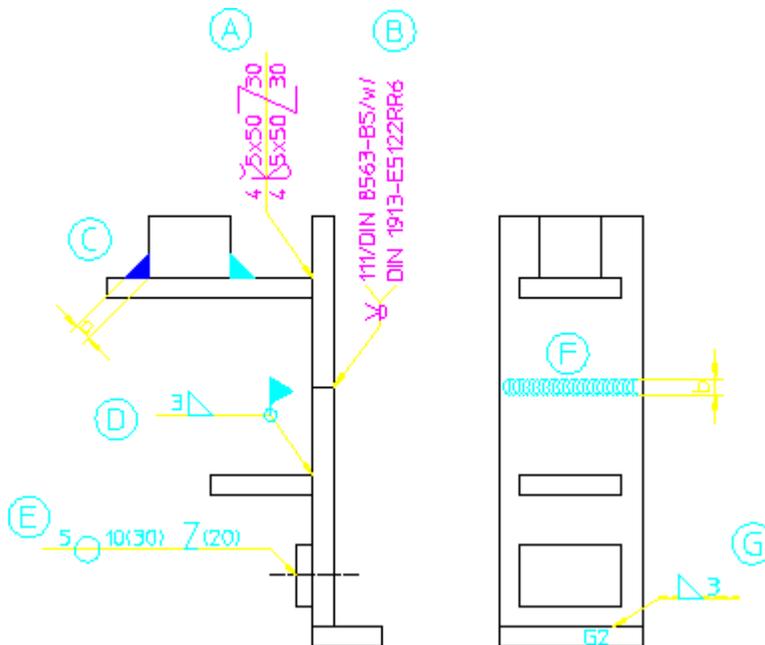
Use **Load** to load this symbol set.

Illustrated Examples of Welding Symbols

To create your own symbols, either modify one of the preconfigured symbols supplied with Creo Elements/Direct Drafting or create symbols "from scratch". Both methods are illustrated in the following section.

In the following example, a number of welding symbols are added to a simple sheet metal design.

Sheet Metal Design plus Welding Symbols



The following exercise illustrated the procedure for creating the welding symbols ((A)), ((B)), ((C)), ((D)), ((E)), ((F)) and ((G)). This exercise assumes that the **Welding Symbols** dialog box is open.

Symbol ((A))

1. Load the kehl_gegen welding symbol.

2. Under **Functional Areas**, click .

The **Positioning** box opens.

3. Click .
4. Click the positioning point for the symbol.
5. Click a point vertically above the symbol positioning point.
6. Click the reference line end point.

Symbol ((B))

1. Load the ∇ _back welding symbol.
2. Position this welding symbol and its reference line in your drawing.

Symbol ((C))

1. In the **WELDING SYMBOLS** dialog box, click **Menu 2 ▶ SETUP**.
2. Under **Welding Seam**, click **Hatch Dist**.

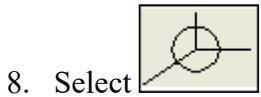
The **LinDist** box opens.

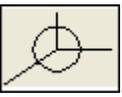
3. Select 0.
4. On **Menu 2**, under **Welding Seams**, click .
5. Type 4 as the fillet weld width in the user input line and press ENTER.
6. Click the vertex for the "left" leg of the fillet weld.
7. Click the first leg.
8. Click the second leg.
9. Repeat the earlier four steps to define the "right" fillet weld.

Symbol ((D))

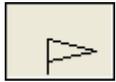
1. In the **Welding Symbols** dialog box, under **Welding Symbols**, click **Text**.
2. Under **Compose Menu**, click **T 1**.
3. Type 3 in the user input line and press ENTER.
4. Under **Text Menu**, click **Sym**.
5. Under **Compose Menu**, click **Sy 2**.
6. Select the delta symbol.
7. Under **Compose Menu**, click **R Sm**.

The **Ring Seam** box opens.



8. Select .
9. Under **Compose Menu**, click **M Sm**.

The **Mounting Seam** box opens.



10. Select .
11. Position the symbol and its reference line.

Note

The **SETUP Weld Symb** dialog box enables you to select defaults for the size of the symbol elements.

Symbol ((E))

1. Load the `zwr_pnt_n` welding symbol.
2. In the **Welding Symbols** dialog box, under **Welding Symbols**, click **Text**.
3. Under **Compose Menu**, click **T 1**.
4. Type '5' in the user input line and press ENTER.
5. Under **Compose Menu**, click **T 2**.
6. Type '10 (30) \v (20)' in the user input line and press ENTER.



7. Click .
8. Click the positioning points for the symbol.
9. To define the reference line direction, click a point horizontally above and to the left of the symbol positioning points.
10. Click the reference line end point.

Symbol ((F))

1. In the **Welding Symbols** dialog box, click **Menu 2**.



2. Under **Welding Seams**, click .
3. Type 5 as the weld contour width in the user input line and press ENTER.
4. Type 4 as the weld contour spacing factor in the user input line and press ENTER.

-
5. Click the start point of the weld.
 6. Click the end point of the weld.

Symbol ((G))

1. In the **Welding Symbols** dialog box, under **Welding Symbols**, click **Text**.
2. Under **Compose Menu**, click **T 1**.
3. Type '3' in the user input line and press ENTER.
4. Under **Text Menu**, click **Sym**.
5. Under **Compose Menu**, click **Sy 2**.
The **Weld Symbol** box opens.
6. Select the delta symbol.
7. Under **Compose Menu**, click **--**.
The **Position** box opens.
8. Select **up**.
9. Position the symbol such that its reference line points to the left.

Grouping

You can group annotations together. [Groups on page 241](#) may consist of a single dimension, multiple text elements and multiple symbols.

Further Information

For further technical information about symbols, refer to the Creo Elements/Direct Drafting help system.

Type the following in the user input line and press ENTER:

- `help edit_text`
- `help wui_spec_char_browser`

Creo Elements/Direct Drafting displays further information about symbols.

14

Handling Parts and Assemblies

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This chapter describes how to use parts in your drawing.

Introduction to Parts

Parts are most useful in large complex drawings. Therefore, in your first few weeks of using Creo Elements/Direct Drafting you may not want to complicate the drawing process by using parts. However, as you become a proficient user, you will begin to see how the use of parts will simplify complex drawings and increase your productivity. Using parts in a drawing is a matter of personal choice; different users will use a different parts structure on the same drawing.

With these points in mind, now consider some ideas and definitions of parts.

What is a Part?

A part is a logical grouping of elements, for example, a group of elements that make up a component in a sub-assembly.

A drawing created without the use of parts consists of a number of lines, circles, arcs, pieces of text, and so on. Each line, circle, arc, and so on, is known as an element.

If you use parts in a drawing, each part is a collection of elements. Thus, without parts, a drawing consists of a collection of elements, but with parts a drawing consists of several parts, each part being a collection of elements.

The Top Part

Remember that without parts a drawing is a collection of elements. You could also say that a drawing without parts is in effect a drawing containing one part; the part and the drawing are one and the same. Each part has a name (more details are given later in this section), and the name of the part you are using is displayed in the status bar. In a drawing without parts you will see the name associated with the top part (default name `TOP`) displayed in the status bar; all the elements of the drawing are in this single part.

In an assembly drawing certain parts will contain drawing elements, a name, and possible subparts. Other parts contain only a name and possible subparts, these parts are used as pointers to assemblies.

Parts Structure Explained

A parts structure can be thought of as a family tree showing the hierarchical relationships of all the parts. Because of this hierarchical nature, we talk about the different levels of the structure. At the highest level a single part exists; this is the `TOP` part already discussed. For your first few drawings using parts, it is best to use a parts structure with only one level; more experienced users will use a multi-level structure. The two types of parts structure are shown below:

Figure 171. Single-Level Parts Structure

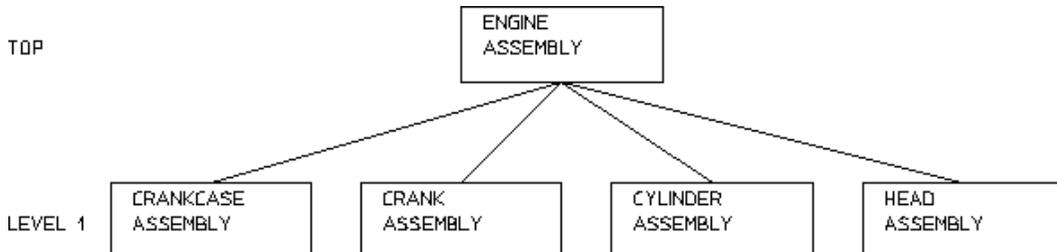
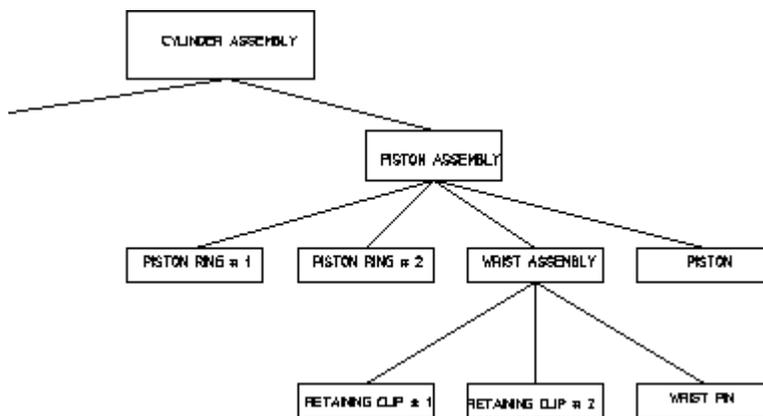


Figure 172. Multi-Level Parts Structure



Hierarchical Parts Structure

The hierarchical structure of parts means that one part can own other parts. Refer to the CYLINDER ASSEMBLY hierarchy shown in [Figure 181 on page 431](#). The parts structure is as follows:

PISTON ASSEMBLY can be thought of as the owner of PISTON, PISTON RING # 1, PISTON RING # 2, and WRIST ASSEMBLY.

We refer to the owning part (PISTON ASSEMBLY) as the parent and the owned parts (PISTON, PISTON RING # 1, PISTON RING # 2, and WRIST ASSEMBLY) as subparts.

Note that the hierarchical level of the current part is indicated through an integer in the status bar.

Note

This ownership mechanism implies that deleting a parent part will also result in the deletion of all its subparts.

Setting the Active Part

Except for a few special cases, all newly created elements (lines, arcs, text, and so on) are placed in the active part. Also, most of the commands using the  **Select** buttons (for example,  **Delete** and **Change**) only operate on the active part. The  **Part** button under  **Edit** in the **Parts** group on the **Structure** tab allows you to set the active part. Thus, by using the  **Edit** buttons to make different parts active, you can place elements in different parts.

Note

The name of the active part is displayed in the status bar.

Rules for Part Names

Follow these rules when entering part names:

- Part names are limited to 255 characters in length.
- Any characters may be used in a part name with the exception of:
 - / (Only allowed as the name of the top part when not used together with other characters.)
 - ~
 - . (Invalid part name when used alone, but may be used together with other characters.)

Examples of impossible names are:

- .
- ..
- /hello
- hello~

Examples of possible names are:

- hello
- / (Can be used only for the top part.)
- Whenever possible, give your parts meaningful names. This helps to recognize the part.

 **Note**

Creo Elements/Direct Drafting can assign a default name to a part. The Creo Elements/Direct Drafting default name will be in the series .1, .2 , .3 and so on. See [Initiating a Part on page 441](#).

The Advantages of Parts

Using parts has the following advantages:

- You can arrange a complex drawing into a convenient logical structure.
-  **Modify**,  **Delete** and most of the commands using the  **Select** buttons can be used on complete parts, thus you need only identify the part and not every element.
- Parts can be stored in or loaded from a file. This allows you to build up libraries of standard parts.
- By making a master drawing into parts, two or more people can work on individual parts of the master drawing. You can also quickly build up a final master drawing.
- Special **Parts** buttons (such as the **Symbol** check box) allow you to create custom symbols.

Creating an Assembly Drawing

To make full use of the Creo Elements/Direct Drafting parts functionality you should create and maintain a parts and assemblies structure for your drawing files.

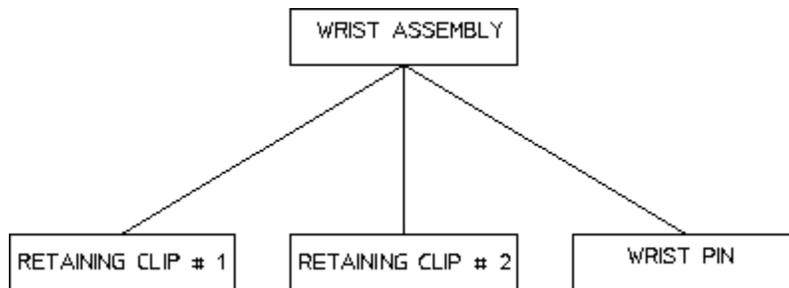
This approach to drawing facilitates the link between your Creo Elements/Direct Drafting design data and a relational database management system.

The following example shows one way to create the assembly drawing WRIST ASSEMBLY from the following component drawings:

- RETAINING CLIP # 1
- RETAINING CLIP #2
- WRIST PIN

[Figure 173 on page 434](#) shows the WRIST ASSEMBLY structure.

Figure 173. WRIST ASSEMBLY Structure



Initial Stages

For the purposes of this example, assume that each component drawing has not yet been given a part name.

Note

Drawings have two names associated with them:

1. The drawing storefile name — indicated in lower case.
 2. The drawing partname — indicated in CAPITALS.
-

First load each individual component drawing, give it a part name and set a reference point to enable accurate positioning in the final assembly drawing.

1. Load the drawing file `retain_clip1_d.mi`.
 - a. Click **File**  **Open**.

The **Open** dialog box opens.
 - b. Click `retain_clip1_d.mi`.

Creo Elements/Direct Drafting displays the drawing of RETAINING CLIP #1.
2. Set a reference point on the drawing to enable accurate positioning in the final assembly drawing.
 - a. Click **Structure** and then, in the **Parts** group, click  **Set Ref Pnt**.
 - b. Specify a reference point.
3. Delete all unnecessary construction lines.
 - a. In the **Utilities** group, click  **Delete**.

-
- b. In the **Utilities** group, click  **Select** ►  **Advanced**.
The **Select** window opens.
Alternatively, press F2 to directly open the **Select** window.
 - c. Click **CGeo** and then click **All**.
 - d. Click **OK**.
4. Provide a part name to the drawing.
- This part name is displayed in the **Part Browser**.
- a. Click **Structure** and then, in the **Parts** group, click  **Browser**.
The **Part Browser** opens.
Alternatively, press F12 to open the **Part Browser**.
 - b. In the **Part Browser**, right-click the part name you want to rename and click  **Rename**.
 - c. Rename the current part name as `RETAINING CLIP # 1`.
5. Store the new drawing (without construction lines).
- a. Click **File** ►  **Save**.
 - b. Type the new file name (for example `retainclip1`) and then click **Save**.

 **Note**

The `mi` suffix is appended automatically.

- c. In the **Utilities** group, click  **Delete**.
 - d. In the **Utilities** group, click  **Select** ►  **All**.
 - e. Click  **Confirm**.
6. Repeat these steps for the component drawings:
- `retain_clip2_d.mi`
 - `wrist_pin_d.mi`

Creating WRIST ASSEMBLY

Next create the assembly drawing WRIST ASSEMBLY:

1. Click  **Spotlight** in the status bar.

As a result, the active part has a normal appearance and all non-active parts are shown in magenta dotted lines.
2. Ensure that the Creo Elements/Direct Drafting top part is empty.
 - a. Click **Structure** and then in the **Parts** group, click  **Edit**.
 - b. Click  **Top**.
 - c. In the **Utilities** group, click  **Delete**.
 - d. In the **Utilities** group, click  **Select** ►  **All**.
 - e. Click  **Confirm**.
3. Rename the top part with the assembly drawing name you wish to see in the parts list.
 - a. Click **Structure** and then in the **Parts** group, click  **Browser**.

The **Part Browser** opens.
 - b. In the **Part Browser**, right-click **Top** and click  **Rename**.
 - c. Rename **Top** as WRIST ASSEMBLY.

The name of the active part in the status bar changes to WRIST ASSEMBLY. This part contains only the assembly name WRIST ASSEMBLY but no drawing elements.
4. Now load the first part of the assembly, for example the wrist pin.
 - a. Click **File** ► **Open** .
 - b. Select `wristpin` as the filename of the drawing to be loaded.

Creo Elements/Direct Drafting prompts you to indicate a reference point to position the drawing.
 - c. Specify a reference point.
5. Repeat steps 2 to 4 for the drawings:
 - `retainclip1.mi`
 - `retainclip2.mi`

-
6. Store the final assembly drawing as `wrist_assy`.

Creating Further Assemblies

The next stage in your assembly would be to create the `PISTON ASSEMBLY` drawing. The procedure is the same as previously described:

1. Create an empty top part in Creo Elements/Direct Drafting.
2. Rename the top part `PISTON ASSEMBLY`.
3. Construct `PISTON ASSEMBLY` from the following parts:
 - `PISTON RING # 1` — a single component part.
 - `PISTON RING # 2` — a single component part.
 - `WRIST ASSEMBLY` — an assembly.
 - `PISTON` — a single component part.

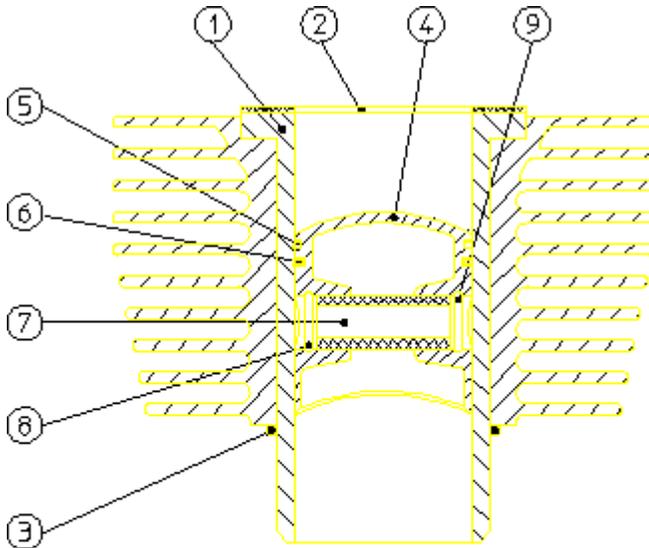
Creating a Final Engine Assembly Drawing

The ultimate industrial requirement would normally involve the creation of several assembly and sub-assembly drawings. In the case of our example, the following drawings would be incorporated into a final assembly drawing at the highest level of detail, for example `ENGINE ASSEMBLY`:

- `CRANKCASE ASSEMBLY`
- `CRANK ASSEMBLY`
- `HEAD ASSEMBLY`
- `CYLINDER ASSEMBLY` (which contains the `PISTON` and `WRIST` assemblies)

[Figure 174 on page 438](#) shows how the completed cylinder assembly drawing might appear:

Figure 174. Completed Cylinder Assembly Drawing

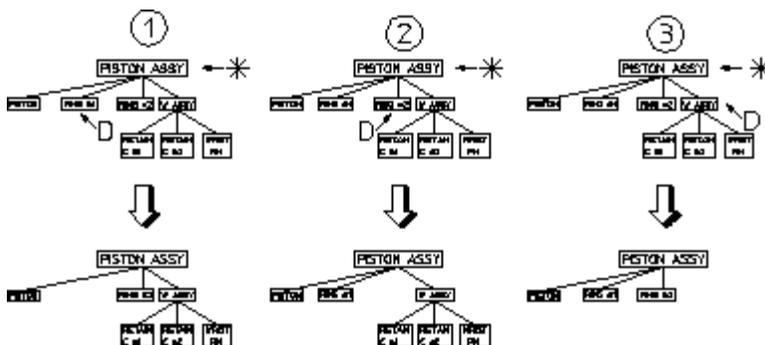


1	CYLINDER	6	PISTON RING #2
2	GASKET	7	WRIST PIN
3	BOTTOM 'O' RING	8	RETAIN CLIP #1
4	PISTON	9	RETAIN CLIP #2
5	PISTON RING #1		

Deleting Parts

Consider the CYLINDER ASSEMBLY hierarchy with PISTON ASSEMBLY active. Figure 175 on page 438 shows three examples of the deletion mechanism. The star shows the active part; the D shows that an element in the part was identified for deletion.

Figure 175. Deleting Parts



Note that in the third case:

-
- The entire WRIST ASSEMBLY would be deleted if any element of RETAIN CLIP 1, RETAIN CLIP 2, or WRIST PIN were identified.
 - If WRIST ASSEMBLY was made active and an element in RETAIN CLIP # 1 identified for deletion, only RETAIN CLIP # 1 would be deleted.

 **Note**

You cannot delete the current active part or the Top part. A link is always maintained to the Top part.

To delete parts:

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.
2. Click  **Part**.
3. In the parts hierarchy, select the appropriate part you want to make active.
The selection of the active part controls what is actually deleted.
4. Click  **Delete**.
5. Click the part or parts you wish to delete.

Modifying Parts

Creo Elements/Direct Drafting allows you to modify your parts using the buttons on the **Modify** tab.

Consider the CYLINDER ASSEMBLY hierarchy again. Suppose you wish to move the entire piston assembly, that is the piston, rings, wrist pin, and so on. The simplest way to identify all these is to first set a part above PISTON ASSEMBLY active, in this case CYLINDER ASSEMBLY:

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.
2. Click  **Part**.
3. Select the CYLINDER ASSEMBLY.

With CYLINDER ASSEMBLY active, you only have to identify an element in PISTON ASSEMBLY. Creo Elements/Direct Drafting accepts the assembly and all its subparts.

4. Complete the required move operation using the buttons on the **Modify** tab.

Note

To modify parts or elements above the active part you must click  **Select**  **Global**.

Editing Parts

This section describes how to set a part as active so that you can then change it. All new drawing elements created from then on belong to the active part.

Similarly, whichever part is active affects the results of the **Modify** operations and  **Delete**. If you have used  **Spotlight** (refer to [Spotlighting Parts on page 464](#)), the active part is shown in normal linetypes and colors, while all other parts are shown in magenta dotted lines. The name of the active part is displayed in the status bar.

Editing a Part

To edit a part (make a part active):

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.

2. Click  **Part**.

Creo Elements/Direct Drafting prompts for the name of a part.

3. Type the name of a part in the user input line and press ENTER.

Alternatively, select an element of the part in any viewport.

Creo Elements/Direct Drafting sets the named, or selected, part as the active part.

Editing Top

To edit the top part (set the highest part of the parts structure as the active part):

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.

2. Click  **Top**.

Creo Elements/Direct Drafting sets the Top part as the active part.

Editing the Parent

You can move the active part status to the parent of the currently active part. In relation to a given part, the parent is always one level higher in the parts structure.

To edit the parent part:

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.
2. Click  **Parent**.

Creo Elements/Direct Drafting sets the parent of the currently active part as the active part.

Initiating a New Empty Part

This section describes how to create a new empty part. The active-part status will be altered after the command has been executed so that the new part is active.

Initiating a Part

Create a new empty part under the Top part and make this new part the active part so you are ready to draw in it.

Note

It is not necessary to make the Top part active before you use the  **Part**. Every new part you create is automatically positioned immediately below the Top part.

To create a new empty part immediately below Top:

1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **New**.
2. Click  **Part**
3. Type a name for the new part in the user input line and press ENTER.
Alternatively, allow Creo Elements/Direct Drafting to assign a default name.
If you want Creo Elements/Direct Drafting to assign a default name to the new part, click  **OK**. Creo Elements/Direct Drafting generates a default name based on incremental numbers (.1, .2, .3 and so forth).

Creo Elements/Direct Drafting creates the new empty part below the Top part and sets it as the active part. You can now begin adding drawing lines, text, and so on to the part.

 **Note**

Place at least one element (or subpart) in the new part otherwise it is deleted when you change the active part status.

Initiating a Subpart

You can create a new empty part under the active part and set the new part as the active part so you are ready to draw in it. This means that a subpart can be created below an existing part, or below an existing subpart.

Before clicking  **Subpart**, ensure you have the correct part active as the new part will belong to it and be one level below.

To create a new empty subpart:

1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **New**.
2. Click  **Subpart**.
3. Type a name for the new subpart in the user input line and press ENTER.

Creo Elements/Direct Drafting creates the new empty part and sets it as the active part. You can now begin adding drawing lines, text, and so on, to the part.

Place at least one element in the new part, otherwise it is deleted when you change the active part status.

Creating Parts

This section describes how to make a new part from existing elements in the active part. The active part status is unaffected after the command has been executed.

Creating a Subpart

You can create a new part by transferring elements from the active part into the newly created one. The new part is placed below the active part.

Before selecting  **Subpart**, ensure you have the correct part active as the new part will belong to it and be one level below.

To create a subpart:

1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **New**.
2. Click  **Subpart**.
3. Type a name for the new part in the user input line and press ENTER.
4. Select the elements in the active part you wish to transfer to the new part.

 **Note**

If you incorrectly select an element for transfer, click  **Undo** to cancel the transfer.

5. When you have selected all of the elements you want to include in the subpart, click  **OK** (or any other button except  **Cancel**) to complete the command.

Creo Elements/Direct Drafting transfers all the selected elements to the new part.

Creating Detail Drawings

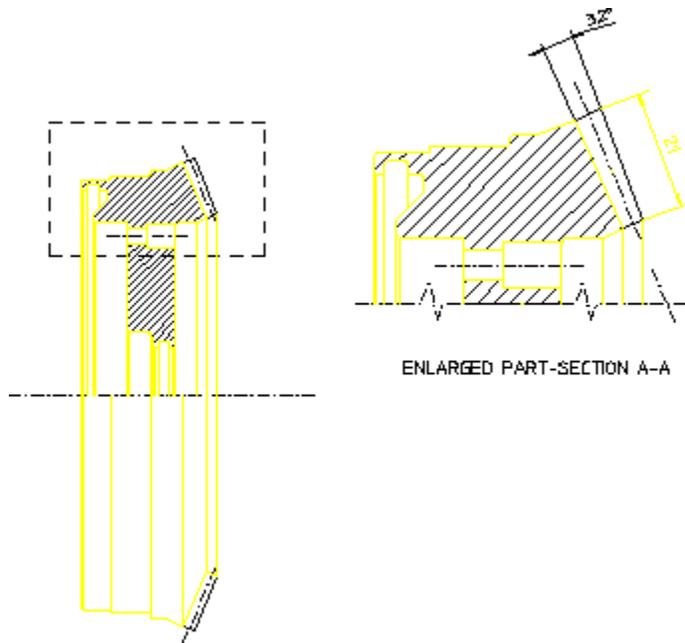
You can create detail drawings in your drawing. Each detail is a magnified copy of an area of the active part.

 **Note**

Dimensions are not affected by the magnifying factor; thus the dimensions on your detail will agree with the part from which it was taken.

Figure 176 on page 444 shows how to create a detail drawing within your main drawing:

Figure 176. Creating a Detail Part



Finish drawing the part to be detailed and make it the active part.

This technique of finishing the part before creating a detail differs from traditional drafting methods. However, using the buttons on the **View** tab, you can just as easily draw intricate features on the main drawing as on the detail drawing.

Creo Elements/Direct Drafting allows you to create a magnified copy for

dimensioning purposes using  **Detail**.

To produce a detail drawing:

1. Click **Structure** and then in the **Parts** group, click the arrow next to  **New**.
2. Click  **Detail**.
3. Type the required magnifying factor for the detail in the user input line and press ENTER.
4. Enclose the area of the active part from which you wish to form the detail in a box.

 **Note**

The edges of the box determine where elements are cut, so specify the corners carefully (in the detail drawing, the dashed lines indicate where the box was placed).

5. Specify a reference point for the movement of detail part and then specify the desired end point.

Creo Elements/Direct Drafting creates the detail part.

 **Note**

- The detail part is automatically placed in level 1 of the parts structure (that is, below the Top part).
 - The detail part is assigned the next default name generated by Creo Elements/Direct Drafting in the sequence .1, .2, .3, and so on. (Note that a postfix [shared #1] [detail] is displayed in the parts tree; this is for information only and is not part of the name).
 - The detail part will be a unique shared part (refer to [Sharing Parts on page 456](#)). However, it is not shared with the part from which it was detailed. Thus, if you modify the original part, its detail will not be changed automatically.
 - To dimension a detail part, you must make it the active part.
-

Gathering Elements in Other Parts

During the drawing process, it may prove useful to transfer the ownership of existing elements in a part to the active part.

To transfer elements:

1. Click **Structure** and then, in the **Parts** group, click  **Gather**.
2. Click  **Element**.
3. Click one of the following:
 - Click  **Delete Old Keep Scale** if you want to move the element to the active part. It is drawn to the scale in the active part.

-
- Click  **Copy Keep Scale** if you want to copy the element to the active part. It is drawn to the scale in the active part.

 **Note**

- If the element to be transferred is the master element of a group, the group is ungrouped before gathering.
 - If the element to be transferred is a normal element of a group, the element is removed from the group before gathering.
-

Delete Old

The old element is deleted and the transferred elements only exist in the active part.

 **Note**

- Identify the correct elements as mistakes can only be rectified by changing the active part and gathering the element back.
 - Click  **Spotlight** in the status bar to see the changes made in your drawing.
-

To transfer elements from one part to the active part:

1. Click **Structure** and then, in the **Parts** group, click  **Gather**.
2. Click  **Element**.
3. Click  **Delete Old**.
4. Select the elements (in non-active parts) that you wish to transfer to the active part.

As you select each element, it gets deleted from its original part and appears in the active part.

Copy

The elements indicated are duplicated in the active part.

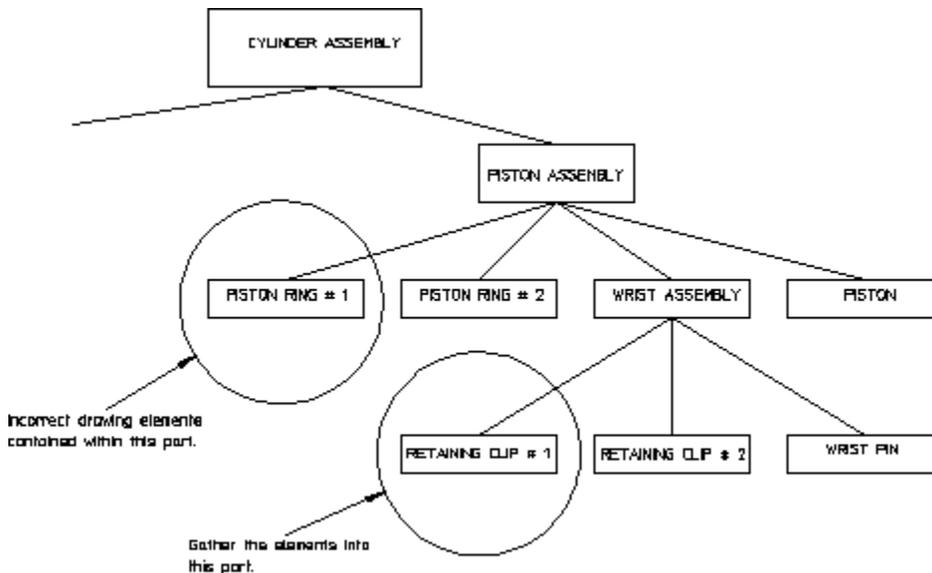
To copy elements from any part to the active part:

1. In the status bar, click  **Spotlight**.
2. Click **Structure** and then, in the **Parts** group, click  **Gather**.
3. Click  **Element**.
4. Click  **Copy**.
5. Select an element (in any non-active part) that you wish to copy to the active part.
6. Specify a reference point for the copy operation.
7. Specify the desired location for the reference point.

Example

Refer to [Figure 177 on page 447](#). You have incorrectly added drawing elements to PISTON RING # 1. Use  **Element** buttons in  **Gather** in the **Parts** group on the **Structure** tab to transfer these elements to RETAINING CLIP # 1.

Figure 177. Including Elements



To transfer elements from PISTON RING # 1 to RETAINING CLIP # 1:

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.
2. Click  **Part**.
3. Click RETAINING CLIP # 1.

Creo Elements/Direct Drafting makes RETAINING CLIP # 1 the active part.

4. Click **Structure** and then, in the **Parts** group, click  **Gather**.
5. Click  **Element**.
6. Click  **Delete Old**.
7. Select elements in PISTON RING # 1 you wish to transfer to the active part. As you select each element, it is deleted from its original part and appears in the active part.

Including Dimensions in Other Parts

Creo Elements/Direct Drafting allows you to transfer the ownership of existing dimensions using  **Dimension** buttons of  **Gather** in the **Parts** group on the **Structure** tab.

Move

To move the dimension to the active part when the reference geometry of the selected dimension is in the active part or below the active part:

1. Click **Structure** and then, in the **Parts** group, click  **Gather**.
2. Click  **Dimension**.
3. Click  **Move**.
4. Select the dimension you want to move to the active part.

Move All Below

To move all dimensions in parts hierarchically below the active part to the active part:

1. Click **Structure** and then, in the **Parts** group, click  **Gather**.
2. Click  **Dimension**.
3. Click  **Move All Below**.
4. Select the dimension you want to move to the active part.

Move All Above

To move all dimensions in parts hierarchically above the active part with reference geometry in the active part or below the active part to the active part:

1. Click **Structure** and then, in the **Parts** group, click  **Gather**.
2. Click  **Dimension**.
3. Click  **Move All Above**.
4. Select the dimension you want to move to the active part.

Including Parts in Other Parts

This section describes how to transfer the ownership of existing parts to the active part. The gathered part belongs to the active part and it is placed one level below the active part in the parts structure.

Note

- If the part to be transferred is the master element of a group, the group is ungrouped before gathering.
 - If the part to be transferred is a normal element of a group, the part is removed from the group before gathering.
-

Move

To transfer the ownership of a part to the active part:

1. Click **Structure** and then, in the **Parts** group, click  **Gather**.
2. Click  **Part**.
3. Click  **Move**.
4. Select the part to be gathered or type the name of the part in the user input line and press ENTER. You cannot gather a part from above the level of the active part.

Creo Elements/Direct Drafting transfers the part to the ownership of the active part and deletes the original.

Copy

To copy a part to the active part:

1. Click **Structure** and then, in the **Parts** group, click  **Gather**.
2. Click  **Part**.
3. Click  **Copy**.
4. Select the part to be included or type the name of the part in the user input line and press ENTER.
5. Specify a reference point for the copy operation.
6. Specify the location for the reference point.

Creo Elements/Direct Drafting copies the part such that the new part is owned by the active part.

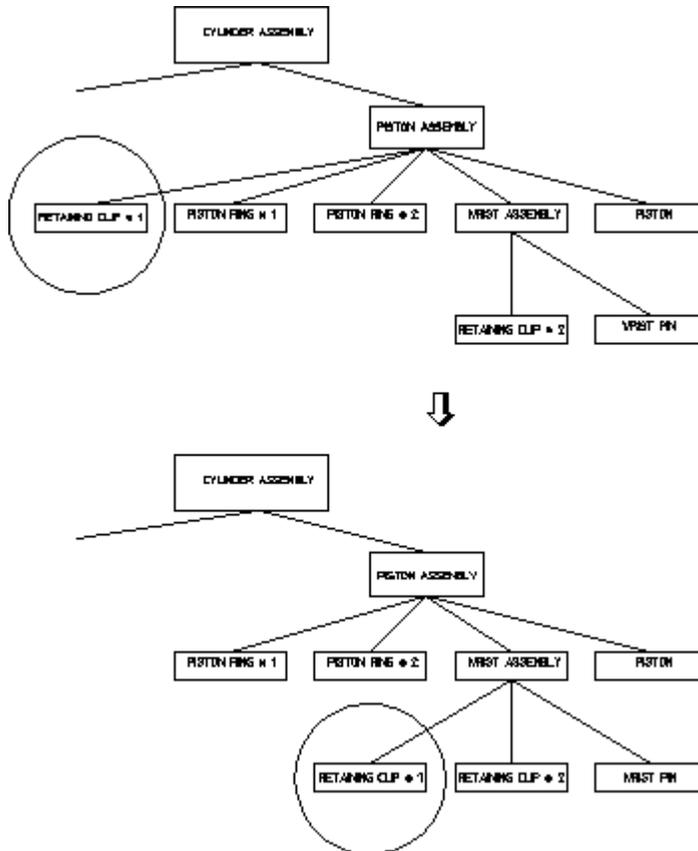
Example

Refer to [Figure 178 on page 451](#). You have incorrectly created the part

RETAINING CLIP # 1 as a subpart of PISTON ASSEMBLY. Use the  **Part**

buttons of  **Gather** in the **Parts** group on the **Structure** tab to transfer RETAINING CLIP # 1 to its correct position as a subpart of WRIST ASSEMBLY.

Figure 178. Including a Part



To transfer RETAINING CLIP # 1 to its correct position as a subpart of WRIST ASSEMBLY:

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.
2. Click  **Part**.
3. Click WRIST ASSEMBLY.
Creo Elements/Direct Drafting makes WRIST ASSEMBLY the active part.
4. Click **Structure** and then, in the **Parts** group, click  **Gather**.
5. Click  **Part** and then click  **Move**.
6. Click RETAINING CLIP # 1.

Creo Elements/Direct Drafting transfers the part so that it becomes a subpart of WRIST ASSEMBLY.

Showing Parts

This section describes how to examine a selected groups of parts. Those you do not want to see are replaced by representative boxes. This can be useful where parts overlap.

Part

You can either restore a part presently represented by a box, or look at certain levels in the parts structure.

If you want to restore a boxed part:

1. Click **Structure** and then, in the **Parts** group, click  **Show**.
2. Click  **Part**.
3. Click the part.

Creo Elements/Direct Drafting redraws it normally.

However, if you wish to look down to a certain part level and no further:

1. Click **Structure** and then, in the **Parts** group, click  **Show**.
2. Click  **Part**.
3. Type the level number in the user input line and press ENTER.
Creo Elements/Direct Drafting redraws the chosen viewport with all parts in the entered level and below, shown as boxes.
4. For example, if you type 3, the chosen viewport is redrawn to show all parts in levels 1 and 2 with those in lower levels indicated by boxes. If a boxed part belongs directly to the active part, this is indicated by a yellow box.

Boxed

You can change a part and all its subparts to a box representation in the chosen viewport. The box encloses all elements of the part.

1. Click **Structure** and then, in the **Parts** group, click  **Show**.
2. Click  **Boxed**.
3. Select the part to be boxed or type its name in the user input line and press ENTER.

Creo Elements/Direct Drafting redraws the chosen viewport with the desired part shown as a box.

All

You can restore all parts in the chosen viewport to their normal appearance:

1. Click **Structure** and then, in the **Parts** group, click  **Show**.
2. Click  **All**.

Creo Elements/Direct Drafting redraws the chosen viewport with all parts visible.

Hiding Parts

You can hide specific parts or all parts in your drawing.

Part

To hide a specific part in your drawing:

1. Click **Structure** and then, in the **Parts** group, click  **Hide**.
2. Click  **Part**.
3. Click the part.

All

To hide all parts in your drawing:

1. Click **Structure** and then, in the **Parts** group, click  **Hide**.
2. Click  **All**.

You can use the **Show** buttons to unhide any or all parts.

Smashing Subparts

When you smash a subpart, the elements that comprise the identified part will be merged into the active part and the subpart will no longer exist.

To merge a subpart into the active part:

1. Ensure that the correct part is active.

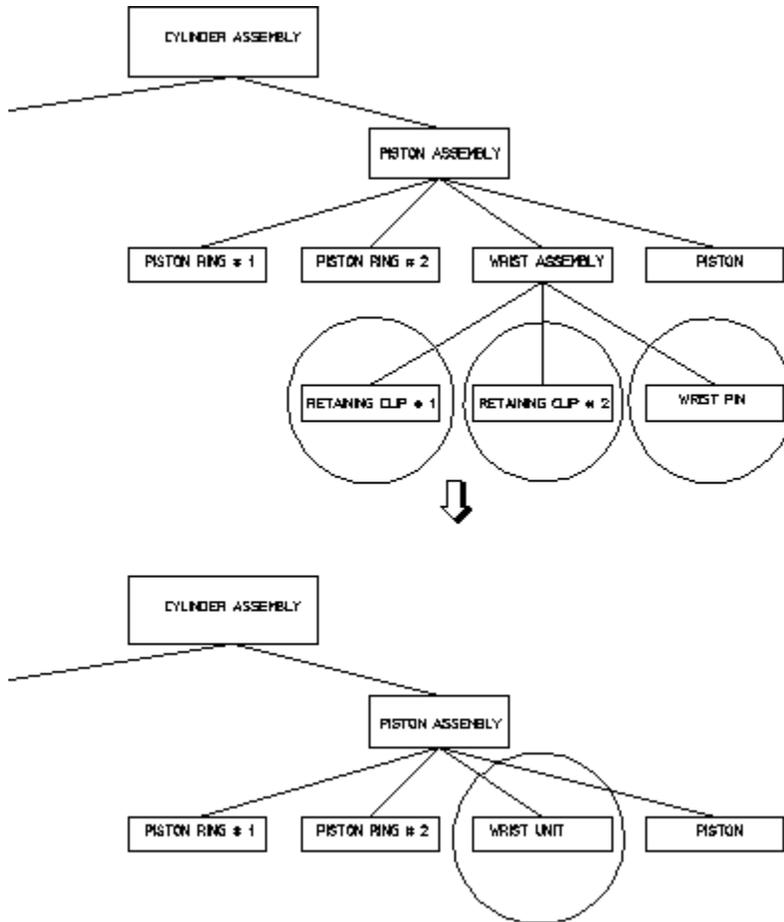
-
2. Click **Structure** and then, in the **Parts** group, click  **Smash**.
 3. Click any of the following:
 -  **Subpart**
The subpart is merged with the active part.
 -  **Subpart Keep Scale**
The subpart is merged with the active part, and it is drawn to the scale of the top part (scaling is maintained – drawing A).
 -  **Subpart Keep Display**
The subpart is merged with the active part, and it is brought into the top part as it is displayed on the screen (scaling may be lost) – drawing B.
 4. Select the part to be smashed or specify the part name. The part to be smashed must belong to the active part.

Creo Elements/Direct Drafting transfers the elements of the identified part to the active part and deletes the subpart.

Example

Refer to [Figure 179 on page 455](#). You want to make a single part drawing (WRIST UNIT) from the assembly drawing WRIST ASSEMBLY.

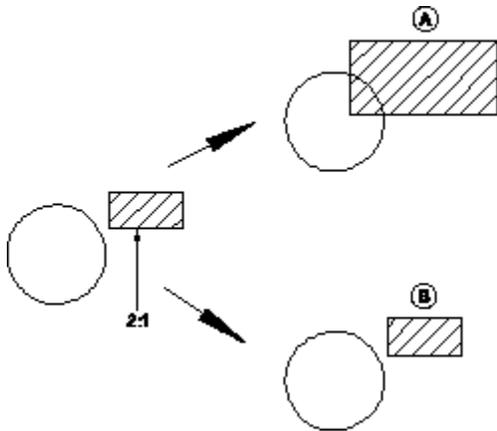
Figure 179. Smashing Parts



To create WRIST UNIT from WRIST ASSEMBLY:

1. Click **Structure** and then, in the **Parts** group, click  **Edit**.
2. Click  **Part**.
3. Click WRIST ASSEMBLY.
Creo Elements/Direct Drafting makes WRIST ASSEMBLY the active part.
4. Click **Structure** and then, in the **Parts** group, click  **Smash**.
5. Click RETAINING CLIP # 1, RETAINING CLIP # 2 and WRIST PIN.
Creo Elements/Direct Drafting transfers all drawing elements to the active part and then deletes the subparts.
6. In the **Part Browser**, right-click the active part and click  **Rename**.
7. Rename the part to WRIST UNIT.

Figure 180. Effects of Smashing Scaled Parts



Note

- If the part to be selected is the master element of a group, the group is ungrouped before smashing.
- If the part to be selected is a normal element of a group, the part is removed from the group before smashing.

Sharing Parts

This section shows how to make a shared part. Shared parts are useful if you require a particular part to appear several times in a drawing.

Examples of parts suitable for sharing are nuts, bolts, rivets, and other standard parts.

The geometry of a shared part always remains the same as the original part. However, the location of each shared part can be different and the overall size of each shared part can be altered by using the  **Scale** buttons.

You can also determine the plane in which the shared part lies by using  **Isometric** in the **General** group on the **Modify** tab. If you place a shared part in a different plane, make it active and then enable the **Ruler** or **Grid**, so that they will also be displayed in the same plane.

The advantages of shared parts:

- Although you have several copies of the shared part displayed on your drawing, in memory you have only one. This saves you memory and allows you to carry out certain commands faster.

- If you change any geometry of a shared part, the changes will automatically appear in all other occurrences of that shared part in their respective planes. For example, if you add a horizontal line to a shared part, it will be displayed on the respective horizontal axis of each occurrence of that shared part.

To share a part:

1. Click **Structure** and then, in the **Parts** group, click  **Share**.
2. Select the part to be shared.

The part must be non-active and may not be at a level above the active part or at the same level. In the **Share** column of the **Part Browser**, a shared part is identified by one or multiple hash characters followed by the appropriate number (such as: #3).

Copying Shared Parts

To create copies of a shared part:

1. Click **Modify** and then, in the **General** group, click the arrow next to  **Move**.
2. Click  **Multiple**.
3. Select the shared part to be copied.
4. Specify a reference point for the move.
5. Specify the destination point for the move.

The following naming conventions apply to copies of shared parts:

- When you copy a shared part as described above, note that each copy will reflect the name of the original shared part. Each of these copies automatically receives a unique identifier. For example, when you copy a shared part named `rim3`, its copies will also be named `rim3`; however, each of these copies will have its own unique identifier. (`~2`, `~3`, etc).

The following example shows the parts structure of an assembly containing a shared part named `RIM` and a copy of this shared part.

	Uniq	Share	Scale	Symbol
WHEEL ASSEMBLY	~1		1:1	
RIM	~2	#3	1:1	
RIM	~3	#3	1:1	

- When you copy a shared part which owns another shared part (i.e. a shared subpart of a shared part), the copies of these shared subparts retain the same common name and identifier. However, each of the "owning" shared parts has its own unique identifier.

The following example shows the parts structure of an assembly containing a shared part named RIM which in turn owns a shared part named LINER.

	Unique	Share	Scale	Symbol
WHEEL ASSEMBLY	~1		1:1	
RIM	~2	#3	1:1	
LINER	~3	##4	1:1	

When you copy the RIM, the resulting parts structure will look like this:

	Unique	Share	Scale	Symbol
WHEEL ASSEMBLY	~1		1:1	
RIM	~4	#3	1:1	
LINER	~3	##4	1:1	
RIM	~2	#3	1:1	
LINER	~3	##4	1:1	

Refer to [Unique Partnames on page 468](#) for further information.

Note

- Any changes made to a shared part automatically appear in all other occurrences of that shared part.
- You cannot share a part above the active part or at the same level. Therefore you cannot share the Top Part.
- All subparts of a shared part have their status set to shared.
- When you rename a shared part, all occurrences of that shared part are given the new name.

Unsharing Parts

This section describes how to change a shared part back into a normal part.

This can be useful if you want to alter one occurrence of your shared part in isolation, as otherwise all occurrences are altered simultaneously.

Note

You cannot unshare a part above the active part or at the same level.

To unshare a part:

1. Click **Structure** and then, in the **Parts** group, click  **Unshare**.
2. Select the part to be unshared.

Creo Elements/Direct Drafting alters the parts structure so that the identified part is a normal part positioned alongside the shared parts and with the same parent. All other occurrences of the shared part stay shared.

The active status does not affect  **Unshare** except that the part you wish to unshare cannot be active.

Part Browser

To display the **Part Browser**:

- Click **Structure** and then, in the **Parts** group, click  **Part Browser**.
Alternatively, press F12 to display the **Part Browser**.

To perform an action on a given part:

- Right-click the part in the **Part Browser** and click the required button.

Note

The **Part Browser** can be docked anywhere in the Creo Elements/Direct Drafting window.

What is Shown in the Part Browser?

The **Part Browser** provides a graphical overview of the current parts structure in your Creo Elements/Direct Drafting drawing.

- Starting at the top part, this tree-like structure includes a separate branch for each part. Subparts of a part are shown as sub-branches of that part.
- Click the + (plus) sign icon to expand the tree.
- Click the - (minus) sign icon to collapse the tree.
- Click the check box to the left of any part icon to view the associated part (and any parts hierarchically below this part).
- Clear the check box to the left of any part icon to hide the associated part (and any parts hierarchically below this part).

The individual columns of the **Part Browser** provide the following information:

Column	Description
Name	Specifies the name of the part. The following icons are used to indicate the nature of the part:
	 The currently active part. Note that the currently active part is represented by a red icon.
	 Normal part or assembly. Any non-active parts and/or parts hierarchically below the active part are indicated by a green icon.
	 Shared part.
	 Detail.
 The associated part is hidden. This is because you have applied the View command to another part.	
Unique	Specifies a unique sequential number associated with the part.
Share	A "hash" (#) character in this column indicates that the part is shared.
Scale	Indicates the drawing scale of the part.
Symbol	A small "x" indicates that the part is symbolic.

The Part Browser Tree Context Menu

To display the **Part Browser** context menu, right-click any part icon in the **Part Browser**. The context menu provides the following buttons (some of which are also available in the **Parts** group on the **Structure** tab):

View	Shows only the selected part. Creo Elements/Direct Drafting fits this part into the current viewport. All other parts are hidden, and their icons are marked by a yellow X (see list of Part Browser icons above).
Rename	Enables you to change the name of the selected part.
Smash	Smashes a subpart. See Smashing Subparts on page 453 for further information.
Share	Creates a shared part. See Sharing Parts on page 456 for further information.
Symbol	Changes the part to a symbol. See Changing Parts to Symbols on page 464 for further information.
Show	In addition to the options explained in Showing Parts on page 452 , this command provides the ShowFlat option which shows the selected part but not its subparts (if any).
Hide	In addition to the options explained in Hiding Parts on page 453 , this command provides the HideFlat option which hides the selected part but not its subparts (if any).
Options	The Icons option enables you to display the Part Browser with or without icons. The State Icons option enables you to show or hide the (checkbox-like) state icons that indicate whether or not a part is visible.
Save	You can click Save to save an ASCII representation of the current parts tree structure in a Creo Elements/Direct Drafting list file.
Delete	Deletes the selected part.

Changing the Ownership of Parts

The **Part Browser** provides drag-and-drop functionality to facilitate changes in part ownership. For example, if you want a subpart to be owned by another part, simply drag the subpart away from its present owner part and drag it to the new owner part.

Renaming a Part in the Part Browser

To rename a part:

1. In the **Part Browser**, right-click the part you want to rename and click  **Rename**.
2. Replace the current part name by the new name.

Note

When you rename a shared part, all occurrences of that shared part are given the new name.

Viewing Parts

This section describes how to view individual parts of your drawing. You can only view drawings in the current viewport.

Use the  **View** buttons in the **Parts** group on the **Structure** tab in the following cases:

- During the design stage if parts overlap one another in a confusing way.
- To quickly inspect a finished part and check that it is complete.
- To reduce redraw time by viewing only the part or parts you are currently working on.

Top

You can view your complete drawing in the current viewport.

This is useful if you have been viewing a lower level and want to quickly restore the complete drawing to the viewport

To view the complete drawing in the current viewport:

1. Click **Structure** and then, in the **Parts** group, click  **View**.
2. Click  **Top**.

The chosen viewport is redrawn to show all the parts of your drawing. The name of the top part is displayed in the top-left corner of the viewport.

Part

To look at an individual part:

1. Click **Structure** and then, in the **Parts** group, click  **View**.
2. Click  **Part**.

-
3. Type the name of the part to be viewed, or click an element of the part in any viewport.

Creo Elements/Direct Drafting redraws the chosen viewport containing only the desired part and any subparts belonging to it. The name of the part being viewed is displayed in the top left corner of the viewport.

Parent

You can view the parent of the presently viewed part.  **Parent** allows you to view the next level up in the parts hierarchy:

1. Click **Structure** and then, in the **Parts** group, click  **View**.
2. Click  **Parent**.

The current viewport is redrawn displaying the parent of the active part and all parts belonging to the active part.

Creating a Parts List

Creo Elements/Direct Drafting allows you to create a list of parts in your drawing. The parts list indicates the relationship between the active part and all other parts. You can view the list of parts that are under an active part (parent part). You can also create a list of all parts in your drawing. Creo Elements/Direct Drafting can display the parts list in a standard text editor or print the list on a local printer.

Counting Parts

You can create a list of all the parts owned by the active part (parent part) and the parts under the active part.

To list the parts:

1. In the **Part Browser**, select the part.
2. Click **Structure** and then, in the **Parts** group, click **More**.
3. Under **Part List**, click **Count**.

Creo Elements/Direct Drafting displays the list of parts in a standard text editor.

Creating a Parts Tree

You can create a list of all parts in your drawing. The active part is indicated by a * next to it in the list. Part names in the list are indented to indicate the hierarchical relationship between each part. The relationship is indicated by indentation of the part name.

To create a full parts list:

1. Click **Structure** and then, in the **Parts** group, click **More**.
2. Under **Part List**, click **Tree**.

Creo Elements/Direct Drafting displays the list of parts in a standard text editor.

Spotlighting Parts

Spotlighting allows you to distinguish between the active part and all other parts. The button works as an on or off switch. We recommend you enable **Spotlight** (by clicking it) when you are creating and modifying parts.

1. To enable **Spotlight**, click  **Spotlight** in the status bar.

Only the active part has its normal appearance. All non-active parts are shown in magenta dotted lines.

2. To disable **Spotlight**, click  **Spotlight** in the status bar again.

All parts appear in their normal linetype and color. This makes it difficult to distinguish between the active part and all other parts.

Changing Parts to Symbols

This section describes how to change existing parts into symbols.

The only difference between a symbol and a normal part is that a symbol's size relative to the plotter paper does not change if you alter the drawing scale (refer to [Setting the Drawing Scale on page 67](#)).

An example of a symbol part could be a border to fit a particular paper size or any standard symbol. Once you've made a symbol part, it is best to store it so that it is ready to be loaded whenever you require it.

To make a symbol part:

1. Create the required shape as a part with the drawing scale set to 1.
2. When the part is finished, click **Structure** and then, in the **Parts** group, click the **Symbols** check box.

3. Select the part or type the name of the part in the user input line and press ENTER.

Creo Elements/Direct Drafting now defines the part as a symbol part. Check this by viewing the **Symbol** column of the **Part Browser**. Note a small "x" in the **Symbol** column.

You can also change a symbol part back into a normal part

To change a symbol to normal part:

1. Click **Structure** and then, in the **Parts** group, clear the **Symbols** check box.
2. Select the symbol part or type the name of the part in the user input line and press ENTER.

Creo Elements/Direct Drafting changes the symbol part back to a normal part.

Setting the Part Reference Point

This section shows how to set a reference point in the active part.

The reference point is used:

- To locate other parts when loading from files.
- For symbols.
- For some  **Gather** buttons.

The Creo Elements/Direct Drafting default reference point is the lower left corner of an imaginary box placed around the elements of the part.

To set a reference point:

1. Click **Structure** and then, in the **Parts** group, click  **Set Ref Point**.

The present reference point for the active part is shown by a line joined to the cursor.

2. Specify the new reference point.

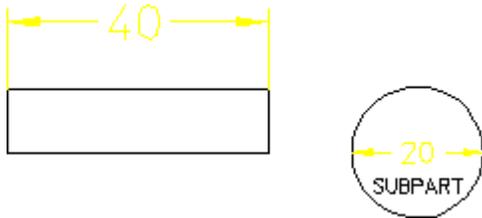
Multiple Drawing Scales

Creo Elements/Direct Drafting supports multiple drawing scales on one drawing. This lets you place small and large parts on the same drawing for manufacturing and documentation purposes.

The scale associated with a part can be seen in the parts tree.

To change the drawing scale:

1. Create a top part and a subpart.



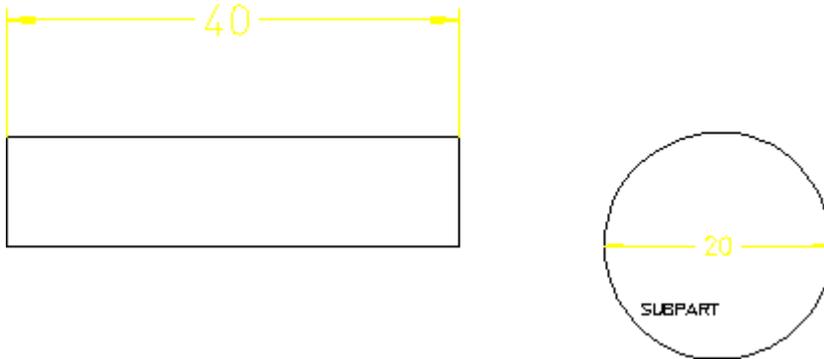
The parts tree in the **Part Browser** should look similar to the following:

```

Top          ~1    1:1
    SUBPART  ~2    1:1
  
```

2. Click **Structure** and then, in the **Parts** group, click  **Scale**.
3. Click  **Part & Subpart**.
4. Select the top part (both parts are highlighted).
5. Type 2 in the user input line and press ENTER.

Creo Elements/Direct Drafting scales both the top part and subpart by the scaling factor 2.



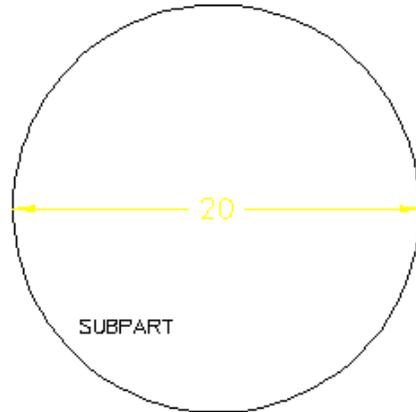
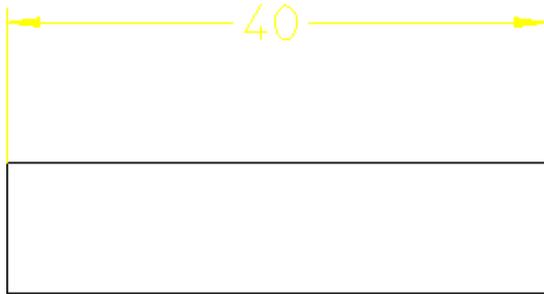
Now look at the parts tree again to see how the scaling factors change:

```

Top          ~1    2:1
    SUBPART  ~2    2:1
  
```

1. Click **Structure** and then, in the **Parts** group, click  **Scale**.
2. Click  **Part Only**.
3. Select the subpart (it does not need to be active).
4. Type 3 in the user input line and press ENTER.

Creo Elements/Direct Drafting scales the subpart only.



Now look at the parts tree again to see how the scaling factors change:

Top ~1 2:1

SUBPART ~2 3:1

Note that the subpart is scaled according to its original value. In other words, it is scaled by a factor 3 — and not 2 and then 3.

Text, dimensioning text, and symbols are not changed.

Scaling Parts about a Reference Point

To scale parts and subparts about reference point, origin, or center:

1. Click **Structure** and then, in the **Parts** group, click  **Scale**.
2. Click one of the following:
 -  **By Reference Point** to scale parts and subparts about the reference point.
 -  **By Origin** to scale parts and subparts about the origin.
 -  **By Center** to scale parts and subparts about the center.

Renaming Parts

To change the name of the active part:

1. Click **Structure** and then, in the **Parts** group, click **More**.
2. Under **Modify**, click **Rename**.

Creo Elements/Direct Drafting displays the name of the active part and prompts you for a new name.

3. Type a new name enclosed by quotes (single or double) in the user input line and press ENTER.

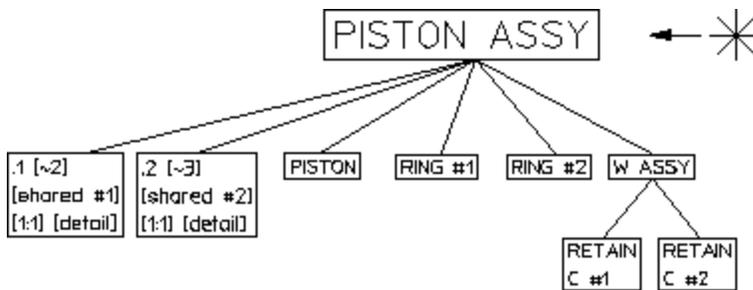
Creo Elements/Direct Drafting renames the active part to the new name.

You can use the **Part Browser** to check the new name of the active part.

Unique Partnames

Creo Elements/Direct Drafting automatically generates names for parts when you create them. These names are shown in the parts tree and are represented by numbers: ~1, ~2, and so on.

This feature is useful when handling parts by macros. For example, a typical parts tree may look like this:



Unique parts naming has the following functionality:

- Each part is specified by a number.
- Shared parts within shared parts are identified with the same unique partname because they exist once only in the data structure.

Shared parts within shared parts are marked with [shared in shared #n]

- Parts can be made active by entering EDIT_PART followed by the name of the part. Two examples are provided below.

```
EDIT_PART 'bracket'
EDIT_PART '~2'
```

If you enter EDIT_PART and the parts structure has more than one part with the same partname, Creo Elements/Direct Drafting prompts you to specify the part number.

With INQ 900 and INQ 901, you get a string list of unique partnames (~2, ~3).

- INQ_ENV 8 INQ 5 for inquiry of the offset. For loading several drawings, the maximum used unique partname can be inquired (useful when running macros).

-
- You can click  **Delete**,  **All** and  **Confirm** to reset the parts numbering.
 - Refreshing partnames can be done by typing the following command in the user input line:
`RESET_PART_NUMBER`

Further Information

For further technical information about parts, the parts structure and so on, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- `help parts`
- `help edit_part`
- `help init_part`
- `help init_subpart`
- `help create_subpart`
- `help create_detail`
- `help gather_dim`
- `help gather`
- `help smash_subpart`
- `help share_part`
- `help unshare_part`
- `help symbol_part`
- `help view`
- `help show_part`
- `help hide_part`
- `help change_part_ref_pt`
- `help scale`
- `help part_drw_scale`
- `help wui_part_browser`

Creo Elements/Direct Drafting displays further information about parts.

15

Generating Hidden Lines

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Introduction to Hidden Lines

The 2D hidden line module is used during the creation of an assembly drawing to automatically generate lines that indicate hidden drawing features. In an assembly drawing, these lines are normally shown dashed.

To activate the **Hidden** module:

1. Click **File** ► **Modules** ►  **Hidden**.

The **Hidden** tab appears.

To create an assembly drawing that includes hidden lines, you must:

- Load parts.
- Generate faces (without faces you cannot generate hidden lines).
- Assign each face a logical level in the z direction (a z-value).
- Use the z-value information to generate the appropriate hidden lines.

During the drawing process you can selectively view faces for clarity (ensure that each face is correctly positioned in relation to the others).

Use one of the following methods to redraw your assembly drawing with hidden lines:

- Click **Hidden** and then, in the **Modify** group, click the **Hidden Mode** check box.
This mode effectively redraws your assembly drawing, hiding any hidden lines and faces.
- Click **Hidden** and then, in the **Modify** group, clear the **Hidden Mode** check box.
This is the normal Creo Elements/Direct Drafting redraw mode.

On completion of your assembly drawing you can generate hidden lines in the dashed linetype (default).

Caution

It is recommended to store your assembly drawing (in `workfile.hid`) before generating hidden lines:

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Generate**.
2. Click  **with Backup**.

The  **Generate** buttons smash the part structure and create the hidden line geometry.

Two tutorial examples have been included:

- A simple example using a Frame Assembly will clarify the general procedure.
- A realistic example creates a component assembly drawing (CRANK_ASSEMBLY).

What are Faces?

Faces can be thought of as freely definable, infinitely thin, shaped areas that form part of an assembly. Faces can partially obscure or completely cover other faces (depending upon their z-values).

To delete any covering face:

1. Click **Hidden** and then, in the **Modify** group, click  **Delete Face**.
2. Select the face.

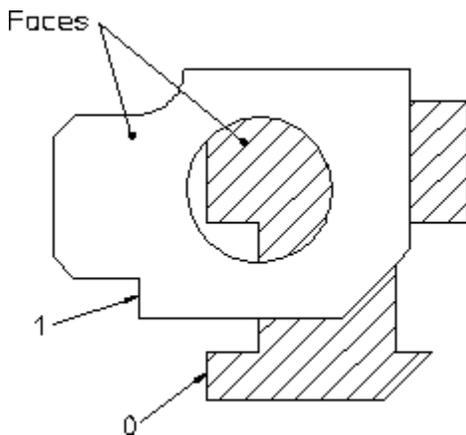
Note

The face to be deleted must be in the active part.

In simple terms, you could imagine faces as pieces of paper, laid one on top of another, thus obscuring elements with a lower z-value.

Figure 181. Faces — With Z-Values 0 and 1 on page 472 shows the faces of two components (one with z-value 0 and the other with z-value 1).

Figure 181. Faces — With Z-Values 0 and 1



 **Note**

A face does not necessarily need to have the same z-value as its boundary lines. In this case, the boundary lines of the face may be partially covered by the face itself (solid lines may appear dashed in some cases). The linetype of the boundary lines is not changed.

Faces are defined in one of the following ways:

- Automatic face search for an entire part (Creo Elements/Direct Drafting tries to find a maximum number of faces contained within a part. All faces are assigned the same z-value).
- Automatic face search for an entire part (with hole recognition).
- Automatic face search by clicking (similar to automatic hatching).
- Manual face specification (similar to manual hatching).

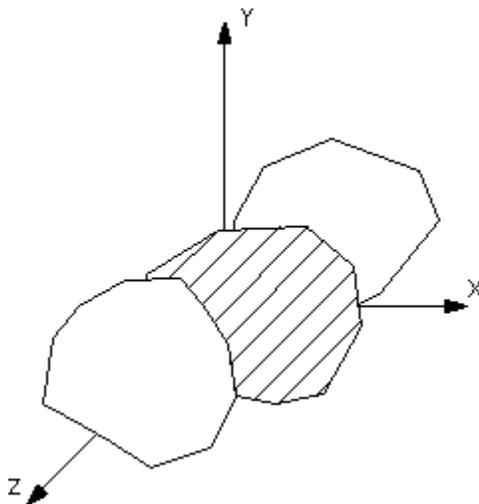
What are Z-Values?

A z-value is the numerical value assigned to an element to position it in the drawing at the appropriate logical level in the z-direction.

Newly created elements are assigned the current z-value. The current z-value is always displayed in the prompt line.

See [Figure 182. Visualizing Z-Values on page 473](#).

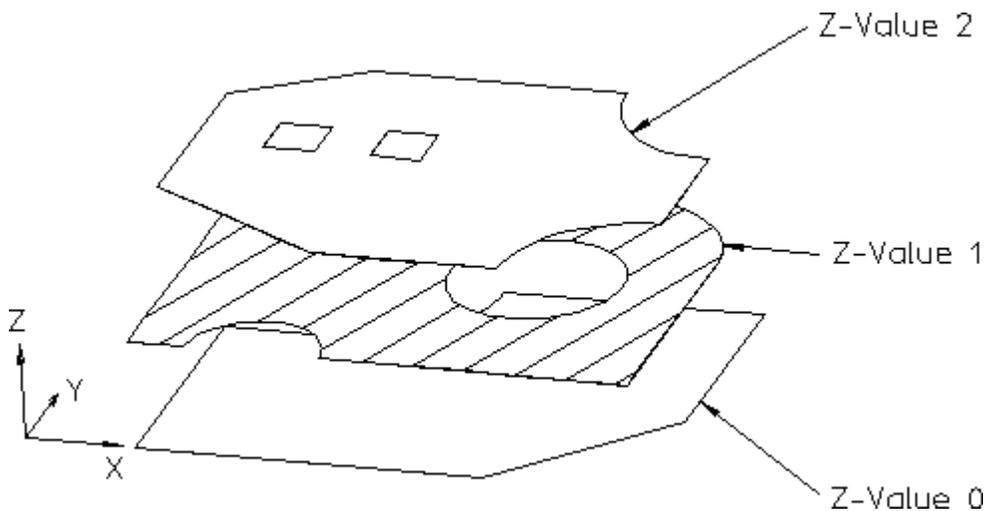
Figure 182. Visualizing Z-Values



For example, three faces could be given the z-values 0, 1, 2. The face with z-value 1 would be positioned above that with z-value 0. The face with z-value 2 would be positioned above those with z-values 1 and 0. Depending on where these faces were positioned with respect to each other, certain areas would be obscured.

Figure 183. Z-Values 0, 1 and 2 on page 474 shows 3 faces assigned the z-values 0, 1 and 2.

Figure 183. Z-Values 0, 1 and 2



Hatches are treated as faces. This means hatched areas can cover geometry and faces. Hatches that are partially covered are displayed partially obscured.

Construction geometry, texts, leader lines and dimensions are always visible.

Parts cannot be given a z-value, with the exception of shared parts (see [Z-Values and Shared Parts on page 475](#)).

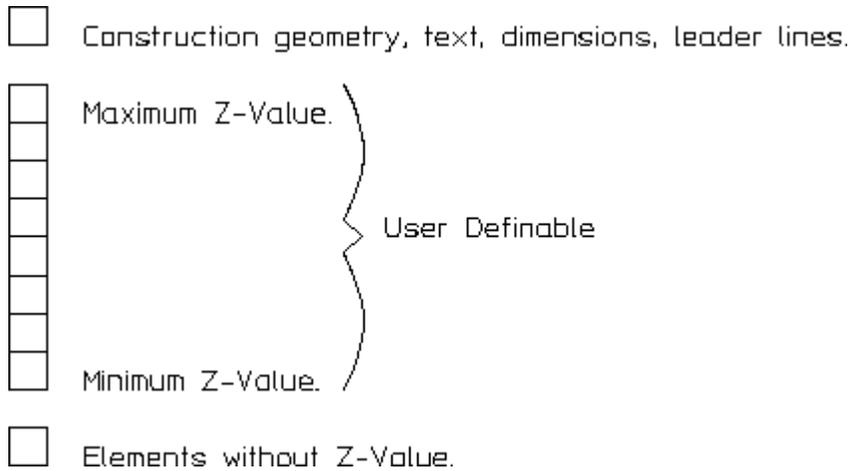
You can specify z-values in three ways:

- Automatically – if you have clicked **File** ► **Modules** ► **Hidden** and thus loaded the hidden line module. Future elements you create are assigned the current z-value.
- By defining "element-to-element" relations ( **Above**,  **Below**,  **Same**,  **Between**,  **None**).
- By entering a z-value (real number) for the specified elements.

Elements and geometry without a z-value are automatically positioned below the element(s) with the lowest z-value. Construction geometry, text, dimensions and leader lines are always at the top.

Figure 184. Z-Value Structure within a Drawing on page 475 shows how the z-value structure relates to your drawing:

Figure 184. Z-Value Structure within a Drawing



Z-Values and Shared Parts

You can assign z-values to shared parts to enable them to be placed on different logical levels.

The z-values of elements within a shared part are calculated as the sum of their own z-value and the z-values of all their shared parent parts.

Hidden Line Modes

The two hidden line modes are:

- View Mode – produces a hidden line display, does not compute hidden lines, does not change geometry.
- Hidden Line Generation Mode – used to finish the assembly drawing before plotting, changes geometry.

View Mode

This mode produces a hidden line display of the assembly on the screen without computing the hidden lines. It does not change the geometry or the parts structure. Therefore you can assign z-values to elements during construction. Changes in z-value are possible at any time and are displayed immediately.

Generation Mode

Hidden Line Generation Mode is used to finish an assembly drawing before plotting.

In contrast to the view mode, geometry is modified:

- Elements are split.
- Shared parts will be unshared.
- The parts structure is completely smashed.

You can enable or disable the hidden lines by clicking or clearing the **Hidden Mode** check box in the **Modify** group on the **Hidden** tab, respectively.

Hidden Line – Tutorial Example 1

This tutorial shows how to make a simple assembly drawing (Frame_Assy) and use it to create:

- An assembly view with the hidden lines to be generated as yet invisible (View Mode).
- An assembly view showing dashed hidden lines (Generation Mode).

Creating Frame Assembly

First create the assembly drawing Frame Assembly by creating two parts, Frame 1 and Frame 2.

Initial Steps

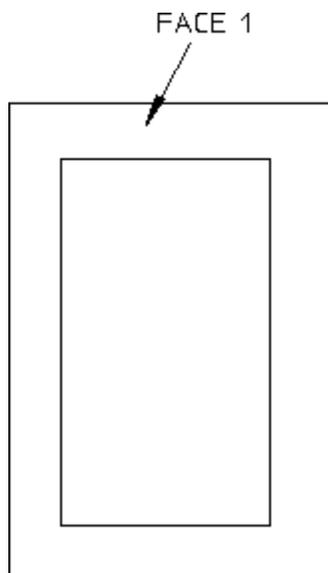
1. Click **Drafting** and then in the **Utilities** group, click  **Delete**.
2. In the **Utilities** group, click  **Select**.
3. Click  **All**.
4. Click  **Confirm**.
5. Click **Structure** and then, in the **Parts** group, click  **Browser**.
The **Part Browser** opens.
6. In the **Part Browser**, right-click  **Top** and click  **Rename**.
7. Rename  **Top** to Frame Assembly.

Drawing Frame 1

1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **New**.
2. Click  **Part**.

3. Type the part name `Frame 1` in the user input line and press ENTER.
4. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **Inquire Z Value**.
5. Click  **Current Z Value**.
6. Type `1` in the user input line and press ENTER.
7. Click **Drafting** and then, in the **Draw** group, click  **Rectangle**.
8. Draw two rectangles as shown in [Figure 185. Frame 1 on page 477](#).

Figure 185. Frame 1



Generating Face 1

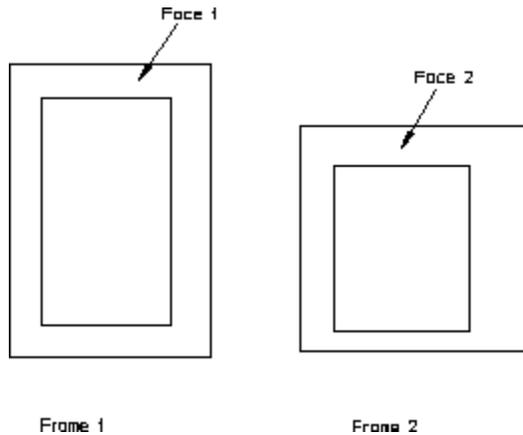
1. Click **Hidden** and then, in the **Create Face** group, click  **Single Automatic**.
2. Click inside `Face 1` (see [Figure 185. Frame 1 on page 477](#)).
Creo Elements/Direct Drafting creates `Face 1`.
3. Click  **OK**.

Drawing Frame 2

1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **New**.
2. Click  **Part**.

3. Type **Frame 2** as the part name in the user input line and press ENTER.
4. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **Inquire Z Value**.
5. Click  **Current Z Value**.
6. Type **2** in the user input line and press ENTER.
7. Click **Drafting** and then, in the **Draw** group, click  **Rectangle**.
8. Draw two more rectangles as shown in [Figure 186. Frame 1 and Frame 2 on page 478](#).

Figure 186. Frame 1 and Frame 2



Generating Face 2

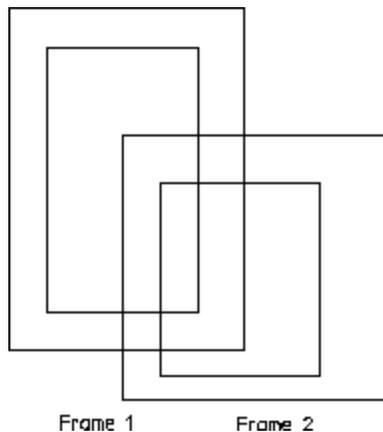
1. Click **Hidden** and then, in the **Create Face** group, click  **Single Automatic**.
2. Click inside **Face 2**.
3. Click  **OK**.

Positioning Frame 2 Above Frame 1

1. Click **Modify** and then, in the **General** group, ensure that the **Keep** check box is clear.
2. Select **Frame 2** by enclosing it in a box.
3. In the **General** group, click the arrow next to  **Move**.
4. Click  **2 Points**.

-
5. Specify a reference point for the move.
 6. Reposition Frame 2 above Frame 1 as shown in [Figure 187. Frame 2 Above Frame 1](#) on page 479.

Figure 187. Frame 2 Above Frame 1



Assembly – Hidden Lines Invisible

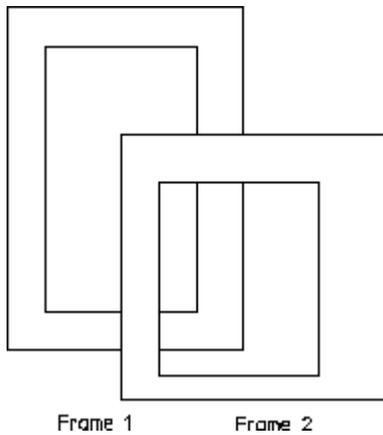
You can display your assembly drawing so that the lines that will be generated as hidden are removed from the view. This is useful as a check to see exactly which lines in your assembly will be generated as hidden lines.

1. Click **Hidden** and then, in the **Modify** group, click the **Hidden Mode** check box.
2. Click the current viewport.

To confirm hidden mode, Creo Elements/Direct Drafting displays **-H-** in the top-left corner of the viewport (after the part name of the viewport top part).

Creo Elements/Direct Drafting redraws the assembly. The hidden lines are now invisible. See [Figure 188. Assembly View with Hidden Lines Invisible](#) on page 480.

Figure 188. Assembly View with Hidden Lines Invisible



Assembly – Hidden Lines Shown Dashed

At this point you will generate the actual hidden lines.

Caution

It is recommended to store your assembly drawing (in `workfile.hid`) before generating hidden lines:

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Generate**.
2. Click  **with Backup**.

The  **Generate** buttons smash the part structure and create the hidden line geometry.

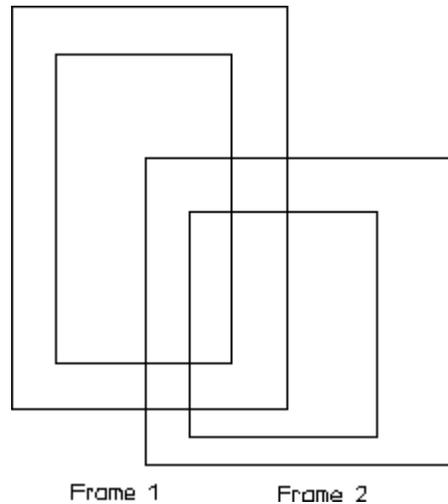
To generate hidden lines:

1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **Edit**.
2. Click  **Top**.
3. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Generate**.
4. Click  **with Backup**.
5. Click  **Confirm**.

Creo Elements/Direct Drafting stores your original drawing (in your current directory) and generates hidden lines in the version displayed on the screen.

Creo Elements/Direct Drafting redraws the assembly with the hidden lines shown dashed; see [Figure 189. Assembly with Hidden Lines Shown Dashed](#) on page 481.

Figure 189. Assembly with Hidden Lines Shown Dashed



Setting the Current Offset

The current offset in the z-direction is used to calculate the z-value used in the above or below or between relationship with another element. The current offset z-value is added to or subtracted from the z-value of a reference element.

To set the current offset:

1. Click **Hidden** and then, in the **Create Element** group, click  **Set Z Values**.
The **HL Z Values** dialog box opens.
2. In the **Current Offset** box, type the current offset z-value.
3. Click **OK**.

Setting Z-Values

This section provides information about assigning and changing element z-values.

Use  **Element** to assign z-values.

Use the normal Creo Elements/Direct Drafting selection techniques to select the elements to which you want to assign z-values.

To assign the specified z-value to all elements of a part and its subparts:

1. Select the part.
2. Click **Hidden** and then, in the **Create Element** group, click the arrow next to



3. Click **Element.**

All elements in the part and are assigned the specified z-value. This works recursively down to all subparts.

4. To select additional options for **Element:**

- a. Click **Hidden** and then, in the **Modify** group, click the arrow next to **Options.**

- b. Click the required **Options** button:



To set a z-value:

1. Click **Hidden** and then, in the **Create Element** group, click **Inquire Z Value.**

2. Click **Current Z Value.**

3. Type a numerical value in the user input line and press ENTER.

The z-value is assigned to each drawing element in your selection.

The Creo Elements/Direct Drafting default is that any existing z-values will be overwritten. If you don't want to overwrite z-values that have already been assigned:

1. Click **Hidden** and then, in the **Create Element** group, click the arrow next to



2. Click **Element.**

3. Select the elements to which you wish to assign z-values.

4. Type `KEEP` in the user input line and press ENTER.

5. Type a numerical value in the user input line and press ENTER.

You can also click one of the following  **Options** button in the **Modify** group on the **Hidden** tab:

-  **Above**
-  **Below**
-  **Same**
-  **Between**

This results in a computed z-value.

 **Note**

These  **Options** buttons only influence elements that already have a z-value assigned to them.

If you select a group of elements for a z-value change that have a range of z-values (lower limit — upper limit) already associated with them, the range relationship will be maintained.

To understand the effect of using  **Above**,  **Below**,  **Same**, or  **Between**, refer to the following four examples:

Example 1

Suppose you have a group of elements with computed z-values 1 to 9 and you then specify that they are placed at the same (click  **Same** under  **Options**) z-value as a reference element with a computed z-value of 20.

The result of this operation is that the z-values of all nine elements are computed and set in the range 20 to 28.

Example 2

Suppose you have a group of elements with computed z-values 1 to 9 and you then specify that they are placed above (click  **Above** under  **Options**, specify **Current Offset** as 1) a reference element with a computed z-value of 20.

The result of this operation is that the z-values of all nine elements are computed and set in the range 21 to 29.

Example 3

Suppose you have a group of elements with computed z-values 1 to 9 and you then specify that they are placed below (click  **Below** under  **Options**, specify **Current Offset** as 1) a reference element with a computed z-value of 20.

The result of this operation is that the z-values of all nine elements are computed and set in the range 11 to 19.

Example 4

Suppose you have a group of elements with computed z-values 1 to 9 and you then specify that they are placed between (click  **Between** under  **Options**, specify **Current Offset** as 1) a reference element with a computed z-value of 20 and another reference element with a computed z-value of 40.

The result of this operation is that the z-values of all nine elements are computed and set in the range 21 to 29.

Assigning a Z-Value to an Element

To assign a z-value to an element:

1. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
2. Click  **Element**.
3. Select the element to which you want to assign a z-value.
4. Type the z-value in the user input line and press ENTER.

Assigning a Z-Value to a Shared Part

To assign a z-value to a shared part:

1. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
2. Click  **Shared Part**.
3. Select the shared part to which you want to assign a z-value.
4. Type the z-value in the user input line and press ENTER.

Deleting a Z-Value

To delete a z-value of an element or a shared part:

-
1. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
 2. Click  **Element**.
 3. Select the element or elements.
 4. In the **Modify** group of the **Hidden** tab, click the arrow next to  **Options**.
 5. Click  **None**.

Positioning an Element at a Z-Value Above Another Element

To position an element at a z-value above another element:

1. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
2. Click  **Element**.
3. Select the elements to be set above the reference element.
4. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.
5. Click  **Above**.
6. Select the reference element.

Positioning an Element at a Z-Value Below Another Element

To position an element at a z-value below another element:

1. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
2. Click  **Element**.
3. Select the elements to be set below the reference element.
4. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.
5. Click  **Below**.
6. Select the reference element.

Assigning an Element's Z-Value to Another Element

To set the same z-value:

1. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
2. Click  **Element**.
3. Select the elements to be set to the same level as the reference element.
4. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.
5. Click  **Same**.
6. Select the reference element.

Positioning an Element at a Z-Value Between Two Other Elements

To insert an element at a z-value between two other elements:

1. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
2. Click  **Element**.
3. Select the elements to be set to a level between two existing elements.
4. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.
5. Click  **Between**.
6. Select the lower reference element.
7. Select the upper reference element.

Generating Faces and Setting Z-Values in a Complete Part

This section provides information about generating faces and setting z-values in a complete part using  **Elements & Faces**. Using  **With Holes**, you can detect the presence of holes to prevent them being generated as faces.

Generating All Faces and Setting All Z-Values

To automatically generate all faces and set all z-values for all elements in a complete part:

1. Click **Hidden** and then, in the **Create Part** group, click  **Elements & Faces**.
2. Select the part.
3. Type the z-value in the user input line and press ENTER.

or

Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.

Click  **Above** or  **Same** or  **Below** and then click the reference element.

4. Click  **OK**.

Creo Elements/Direct Drafting generates all faces and assigns z-values to all elements in your part.

Generating All Faces - Ignoring Holes and Setting All Z-Values

To automatically generate all faces ignoring any holes and set all z-values for all elements in a complete part:

1. Click **Hidden** and then, in the **Create Part** group, click  **With Holes**.
2. Select the part.
3. Type the z-value in the user input line and press ENTER.

or

Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.

Click  **Above** or  **Same** or  **Below** and then click the reference element.

4. Click  **OK**.

Creo Elements/Direct Drafting generates all faces ignoring any holes and assigns z-values to all elements.

Checking Z-Values

This section provides information about how to check z-values.

An Element's Z-Value

To check the z-value of an individual element:

1. Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
2. Click  **Absolute**.
3. Select the element.

Creo Elements/Direct Drafting displays the element's z-value in the user input line (if the specified element has no assigned z-value, an error beep is given). The value is also written in the System Inquiry Array.

An Element Belonging to a Shared Part

If the element whose z-value you wish to know belongs to a shared part, the z-value is computed as the z-value of the element itself plus the z-values of all parent shared parts.

Example

See [Figure 190 on page 489](#).

To check the computed z-value of an individual element with z-value 2 in Shared_part1:

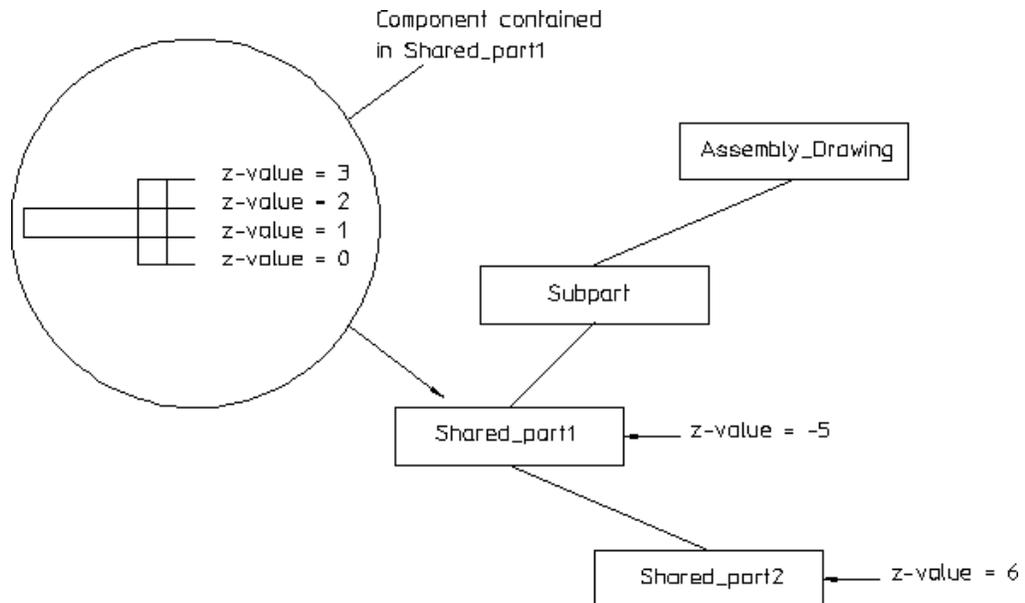
1. Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
2. Click  **Absolute**.
3. Select an element with z-value 2.

Creo Elements/Direct Drafting calculates the element's z-value as follows:

Element at z-value 2 + z-value of Shared_part1.

$$2 + -5 = -3.$$

Figure 190. Z-Values for Shared Parts



The Minimum Z-Value in an Assembly Drawing

To check the minimum z-value:

1. Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
2. Click  **Minimum**.

Creo Elements/Direct Drafting displays the minimum z-value in the assembly drawing in the user input line (if the drawing has no z-values assigned, an error beep is given). The value is also written in the System Inquiry Array.

The Maximum Z-Value in an Assembly Drawing

To find the maximum z-value:

1. Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
2. Click  **Maximum**.

Creo Elements/Direct Drafting displays the maximum z-value in the assembly drawing in the user input line (if the drawing has no z-values assigned, an error beep is given). The value is also written in the System Inquiry Array.

The Z-Value of a Shared Part

To find the computed z-value of a shared part:

1. Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
2. Click  **Shared Part Offset**.
3. Select the shared part.

Creo Elements/Direct Drafting computes the z-value as the sum of the shared part's z-value and that of the shared parent part and displays it in the user input line. The value is also written in the System Inquiry Array.

Example

Refer to [Figure 203 on page 489](#).

To check the z-value of Shared_part2:

1. Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
2. Click  **Shared Part Offset**.
3. Click Shared_part2.

Creo Elements/Direct Drafting calculates the shared part's z-value as follows:

Z-value of Shared_part2 + z_value of parent. In this case, $-5 + 6 = 1$.

The Z-Value of an Element in a Shared Part

To find the z-value of an element in a shared part:

1. Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
2. Click  **Shared Part Element**.
3. Select the element in the shared part.

Creo Elements/Direct Drafting displays the element's z-value in the user input line. The value is also written in the System Inquiry Array.

The Current Z-Value

To check the default current z-value assigned to each newly created drawing element:

- Click **Hidden** and then, in the **Create Element** group, click  **Set Z Values**.

The **HL Z Values** dialog box opens.

The **Current Value** box in the **HL Z Values** dialog box displays the current z-value. The current z-value is also written in the System Inquiry Array.

The Load Value

To check the current load z-value:

- Click **Hidden** and then, in the **Create Element** group, click  **Set Z Values**.

The **HL Z Values** dialog box opens.

The **Load Value** box in the **HL Z Values** dialog box displays the current load z-value. The current load z-value is also written in the System Inquiry Array.

Setting the Default Z-Value

This section provides information about defining a specific z-value which is assigned to each newly created element (the default current z-value is 0).

To set the current z-value:

- Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
- Click  **Current Z Value**.
- Type the default z-value in the user input line and press ENTER.

Alternatively, click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.

Click  **Above** or  **Same** or  **Below** and then click the reference element.

Creo Elements/Direct Drafting assigns this value as the current z-value, assigned by default to each newly created element.

Setting the Load Value

This section provides information about defining a loading offset in the z-direction for elements that are loaded in the future.

The default load z-value is **None**. If you load elements with the load z-value set to **None**, their z-value (if any) remains unchanged.

To set a load offset z-value:

- Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.

- Click  **Load Z Value**.
- Type the relative load z-value in the user input line and press ENTER.
Alternatively, click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.

Click  **Above** or  **Same** or  **Below** and then click the reference element.

Creo Elements/Direct Drafting assigns the required load z-value to any elements loaded in the future.

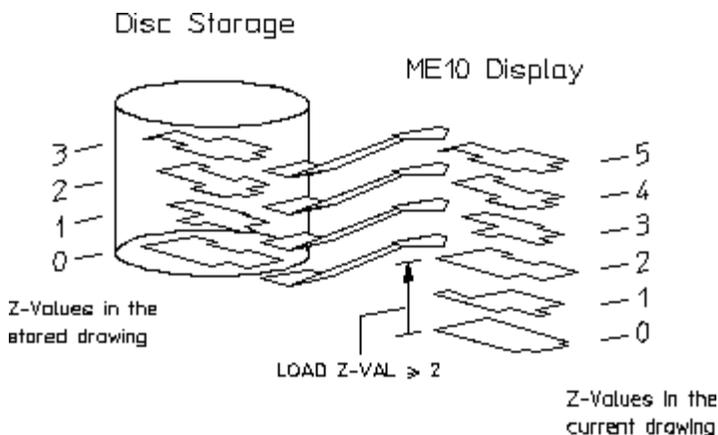
Example

To load drawing elements with a load z-value of 2:

- Click **Hidden** and then, in the **Create Element** group, click  **Inquire Z Value**.
- Click  **Load Z Value**.
- Type 2 in the user input line and press ENTER.
- Load an existing drawing file.

Creo Elements/Direct Drafting loads the drawing elements as shown in [Figure 191 on page 492](#).

Figure 191. Loading Drawing Elements with a Load Z-Value of ≥ 2



Generating Faces

This section describes how to specify covering faces. A covering face will effectively cover all geometry (or faces) with a lower z-value. Newly created faces are assigned the current z-value by default. The covering face will adopt the default face color (note that you can set a default color for faces).

Automatically

You can automatically generate faces if:

- All elements of the contour are in the active part.
- The contour has no gaps.
- Each vertex of the contour is a vertex of all elements there.

To automatically generate faces:

1. Click **Hidden** and then, in the **Create Face** group, click  **Single Automatic**.
2. Specify a point inside the face.
3. Type a z-value in the user input line and press ENTER.

or

Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.

Click  **Above** or  **Same** or  **Below** and then click the reference element.

Creo Elements/Direct Drafting generates the face automatically and assigns the required z-value.

Manually

In order to build up a contour with elements, a vertex of an element must be specified first. Thereafter, the elements which make up the contour can be selected by a simple click.

Creo Elements/Direct Drafting will allow you to skip elements along a contour as long as not more than one element after any branch point is skipped. All elements must be in the active part.

To manually generate a face:

1. Click **Hidden** and then, in the **Create Face** group, click  **Single Manual**.

- Specify points along the outer contour (the points selected do not have to lie on any geometric element).
- Specify points along the inner contour.
- Click  **OK**.
- Type a z-value in the user input line and press ENTER.

or

Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.

Click  **Above** or  **Same** or  **Below** and then click the reference element.

All Faces in a Part Excluding Holes

To automatically generate all faces in a part and exclude any holes:

- Click **Hidden** and then, in the **Create Face** group, click  **With Holes**.
- Type a z-value in the user input line and press ENTER.

or

Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.

Click  **Above** or  **Same** or  **Below** and then click the reference element.

Creo Elements/Direct Drafting generates the faces automatically and assigns the required z-values. Any holes that are present are excluded from face generation.

Automatically Generating All Faces in a Part

You can generate faces automatically in the active part. Creo Elements/Direct Drafting tries to find the maximum number of faces.

To automatically generate faces in the active part:

- Click **Hidden** and then, in the **Create Face** group, click  **All in Part**.
- Type a z-value in the user input line and press ENTER.

or

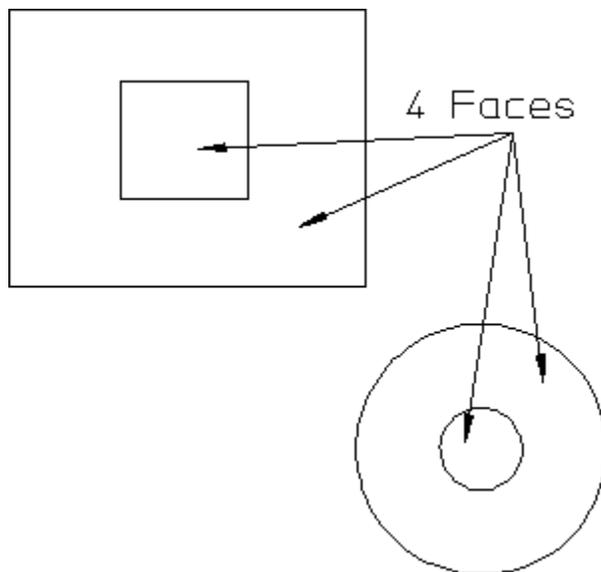
Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Options**.

Click  **Above** or  **Same** or  **Below** and then click the reference element.

Example

Refer to [Figure 192 on page 495](#). In this example, Creo Elements/Direct Drafting would generate the four faces shown.

Figure 192. Automatically Generating Faces In a Part



Undoing a Face

You may want to undo a face you have just generated (before specifying the z-value).

The effect of clicking  **Undo** when using the following buttons in the **Create Face** group on the **Hidden** tab is as follows:

 Single Automatic	The last face specified is removed.
 Single Manual	During creation of a contour, points are removed. If you have finished defining a face and have clicked  OK , clicking  Undo results in the removal of the entire face.

 All in Part	All faces generated automatically are removed.
 With Holes	All faces generated automatically are removed.

Deleting Covering Faces

This section shows how to delete covering faces. You can delete a single face or all faces in a part. The faces identified must be in the current part and they should not contain any hatching.

A Single Face

To delete a single face:

1. Click **Hidden** and then, in the **Modify** group, click  **Delete Face**.
2. Select the face you wish to delete.

Creo Elements/Direct Drafting deletes the face.

All Faces in a Part

To delete all the faces in the current part:

1. Click **Hidden** and then, in the **Modify** group, click  **Delete Face**.
2. Type ALL in the user input line and press ENTER.

Creo Elements/Direct Drafting deletes all faces in the current part.

Redrawing in Hidden or Normal Mode

This section shows how to switch between the normal Creo Elements/Direct Drafting redraw function and the hidden line redraw function.

Note

Hidden Mode affects all **View** functions as each takes the z-value information into account.

Hidden Mode

To redraw in hidden mode:

1. Click **Hidden** and then in the **Modify** group, click the **Hidden Mode** check box.
2. Specify the required viewport to use the z-value information when redrawing.
Creo Elements/Direct Drafting displays the drawing partially covered (depending on the z-values associated with the assembly drawing).

 **Note**

Creo Elements/Direct Drafting displays **-H-** in the upper left-hand corner of the Creo Elements/Direct Drafting window to indicate hidden mode.

Normal Mode

This is the default mode. To redraw normally:

1. Click **Hidden** and then in the **Modify** group, clear the **Hidden Mode** check box.
2. Specify the appropriate viewport.

Hidden Settings

The **Hidden Settings** dialog box allows you to define the required hidden line type, line color and face color settings,

To open the **Hidden Settings** dialog box:

- Click **File** ► **Settings** ►  **Hidden**.

The **Hidden Settings** dialog box enables you to define:

- Line settings
- Face settings

Line Settings

In the **Line** area of the **Hidden Settings** dialog box, define the following settings:

Type	Select the line type for hidden lines in your drawing.
Color	Select the color for hidden lines in your drawing. If you click the Default check box, the default hidden line color is applied. Refer to Defining Custom Colors on page 100 for instructions on defining custom colors.
Width	Type the width of the hidden line.

Face Settings

In the **Face** area of the **Hidden Settings** dialog box, select the color for faces in the hidden mode. If you click the **Default** check box, the default face color is applied. Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

Changing the Linetype and Color of Hidden Lines

The **Change Hidden Line** dialog box enables you to change hidden lines (see [Generating Hidden Lines on page 470](#) for instructions on generating hidden lines). To change the linetype and/or color of existing hidden lines:

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Edit**.
2. Click  **Line**.

The **Change Hidden Line** dialog box opens.

3. In the **Type** box, select a linetype.
4. In the **Color** box, select a color.

Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

5. Click **OK**.

All existing hidden lines in the drawing are redrawn in the linetype and line color you selected.

Changing Face Color

The **Change Hidden Face** dialog box enables you to change the color of faces you created to generate hidden lines.

To change the color of faces:

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Edit**.
2. Click  **Face**.

The **Change Hidden Face** dialog box opens.

3. In the **Color** box, select a color.

4. If you click the **None** check box, the face color is set to none (that is, faces will be transparent). Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.
5. Click **OK**.
6. Select the face or faces to be changed.

The selected face or faces are redrawn in the color you selected.

Viewing Z-Values

The **Show Range** dialog box enables you to view all or a range of z-values in your drawing. You can display all elements or only faces (in a color of your choice) that are in a specified range of z-values.

To open the **Show Range** dialog box:

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Show**.
2. Click  **Range**.

The **Items** area of the **Show Range** dialog box enables you to select one of the following:

- **Elements in Drawing**
- **Faces in Drawing**
- **Elements in Part**
- **Faces in Part**

Elements in All Parts

Click **Elements in Drawing** in the **Items** area of the **Show Range** dialog box to select all elements:

- In a selected range of z-values.
- In all parts (default).

Elements in Current Part

Click **Elements in Part** in the **Items** area of the **Show Range** dialog box to select elements:

- In a selected range of z-values.
- In the current part.

Faces in All Parts

Click **Faces in Drawing** in the **Items** area of the **Show Range** dialog box to select faces:

- In a selected range of z-values.
- In all parts.

Faces in Current Part

Click **Faces in Part** in the **Items** area of the **Show Range** dialog box to select faces:

- In a selected range of z-values.
- In the current part.

Setting a Range of Z-Values

By default, all z-values are shown. In the **Show Range** dialog box, you can limit the visible range of z-values as follows:

1. In the **Range** area of **Show Range** dialog box, click **Limit**.
2. In the **Lower** box, type the "from" z-value.
3. In the **Upper** box, type the "to" z-value.

For example, to limit the visible range to z-levels 3 through 8, you would type 3 in the **Lower** box and 8 in the **Upper** box.

This range is used in the following way:

- In connection with **Elements in Drawing** and **Elements in Part**, this range specifies the range of z-values belonging to elements that are visible.
- In connection with **Faces in Drawing** and **Faces in Part**, this range specifies the range of z-values belonging to faces that are visible and colored.

Applying a Color to Facilitate Visualization

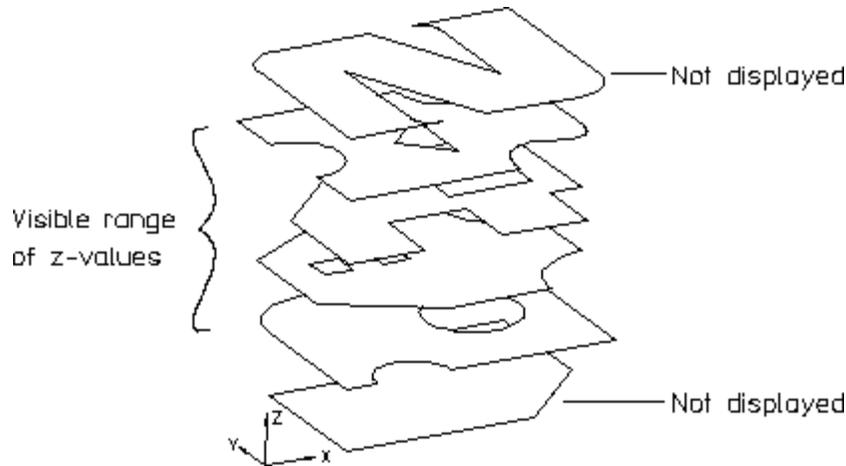
To color faces in a particular range of z-values as an aid to visualization:

1. In the **Items** area of the **Show** dialog box, click the required item for the items to be displayed.
2. In the **Range** area of the **Show** dialog box, specify the range of z-values to be displayed in the same color (see [Setting a Range of Z-Values on page 500](#)).
3. In the **Show** area of the **Show Range** dialog box, click **Color** and select a color. Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

4. Click **OK**.

Creo Elements/Direct Drafting displays the selected faces in the required color and does not display those faces with a z-value outside of the desired range. See [Figure 193 on page 501](#).

Figure 193. Viewing Selected Elements



Resetting the Viewport after Viewing

After using the **Show Range** dialog box to display selected elements or geometry, you need to reset the viewport so that you can see all elements or geometry again. To reset the viewport:

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Show**.
2. Click  **Show Reset**.

Generating Hidden Lines

This section shows how to generate hidden lines and load the backup drawing (`workfile.hid`) produced just before generating hidden lines.

When generating hidden lines, Creo Elements/Direct Drafting does the following:

- Geometry is modified by splitting geometric components if they are covered by faces with a higher z-value.
- Shared parts are unshared.
- The parts tree, starting from the current part, will be smashed.

Caution

It is recommended to store your assembly drawing (in `workfile.hid`) before generating hidden lines:

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Generate**.
2. Click **with Backup**.

The  **Generate** buttons smash the part structure and create the hidden line geometry.

To generate hidden lines:

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Generate**.
2. Click  **with Backup** or  **without Backup**.

Loading the Backup Drawing

If you wish to recover the backup drawing (`workfile.hid`) produced by clicking  **with Backup**:

1. Store your hidden line view.
2. Click **Hidden** and then, in the **Modify** group, click  **Load Backup**.

Creo Elements/Direct Drafting will load the original drawing (`workfile.hid`) created before you generated hidden lines.

Plotting Hidden Lines

It is not possible to plot face colors.

To suppress plotting of hidden lines, you must set the hidden line linetype and color appropriately. Refer to the `PLOT_TRANSFORMATION` command in Writing Macros with Creo Elements/Direct Drafting.

Dumping Hidden Lines

You can use the screen dumping commands to produce a copy of your drawing that includes face colors (if you have a color printer). Screen dumps can be made at any stage in the drawing process. Refer to [Printing and Plotting on page 606](#).

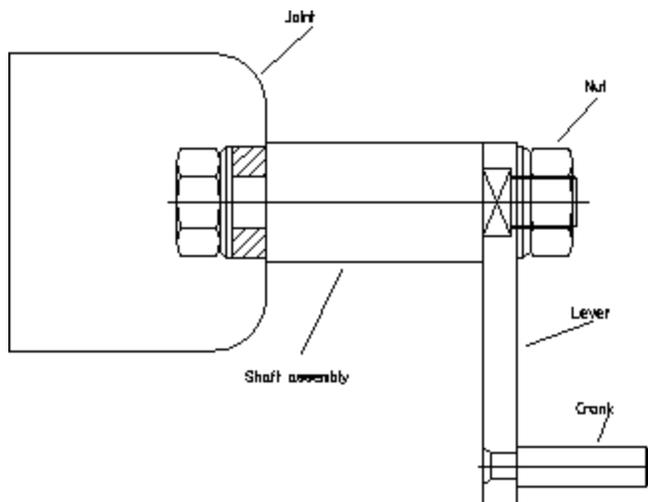
Hidden Line - Tutorial Example 2

This tutorial example shows one way to create an assembly drawing (CRANK_ASSEMBLY) from five individual part drawings. The assembly drawing is then used to demonstrate the various features of the Creo Elements/Direct Drafting 2D Hidden Line Module.

Creating the Crank Assembly

This tutorial is based on the crank assembly shown in [Figure 194 on page 503](#).

Figure 194. CRANK_ASSEMBLY



This crank assembly consists of the top part named CRANK_ASSEMBLY and five subparts. The parts tree is indicated in the following list:

CRANK_ASSEMBLY

Nut

Lever

Crank

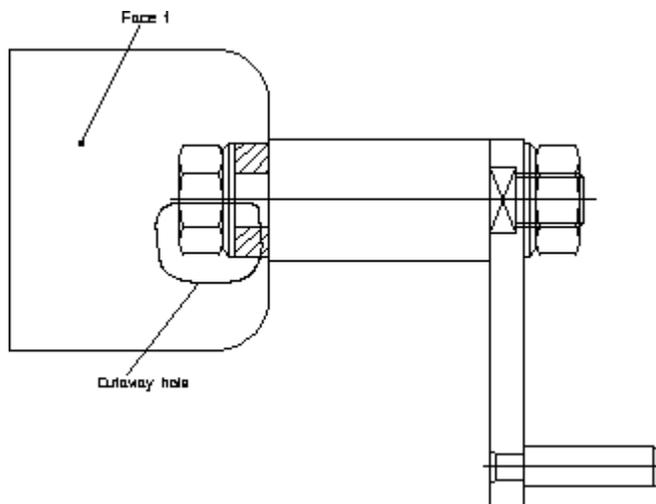
Joint

Shaft assembly

Draw a Cutaway Hole in the Joint

1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **Edit**.
2. Click  **Part**.
3. Click the Joint to make it the active part.
4. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Spline**.
5. Click  **Interpolation Spline**.
6. Click the **Closed** check box.
7. Click points for the spline to create the cutaway hole shown in [Figure 195 on page 504](#).

Figure 195. CRANK_ASSEMBLY With Cutaway Hole



8. Click  **OK**.

Create Face 1

1. Click **Hidden** and then in the **Modify** group, click the **Hidden Mode** check box.
2. Click the current viewport to get visual feedback while creating faces (the face is highlighted).
3. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
4. Click  **Element**.

5. Enclose the Joint in a box.
6. When prompted for the z-value, type 1 in the user input line and press ENTER.
Creo Elements/Direct Drafting assigns a z-value of 1 to all elements in the box.
7. Click  **OK**.

Generate Face 1

1. Click **Hidden** and then, in the **Create Face** group, click  **Single Automatic**.
2. Click a point inside Face 1.
3. When prompted for the z-value, type 2 in the user input line and press ENTER.
4. Click  **OK**.
5. Click **View** and then, in the **Windows** group, click  **Redraw**.

Generate Faces in the Nut

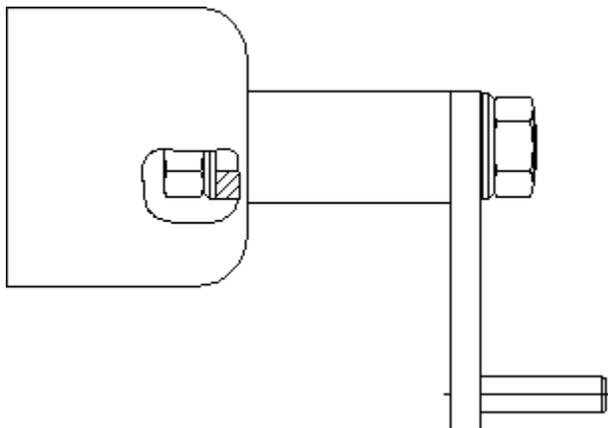
1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **Edit**.
2. Click  **Part**.
3. Click the Nut to make it the active part.
4. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
5. Click  **Element**.
6. Enclose the Nut in a box.
7. When prompted for the z-value, type 1 in the user input line and press ENTER.
Creo Elements/Direct Drafting assigns a z-value of 1 to all elements in the box.
8. Click  **OK**.
9. In the **Create Face** group of the **Hidden** tab, click  **All in Part**.
10. Click **View** and then, in the **Viewing** group, click  **Redraw**.

Generate a Face in the Lever

1. Click **Structure** and then, in the **Parts** group, click the arrow next to  **Edit**.
2. Click  **Part**.
3. Click the Lever to make it the active part.
4. Click **Hidden** and then, in the **Create Element** group, click the arrow next to  **New**.
5. Click  **Element**.
6. Enclose the Lever in a box.
7. When prompted for the z-value, type 1 in the user input line and press ENTER.
Creo Elements/Direct Drafting assigns a z-value of 1 to all elements in the box.
8. Click  **OK**.
9. Click **Hidden** and then, in the **Create Face** group, click  **All in Part**.
10. Click **View** and then, in the **Viewing** group, click  **Redraw**.

At this point you would check for any errors you may have made when assigning z-values and generating faces. See [Figure 196 on page 506](#).

Figure 196. CRANK_ASSEMBLY with Hidden Mode On



Generate Hidden Lines

1. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Generate**.
2. Click  **with Backup**.

Caution

It is recommended to store your assembly drawing (in `workfile.hid`) before generating hidden lines:

- a. Click **Hidden** and then, in the **Modify** group, click the arrow next to  **Generate**.
- b. Click **with Backup**.

The  **Generate** buttons smash the part structure and create the hidden line geometry.

Creo Elements/Direct Drafting generates hidden lines as shown in [Figure 197](#) on page 507, [Figure 198](#) on page 508, and [Figure 199](#) on page 508.

Figure 197. Cutaway Hole and Hidden Lines

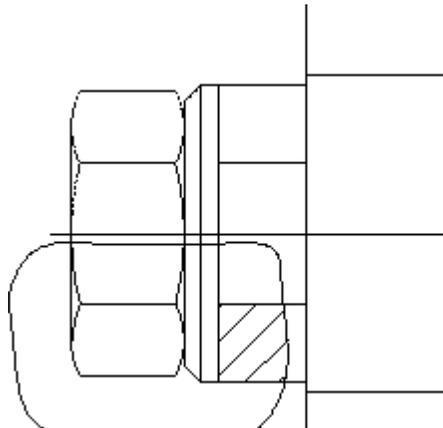


Figure 198. Nut — Showing Hidden Lines

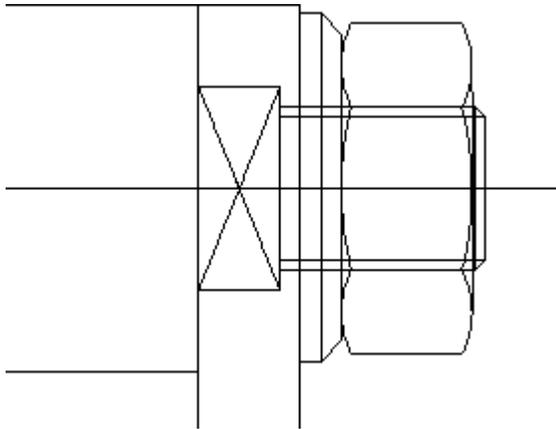
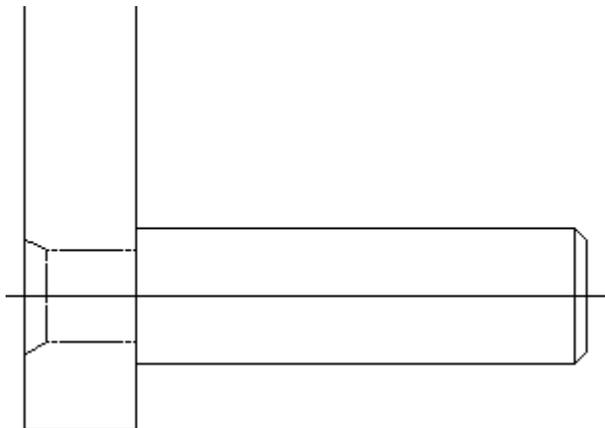


Figure 199. Lever — Showing Hidden Lines



3. Click **Hidden** and then, in the **Modify** group, click or clear the **Hidden Mode** check box to toggle the hidden lines on or off, respectively.

Further Information

For further technical information about hidden lines, refer to the Creo Elements/Direct Drafting help system.

Type the following commands in the user input line and press ENTER:

- `help hl_set_z_value`
- `help hl_inq_z_value`
- `help wui_hl_z_values`
- `help hl_set_curr_z_value`
- `help hl_set_load_value`

-
- `help hl_set_relation_offset`
 - `help hl_gen_all_part`
 - `help hl_generate_face`
 - `help wui_change_hidden_line`
 - `help wui_change_hidden_face`
 - `help hl_delete_face`
 - `help hl_generate_hidden`
 - `help hl_redraw_mode`
 - `help wui_hidden_show_range`
 - `help wui_hidden_settings`

Creo Elements/Direct Drafting displays further information about hidden lines.

16

Using Layers and Infos

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Drawing Using Layers

Creo Elements/Direct Drafting allows you to split up your drawing into a number of layers using the buttons in the following groups on the **Structure** tab:

- **Info**
- **Layer**
- **URL**

The advantage of working this way is mainly in visualization. By switching layers on and off you can simplify what you see on the screen.

Example

Suppose you wish to create an assembly drawing of a cap bearing consisting of four components. Each component was stored as a drawing, and you must load each one to create a four part drawing.

Figure 190. Cap Bearing Parts Structure on page 511 shows the parts structure.

Figure 190. Cap Bearing Parts Structure



Irrespective of the parts structure, you want to be able to see the assembly in three different ways:

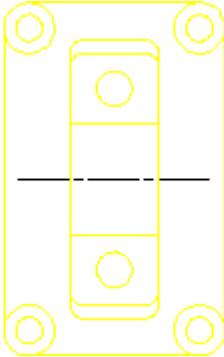
- Each of the components individually.
- With the cap removed.
- Fully assembled.

To do this we can use the three layer commands and place the various elements into layers; some elements would be contained in more than one layer.

Finally, via the **Layer** tab in the **Show** dialog box, you can switch the various layers on and off to produce the required views. The layers would be as follows:

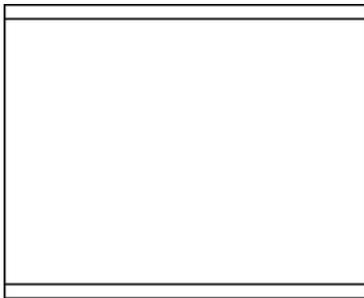
- Layer: 1
Contains all the HOUSING elements.

Figure 191. Layer 1 — Housing Elements



- Layer: 2
Contains all the BEARING elements.

Figure 192. Layer 2 — Bearing Elements



- Layer: 3
Contains all the SHAFT elements.

Figure 193. Layer 3 — Shaft Elements



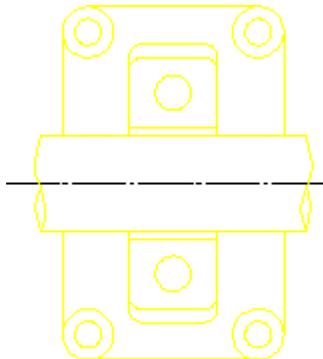
- Layer: 4
Contains all the CAP elements.

Figure 194. Layer 4 — CAP Elements



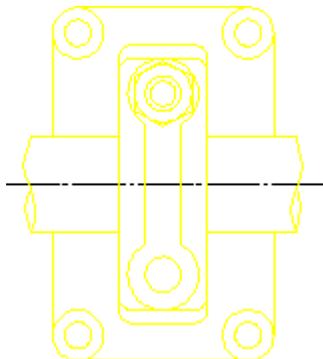
- Layer: 5
Contains most of the HOUSING elements, all the BEARING elements and some of the SHAFT elements.

Figure 195. Layer 5 — Housing, Bearing and Shaft Elements



- Layer: 6
Contains some of the HOUSING and SHAFT elements, plus all the CAP ones.

Figure 196. Layer 6 — Housing, Shaft and CAP Elements



Adding a New Layer

To add a new layer to your drawing:

1. Click **Structure** and then, in the **Layer** group, click  **Add**.
2. Type a unique number or name for this new layer in the user input line and press ENTER.
3. Select the elements to be added to the new layer by using one of the following methods:
 - Click or box the elements to be added. This method is most suitable for simple selections of one or two elements.
 - Alternatively, click **Utilities** and then, click  **Select** >  **Advanced** to use the various **Select** options. When your selection is complete, click **Confirm** and **OK**. Creo Elements/Direct Drafting adds the elements to the selected layer. This method is most suitable for large selections as you can see which elements have been identified so far.

Setting the Active Layer

When a particular layer is active, all subsequently drawn elements will be placed in it.

To set a layer active:

1. Click **Structure** and then, in the **Layer** group, click  **Current**.
2. Type the unique number or layer name in the user input line and press ENTER.

Duplicating Elements in Other Layers

This section describes how to duplicate elements contained in one layer in another. After using the function the selected elements will be in both layers. The function can be used irrespective of which layer is current.

To duplicate elements contained in one layer in another:

1. Click **Structure** and then, in the **Layer** group, click  **Add**.
Creo Elements/Direct Drafting prompts for the layer number or layer name to which elements are to be added.
2. Type the required layer number or layer name in the user input line and press ENTER.

-
3. Select the elements to be duplicated to the new layer by using one of the following methods:
 - Click or box the elements to be added. This method is most suitable for simple selections of one or two elements.
 - Alternatively, click **Utilities** and then click  **Select** ►  **Advanced** to use the various **Select** options. When your selection is complete, click **Confirm** and **OK**. Creo Elements/Direct Drafting adds the elements to the selected layer. This method is most suitable for large selections as you can see which elements have been identified so far.

Moving Elements Between Layers

This section describes how to transfer elements from one layer to another. The function can be used irrespective of which layer is current.

To transfer elements from one layer to another:

1. Click **Structure** and then, in the **Layer** group, click  **Change**.
2. Type the layer number or layer name to which elements are to be transferred.
3. Select the elements to transfer by using one of the following methods:
 - Click or box the elements to be added. This method is most suitable for simple selections of one or two elements.
 - Alternatively, click **Utilities** and then click  **Select** ►  **Advanced** to use the various **Select** options. When your selection is complete, click **Confirm** and **OK**. Creo Elements/Direct Drafting adds the elements to the selected layer. This method is most suitable for large selections as you can see which elements have been identified so far.

Using Layer in Show Dialog Box

To use the **Layer** tab on the **Show** dialog box:

1. Click **File** ► **Settings** ► **Show**.
Alternatively, click **Show** in the **Utilities** group.
The **Show** dialog box opens.
2. Click the **Layer** tab.

In the **Layer** tab, you can:

- Switch layers on and off.

-
- Box an area to define a layer.
 - Select a color for each layer.

Switching Layers On & Off

To switch layers on or off:

1. Click **File** ► **Settings** ► **Show**.

Alternatively, click **Show** in the **Utilities** group.

The **Show** dialog box opens.

2. Click the **Layer** tab.
3. Select the layer or layers you want to switch on or off.
4. Click **On** or **Off** to switch on (displayed) or off (hidden), respectively.

Showing Layer Color

You can color a layer to aid visualization. This does not affect plotter pen numbering.

To color a layer:

1. Click **File** ► **Settings** ► **Show**.

Alternatively, click **Show** in the **Utilities** group.

The **Show** dialog box opens.

2. Click the **Layer** tab.
3. Select the required layer or layers.
4. In the **Color** box, select a color.
5. Click **Color** to apply the layer color change.

Creo Elements/Direct Drafting redraws the elements of the specified layer in the selected color.

To restore the original color of the layer, select the layer and then click **On** in the **Layer** tab.

Modifying and Deleting Layers

There may be occasions when you wish to modify or delete complete layers. In such cases use the **Layer** option in the **Select** window.

To modify or delete complete layers:

-
1. Click  **Delete** or click the required  **Modify** button.
 2. In the **Utilities** group, click  **Select** >  **Advanced**.
The **Select** window opens.
Alternatively, press F2 to open the **Select** window.
 3. Click **Layer**.
The **Layer** dialog box opens.
 4. Type the layer number or layer name that you want to delete or modify.
 5. Click **Confirm** and then click **OK**.

What are Infos?

The major components of a drawing are drawing elements such as lines, circles, arcs, and so on. Each element is stored in memory together with information to describe it. These pieces of information attached to each element are known as attributes.

Examples of attributes are an element's linetype and color. In addition to these, Creo Elements/Direct Drafting allows you to enter your own attributes, called Infos.

An info is simply a piece of text attached to an element. The info is not visible on the screen, but is attached to the element in memory. The use of infos is a matter of personal or company preference. They can be used to:

- Store information concerning a design update. The revised elements could all have the following infos added to them:
 - Revision number:003
 - Nature of revision: Dimensional changes to accommodate
 - uprated bearings
- Store information concerning dates and names:
 - Designer: Bill G
 - Date: 03.03.93
 - Job No: 147/2

In addition to whatever use you make of infos, Creo Elements/Direct Drafting uses them when you use the **Layer** buttons. Thus if you place an element in layer 1, Creo Elements/Direct Drafting will attach the info `LAYER: 1` to it.

Adding Infos

This section describes how to add infos to drawing elements.

Adding the Current Info

You can create infos as current to attach them to all subsequently drawn elements.

To add the current info:

1. Click **Structure** and then, in the **Info** group, click the arrow next to  **Add**.
2. Click  **Current Element**.
3. Enter the info text using one of the following methods:
 - Type the required text enclosed in quotes (' ' or " ") in the user input line. This is the best method to use for infos having length of one line.
 - Alternatively, click the arrow next to  **Add** and click  **Editor** to display a text box (based on a standard text editor) in which you can type the info text. Avoid enclosing the text in quotes. When you have finished typing the info, close this text box to return to the normal Creo Elements/Direct Drafting display.

Once the current info has been set, you can continue drawing. All new elements will automatically contain the current infos.

To a Selected Element

To add infos to existing drawing elements:

1. Click **Structure** and then, in the **Info** group, click the arrow next to  **Add**.
2. Click  **Element**.
3. Enter the info text using one of the following methods:
 - Type the required text enclosed in quotes (' ' or " ") in the user input line. This is the best method to use for infos having length of one line.
 - Alternatively, click the arrow next to  **Add** and click **Editor** to display a text box (based on a standard text editor) in which you can type the info text. Avoid enclosing the text in quotes. When you have finished typing the info, close this text box to return to the normal Creo Elements/Direct Drafting display.

Once the info text has been entered, you can add it to any existing elements.

-
4. Select the required elements. Creo Elements/Direct Drafting adds the info to them. Alternatively, you can use the **Select** window to select the elements.

Editing an Info

This section describes how to edit any existing info.

The Current Info

To edit the current info:

1. Click **Structure** and then, in the **Info** group, click  **Edit**.
2. Click  **Current**.
A text box (based on a standard text editor) opens using which you can edit the info text
3. Edit the info text.
4. When you have finished editing, close the text box to return to the normal Creo Elements/Direct Drafting display. All newly drawn elements contain the revised current infos.

An Existing Element

To edit the info for any existing element:

1. Click **Structure** and then, in the **Info** group, click  **Edit**.
2. Click  **Element**.
A text box (based on a standard text editor) opens using which you can edit the info text
3. Select the drawing element to be edited.
4. Edit the info text.
5. When you have finished editing, close the text box to return to the normal Creo Elements/Direct Drafting display.

Deleting an Info

This section describes how to delete the current info or the element info.

Deleting the Current Info

To delete the current info:

1. Click **Structure** and then, in the **Info** group, click  **Delete**.
2. Click  **Current**.

Creo Elements/Direct Drafting deletes the current info. If you continue drawing, all new elements are without infos.

Deleting Element Info

To delete the infos belonging to any existing drawing elements:

1. Click **Structure** and then, in the **Info** group, click  **Delete**.
2. Click  **Element**.
3. Select the required elements by using one of the following methods:
 - Click or box the elements to be deleted. This method is most suitable for simple selections of one or two elements.
 - Alternatively, click **Utilities** and then click  **Select** ►  **Advanced** to use the various **Select** options. When your selection is complete, click **Confirm** and **OK**. Creo Elements/Direct Drafting adds the elements to the selected layer. This method is most suitable for large selections as you can see which elements have been identified so far.

Creo Elements/Direct Drafting deletes the infos of the selected elements.

Changing Infos

This section describes how to change infos.

For each option, enter the line of info text to be changed, and the new text. You can use wildcards (that is, *, ?, [..], ! and \) in either the info text to be changed, or the new text.

For further information about wildcards, refer to [Storing, Loading and Filing on page 528](#).

Changing the Current Info

To change the current info:

1. Click **Structure** and then, in the **Info** group, click  **Replace**.

-
2. Click  **Current**.
 3. Type the line of the current info to be changed in the user input line and press ENTER.
 4. Type the replacement text in the user input line and press ENTER.
Creo Elements/Direct Drafting changes the current info
Once the change is made, you are ready to continue drawing. The revised current info is added to all new drawing elements.

Changing Element Info

To change any existing info:

1. Click **Structure** and then, in the **Info** group, click  **Replace**.
2. Click  **Element**.
3. Type the line of info text to be changed in the user input line and press ENTER.
For example, "DRAWN ON 28.02.93".
4. Type the replacement text in the user input line and press ENTER.
For example, "DRAWN ON 03.03.93".
5. Select the elements to be changed using one of the following methods:
 - Click or box the elements to be deleted. This method is most suitable for simple selections of one or two elements.
 - Alternatively, click **Utilities** and then click  **Select** ►  **Advanced** to use the various **Select** options. When your selection is complete, click **Confirm** and **OK**. Creo Elements/Direct Drafting adds the elements to the selected layer. This method is most suitable for large selections as you can see which elements have been identified so far.

Creo Elements/Direct Drafting modifies the infos of the selected elements.

Changing All Occurrences

To perform changes to all occurrences of an info:

1. Click **Structure** and then, in the **Info** group, click  **Replace**.
2. Click  **Global**.

-
3. Type the line of info text to be changed in the user input line and press ENTER.
 4. Type the replacement line in the user input line and press ENTER.
Creo Elements/Direct Drafting changes every such occurrence without the need for identifying elements.

Listing Infos

This section describes how to produce a list of all infos used in your drawing.

1. Click **Structure** and then, in the **Info** group, click  **List**.
All infos used in your drawing are displayed in a standard text editor.
2. Close the standard text editor to return to the normal Creo Elements/Direct Drafting display.

Listing Namespaces

Namespaces are used for structuring infos. The use of namespaces in Creo Elements/Direct Drafting enhanced performance through targeted inquiries. For further details, refer to Infotext namespace in the Creo Elements/Direct Drafting Help.

To list all of the namespaces currently in use:

1. Click **Structure** and then, in the **Info** group, click  **Namespaces**.
All namespaces used in your drawing (for example, SYS:HL_SCOPE includes Z-LEVELS) are displayed in the **Info Browser**.
2. Close the **Info Browser** to return to the normal Creo Elements/Direct Drafting display.

Using Info Elem. in Show Dialog Box

To open the **Info Elem.** tab on the **Show** dialog box:

1. Click **File** ► **Settings** ► **Show**.
Alternatively, in the **Utilities** group, click **Show**.
2. Click the **Info Elem.** tab.

In the **Info Elem.** tab, you can:

- Switch infos on or off.

- Box an area to define an info.
- Select a color for each info.

Switching Infos On & Off

To switch infos on or off:

1. Click **File** ► **Settings** ► **Show**.
Alternatively, in the **Utilities** group, click **Show**.
2. Click the **Info Elem.** tab.
3. Select the info you want to switch on or off.
4. Click **On** or **Off** to display or hide the elements containing this info text, respectively.

URLs

Creo Elements/Direct Drafting enables you to attach any industrial standard Web page to drawings, parts and elements. All URLs used by Creo Elements/Direct

Drafting drawings, parts and elements are handled using the  **URL** buttons in the **URL** group on the **Structure** tab.

Adding URLs

To add a URL to the current Creo Elements/Direct Drafting drawing:

1. Click **Structure** and then, in the **URL** group, click the arrow next to  **URL**.
2. Click any of the following:
 - Click  **Current** to add a URL to the current part.
The **Current URLs** dialog box opens.
 - Click  **Element** to add a URL to an element or a group of elements.
Then, select the element or group of elements.
The **Element URLs** dialog box opens.
 - Click  **Part** to add a URL to a part other than the current part. Then select the part.
The **Part URLs** dialog box opens.
3. In the dialog box, type the url you want to add in the text box and click **Add**.

4. Click **OK**.

Changing URLs

To change the URL attached to a part or element:

1. Click **Structure** and then, in the **URL** group, click the arrow next to  **URL**.
2. Click any of the following:
 - Click  **Current** to change the URL of the current part.
The **Current URLs** dialog box opens.
 - Click  **Element** to change the URL of an element or a group of elements. Then, select the required element or group of elements.
The **Element URLs** dialog box opens.
 - Click  **Part** to change the URL of a part other than the current part.
Then select the part.
The **Part URLs** dialog box opens.
3. In the dialog box, select the URL you want to change.
4. Type the revised URL in the text box and click **Change**.
5. Click **OK**.

Deleting URLs

To remove a URL attachment from a part or element:

1. Click **Structure** and then, in the **URL** group, click the arrow next to  **URL**.
2. Click any of the following:
 - Click  **Current** to delete the URL of the current part.
The **Current URLs** dialog box opens.
 - Click  **Element** to delete the URL of an element or a group of elements. Then, select the required element or group of elements.
The **Element URLs** dialog box opens.
 - Click  **Part** to delete the URL of a part other than the current part. Then select the part.

-
- The **Part URLs** dialog box opens.
3. In the dialog box, select the url you want to delete.
 4. Click **Delete**.
 5. Click **OK**.

Listing URLs

To display a list of the URLs attached to the current Creo Elements/Direct Drafting drawing or to a part or to an element in the drawing:

- Click **Structure** and then, in the **URL** group, click  **Show**.
The **Global URLs** dialog box opens. You can view the list of the attached URLs.

Highlighting the Element or Part to which a Given URL is Attached

To highlight the element or part to which a URL is attached:

1. Click **Structure** and then, in the **URL** group, click  **Show**.
The **Global URLs** dialog box opens.
2. Select a url from the list.
3. Click **Show Elem.** to highlight the element associated with the URL.
4. Click **Show Parts** to highlight the part associated with the URL.

Accessing a URL

To access a URL attached to a part or element in a drawing:

1. Click **Structure** and then, in the **URL** group, click the arrow next to  **URL**.
2. Click any of the following:
 - Click  **Current** to access the URL of the current part. The **Current URLs** dialog box opens.
The **Current URLs** dialog box opens.
 - Click  **Element** to access the URL of an element or a group of elements. Then, select the required element or group of elements.
The **Element URLs** dialog box opens.

-
- Click  **Part** to access the URL of a part other than the current part. Then select the part.

The **Part URLs** dialog box opens.

3. In the dialog box, select a URL and then click **Browse** to access this URL by means of your default Web browser or navigator.

Using URL Elem. in Show Dialog Box

To use the **URL Elem.** tab on the **Show** dialog box:

1. Click **File** ► **Settings** ► **Show**.

Alternatively, click **Show** in the **Utilities** group.

The **Show** dialog box opens.

2. Click the **URL Elem.** tab.

In **URL Elem.** tab, you can:

- Switch URLs on and off.
- Box an area to show its URLs.
- Select a color for each URL.

Switching URLs On & Off

To switch URLs on or off:

1. Click **File** ► **Settings** ► **Show**.

Alternatively, click **Show** in the **Utilities** group.

The **Show** dialog box opens.

2. Click the **URL Elem.** tab.
3. Select the URL you want to switch on or off.
4. Click **On** or **Off** to display or hide all the elements associated with the selected URL, respectively.

Assigning a Color to a URL

You can color a URL to aid visualization.

To color a URL:

1. Click **File** ► **Settings** ► **Show**.

Alternatively, click **Show** in the **Utilities** group.

-
- The **Show** dialog box opens.
2. Click the **URL Elem.** tab.
 3. Select a URL.
 4. In the **Color** box, select a color.
 5. Click **Color**.

Creo Elements/Direct Drafting indicates the element in the selected color that contains the URL.

Further Information

For further technical information about layers and infos, refer to the Creo Elements/Direct Drafting help system.

Type the following in the user input line and press ENTER:

- `help infotexts`
- `help add_current_info`
- `help add_elem_info`
- `help screen`
- `help edit_current_info`
- `help edit_elem_info`
- `help change_current_info`
- `help change_elem_info`
- `help change_global_info`
- `help delete_current_info`
- `help delete_elem_info`
- `help url`
- `help wui_edit_global_url`
- `help wui_edit_current_url`
- `help wui_edit_elem_url`
- `help wui_edit_part_url`

Creo Elements/Direct Drafting displays further information about using layers or infos or URLs.

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Storing, Loading, and Filing

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File Organization and Names

This section gives a brief overview of file handling terminology, sufficient to use the **File** functions and commands.

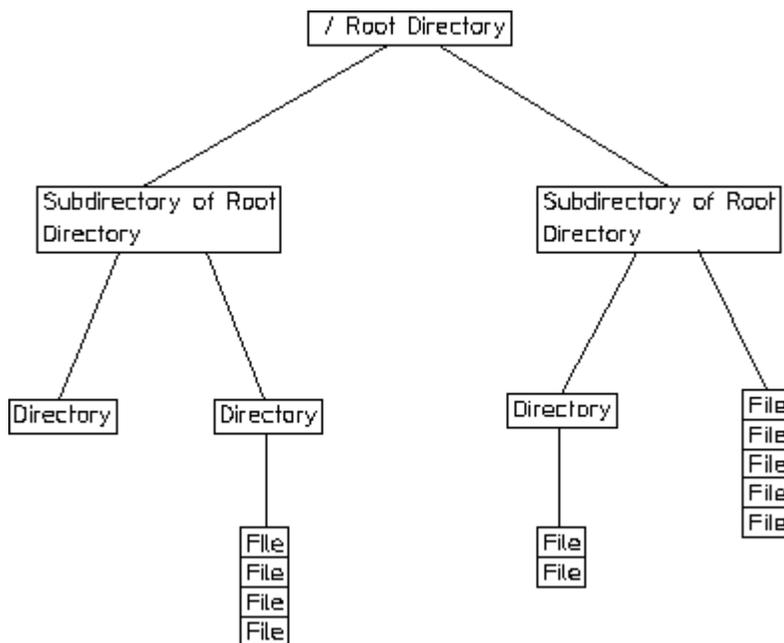
All file access operations can easily be performed using Creo Elements/Direct Drafting filing commands.

File Organization

A file is simply a container into which you can put data. The most important aspect of a file is that it has a name which you have allocated to it. The contents of a file can be displayed, copied, and removed from the system.

A directory is a file containing information about other files and directories. A directory contained within another directory is called a subdirectory, and each subdirectory can itself contain files and more subdirectories. [Figure 197. File System Hierarchy on page 529](#) shows file system hierarchy.

Figure 197. File System Hierarchy



Filenames

In theory, a filename can consist of any characters available on the keyboard. In practice however, many characters are of special significance to the MS-DOS or Windows NT operating systems and these should not be used in filenames and directory names.

Table 4 on page 530 gives the characters with special significance to the operating system.

Table 4. Characters With Special Significance

asterisk	*
question mark	?
less than	<
greater than	>
bar	
forward slash	/
backslash	\
quote	'
colon	:

As a general rule, it is best to restrict yourself to names consisting of letters, digits, and underscore characters.

 **Note**

You can use upper or lower-case letters for filenames, however, do not mix cases when entering commands. For example, File01 is valid for a filename, but Create_Directory is not valid for a command.

Identifying Files

This section provides information about identifying files to ensure the correct file is loaded, stored, and so on.

You can identify files three ways:

- By entering the file path name.
- By using the SEARCH command.
- By using the File Browser (see [Opening and Saving Drawings by Using Creo Elements/Direct Drafting File Browser on page 531](#) later in this chapter).
- By using the Navigator.

Using the File Path Name

The file path consists of the volume or drive identifier, a directory path, and a filename, all separated by a backslash ("\") or a colon (":").

For example, consider the following string:

'D:\larry\nvb_drw_101.mi'

The string can be identified as following:

Filename	nvb_drw_101.mi
Directory	larry
Volume/Drive:	D:

 **Note**

Creo Elements/Direct Drafting recognizes both the slash ("/") and the backslash ("\") as valid separators of the components of a file path. The following string also correctly identifies the file given in the example above.

'D:/larry/nvb_drw_101.mi'

In this guide, backslashes are used in general.

Opening and Saving Drawings by Using Creo Elements/Direct Drafting File Browser

You can open and save drawings quickly and easily by means of the Creo Elements/Direct Drafting File Browser. This File Browser adheres to standard MS-Windows conventions. You do not need to remember long filenames and subdirectories as these are displayed. (You can still type the filename in the user input line if you wish.)

You can close the File Browser once the drawing has been opened or saved. Optionally, you can leave the File Browser open for further selections.

In the **File** related dialog boxes, the File Browser provides the following icons:

	Moves up one level in the directory hierarchy.
	Creates a new folder (directory) on the current directory level.
	Displays files and directories in various views. You can choose the following views: <ul style="list-style-type: none"> • Thumbnails • Tiles • Icons • List • Details

In the dialog boxes associated with the Creo Elements/Direct Drafting File Browser, you can move, copy and delete files and directory using standard MS-Windows techniques.

Opening a Drawing File Using the Creo Elements/Direct Drafting File Browser

When you use  **Open**, the drawing file you select or specify replaces any drawing already loaded in the Creo Elements/Direct Drafting workspace.

The name of the currently loaded drawing file is displayed in the Creo Elements/Direct Drafting title bar (at the top of the Creo Elements/Direct Drafting window).

To open a drawing file by using the Creo Elements/Direct Drafting File Browser:

1. Click **File** ►  **Open** to display the File Browser.

Alternatively, click  **Open** on the Quick Access Toolbar or press CTRL+O.

2. If the file you want to open is not listed in the File pane of the File Browser, select the directory that contains the file in the **Directory** box.

If this required directory is specified in the File pane, double-click the directory name to list the files contained in it.

The system defaults to the most recent file type extension opened (startup default is *.mi). This is convenient if you open several files of the same type in your work session. Thumbnail views are an exception; if you open a Thumbnail, the extension of the file opened before the Thumbnail is kept as most recent.

3. Click the filename.

If the file that you want to open has a format different from the most recent opened, then in the **Filter** box, click the applicable file format:

- **Model Interface Standard (*.mi)**
- **Compressed MI (*.bi)** (compressed by means of the Z-lib mechanism)
- **Bundle (*.bdl)** (Creo Elements/Direct Drafting bundle files)

 **Note**

Only the 2D portions of such files are extracted, while any 3D portions are ignored.

- **Autodesk Drawing Interchange (*.dxf)**
- **Initial Graphics Exchange Format (*.igs)**
- **AutoCAD File Specification (*.dwg)**
- **JPEG - JFIF Compliant (*.jpg)**
- **Portable Network Graphics (*.png)**
- **Tagged Image File Format (*.tif)**
- **Windows or OS/2 Bitmap (*.bmp)**
- **All files (*.*)**

Refer to [Handling External File Formats on page 557](#) for further information on external file format translators used by Creo Elements/Direct Drafting.

A preview image of the drawing file is displayed in the preview pane of the File Browser.

4. Optionally, click the following:

Remember extension filter	Applicable to DXF and DWG files. A dialog box opens for DXF and DWG only.
Configure	Enabled for DXF or DWG files. The Import Configurator dialog box opens.
Check 2D	Creo Elements/Direct Drafting checks this file for invalid elements (such as extremely short lines) and will remove such elements.
Check 2D Retain Dependency	Creo Elements/Direct Drafting re-allocates dependent elements (such as dimensions or hatches) of removed invalid elements to plausible valid elements.

Overwrite fonts	In some cases, a font may have been saved with an MI file. Creo Elements/Direct Drafting will automatically load this font when you load the file. However, if the font in the file matches a font name already in your system, you can instruct Creo Elements/Direct Drafting to overwrite your system font with the file font by checking this box.
Discard fonts	In some cases, a font may have been saved with an MI file. Creo Elements/Direct Drafting will automatically load this font when you load the file. If you do not want new fonts loaded, check this box.

5. Click **Open** to open this file in Creo Elements/Direct Drafting.
Alternatively, type the filename in the **Name** box and press ENTER.
6. If Creo Elements/Direct Drafting contains a drawing, you are prompted to save this drawing before the selected file is loaded. Refer to [Saving a Drawing File via the Creo Elements/Direct Drafting File Browser on page 536](#) for instructions.
7. Once the drawing is displayed in Creo Elements/Direct Drafting, you can edit it as required.

Note

The preview image is pixel-based. It cannot be modified.

Opening a Drawing File by Means of the Navigator

The Creo Elements/Direct Drafting Navigator provides a scrollable sequence of preview images of the drawing files in the current directory. The Navigator thus facilitates the selection of drawings.

To open a drawing file by using the Creo Elements/Direct Drafting Navigator:

1. Click **File** ►  **Open** to display the File Browser.

Alternatively, click  **Open** on the Quick Access Toolbar or press CTRL+O.

2. Click **Navigator** to open the Creo Elements/Direct Drafting Navigator. By default, the **Navigator** points to the Creo Elements/Direct Drafting installation directory. The preview images in the right pane of the **Navigator** show a number of drawing frames supplied with Creo Elements/Direct Drafting.

-
- The **Navigator** displays a preview image for each file provided in the formats listed above.
3. In the left pane of the **Navigator**, click the directory whose drawing files you want to view.
 4. In the right pane of the **Navigator**, double-click the preview image to open its associated drawing file in Creo Elements/Direct Drafting.
 5. If Creo Elements/Direct Drafting contains a drawing, you are prompted to save this drawing before the selected file is loaded. Refer to [Saving a Drawing File via the Creo Elements/Direct Drafting File Browser on page 536](#) for instructions.
 6. Once the drawing is displayed in Creo Elements/Direct Drafting, edit the drawing as required.

 **Note**

To return to the File Browser, close the Navigator.

Opening a Drawing File Using Configure

Configure allows you to open DXF or DWG files.

To open a drawing file using **Configure**:

1. Click **File** ►  **Open** to display the File Browser.

Alternatively, click  **Open** on the Quick Access Toolbar or press CTRL+O.
2. Browse to and select the DWG or DXF file.
3. Click **Configure** to open the **Import Configurator** dialog box.

The **Import Configurator** dialog box has two tabs; **Advanced** and **Configure**.
4. Edit the options on **Advanced** and **Configure** tabs.

See [Preparing the Drawing Environment on page 61](#) for details about the **Advanced** and **Configure** tabs.
5. Click **Open** in the File Browser to load the drawing.

 **Note**

On the **Advanced** tab of the **Import Configurator** dialog box, options such as **Read all layout**, **Model space**, and **Translate paper space** are not available for existing files. The **Advanced** tab also displays information in the file such as layout names, font, line types, and unit.

Opening Most Recently Used Drawing Files

By default, the four most recently used drawing files are listed (complete with filetype extension) in the Creo Elements/Direct Drafting **File** menu. To load one of these files:

To load a recently used drawing file:

1. Click **File**.
2. Under **Recent Documents**, click the name of the drawing file.

Saving a Drawing File via the Creo Elements/Direct Drafting File Browser

To save a drawing by using the Creo Elements/Direct Drafting File Browser:

1. In Creo Elements/Direct Drafting, click **File** ►  **Save** to display the File Browser.

Alternatively, click  **Save** on the Quick Access Toolbar or press CTRL+S.

2. If the filename that you want to use for saving the file is not listed in the File pane of the File Browser, select the directory that contains the file in the **Directory** box.

If the required directory is specified in the File pane, double-click the directory to list the files contained in it.

The system defaults to the most recent file type extension saved (startup default is `.mi`). This is convenient if you save several files of the same type in your work session. Thumbnail views are an exception. If you open a thumbnail, the extension of the file saved before the thumbnail is kept as most recent.

3. Click an existing filename (to overwrite an existing file) or type a new filename (to create a new drawing file) in the **Name** box.

4. If the file (containing the drawing that you want to save) is in a format different from the most recent saved, then in the **Filter** box, click the applicable format:

- **Model Interface Standard (*.mi)**
- **Model Interface Standard (*)** (save MI format files without an extension or with a custom extension)
- **Compressed MI (*.bi)** (compressed by means of the Z-lib mechanism)
- **Initial Graphics Exchange Format (*.igs)**
- **Virtual Reality Modeling Language (*.wrl)**
- **Drawing Web Format (*.dwf)**
- **Scalable Vector Graphics (*.svg)**
- **Text file (*.txt)**
- **Drafting [version] – MI [version] (*.mi)** (MI version 3.40, 3.30, 3.20, and so on)
- **ME 10 [version] – MI [version] (*.mi)**
- **AutoCAD [version] DXF (*.dxf)** (version 2013, 2010, 2007, and so on)
- **AutoCAD [version] Drawing (*.dwg)** (version 2013, 2010, 2007, and so on)

Refer to [Handling External File Formats on page 557](#) for further information on external file format translators used by Creo Elements/Direct Drafting.

5. Click **Add fonts** if you want to ensure that the fonts you used with this drawing is visible for others. This adds to the file size, but ensures that others who do not have the same fonts on their systems can read your drawing text (available for MI format only).
6. Click **Save** to save the drawing file.

Once you have completed this procedure, you can save the current drawing in the same file by simply clicking **Save** in the File Browser.

Saving a Drawing File under a Different Name

To save a drawing under a different name:

1. In Creo Elements/Direct Drafting, click **File**  **Save As**.
2. If the filename that you want to use for saving the file is not listed in the File pane of the File Browser, select the directory that contains the file in the **Directory** box.

If the required directory is specified in the File pane, double-click the directory to list the files contained in it.

-
3. In the **Name** box, type a name for the file.
 4. If you want to save the drawing in a format other than the most recently saved, then in the **Filter** box, select the applicable file format.

Note that for DWF, VRML, and SVG, the file filter will be set back to the previous choice. Refer to [Saving a Drawing File via the Creo Elements/Direct Drafting File Browser on page 536](#) for a list of supported formats.

5. Click **Save** to save the drawing file.

Saving a Drawing File in HPGL formats

To save an entire drawing, or the content of a viewport, or the content of a box in a HPGL format:

1. Click **File** ►  **Print** ►  **Print Setup**.

Alternatively, click  **Print Preview and Print** on the Quick Access Toolbar.

The **Print** dialog box opens.

2. On the **Print** tab, in the **Printer** area, select a generic option in the **Name** box from the following:
 - **HPGL_GENERIC** - Hewlett-Packard plotter language
 - **HPGL2_GENERIC** - Improved version with text defined
 - **HPGL2_RTL_GENERIC** - Improved version with text and images

Note

These formats are based on generic printer drivers. For further information on setting up a print configuration, refer to [Creating a Print Configuration on page 607](#).

Note the boxed plus sign beside the generic options. Expand the plus sign and select the model number, if required.

3. In the **Destination** box, type the destination file path and file name.
4. Click **Print**.

Saving a Drawing File in PDF format

To save an entire drawing, the content of a viewport, or the content of a box in a generic PDF format:

-
1. Click **File** ►  **Print** ►  **Print Setup**.

Alternatively, click  **Print Preview and Print** on the Quick Access toolbar.

The **Print** dialog box opens.

2. On the **Print** tab, in the **Printer** area, select **PDF_GENERIC** in the **Name** box.
3. In the **Destination** box, type the destination file path and file name.
Alternatively, click **Browse** to select the destination path and file name.
4. Click **Print**.

For further information on setting up a print configuration, refer to [Creating a Print Configuration on page 607](#).

Saving Selected Elements

To save a selection of elements in your drawing:

1. Type `STORE` in the user input line and press `ENTER`.
2. Select the element or elements you want to save.
3. Type a name for the file (in apostrophes) in the user input line for saving the selection.
4. Press `ENTER`.

Creating a New Drawing

To create a new drawing:

1. Click **File** ►  **New Session**

Alternatively, click  **Window** on the Quick Access Toolbar or press `CTRL+N`.

2. If Creo Elements/Direct Drafting already contains a current drawing, you are prompted to save the drawing in a file before a new drawing is initiated.
Proceed as described in [Saving a Drawing File under a Different Name on page 537](#).
3. Create the drawing in the usual way.

Creating a New Creo Elements/Direct Drafting Instance

Optionally, you can run multiple instances of Creo Elements/Direct Drafting at the same time.

To create a new Creo Elements/Direct Drafting instance:

- Click **Miscellaneous** and then, in the **System** group, click  **New Drafting**.
Alternatively, click **Start** ► **All Programs** ► **PTC** ► **Creo Elements Direct Drafting [version]** ► **Creo Elements Direct Drafting [version]**.

Note

- Each instance will run in a separate window.
 - All Creo Elements/Direct Drafting instances are independent of each other.
 - Only a single Creo Elements/Direct Drafting license is required for any number of Creo Elements/Direct Drafting instances run by the same user on the same machine/display.
-

Opening an Existing Drawing File in a New Creo Elements/Direct Drafting Instance

To open an existing drawing file in a new Creo Elements/Direct Drafting instance:

1. Click **File** ►  **Open Dwg in New Drafting**.
The **Open** dialog box opens.
2. Select the file.
If the file is not listed, select the directory which contains the file in the **Directory** box and then select the required file.
3. Click **Open**.

Importing Macros, Fonts, Pictures, and HPGL Files

You can import the following files into your current Creo Elements/Direct Drafting drawing:

- Parts and Subparts
- Creo Elements/Direct Drafting Macros
- Fonts
- Pictures
- HPGL Files

Importing a Part

When you use  **Import Part**, the drawing file you select or specify is added to any drawing already loaded in the Creo Elements/Direct Drafting workspace. In the case of an assembly drawing, a part is loaded below the top part. This means that there is no need to clear the memory as the top part has now been renamed, for example, to: EXAMPLE_ASSY.

To load a part:

1. Click **File** ► **Import/Export** ►  **Import Part**.
The **Load File** dialog box opens.
2. If the file that you want to import is not listed in the File pane of the File Browser:
 - a. In the **Directory** box, select the directory that contains the drawing file that you want to import.
 - b. Double-click the directory.
If the directory is specified in the File pane, double-click the directory name to list the files contained in it.
3. Click or specify the filename.
4. If a file is provided in a format other than MI (the default), then in the **Filter** box, select the applicable file format. Refer to [Opening a Drawing File Using the Creo Elements/Direct Drafting File Browser on page 532](#) for a list of supported formats.

Optionally, you can click any or all of the following check box:

As Subpart	Loads a drawing as a subpart of the active part.
Absolute	Loads the drawing back to the same position,

	relative to the origin, from which it was originally stored.
Keep Scale	The original drawing scale is retained when the drawing file is loaded.
Check 2D	Creo Elements/Direct Drafting checks this file for invalid elements (such as extremely short lines) and removes such elements.
Check 2D Retain Dependency	Creo Elements/Direct Drafting reallocates dependent elements (such as dimensions or hatches) of removed invalid elements to plausible valid elements.

5. Click **Open** to add the file as a part to the current drawing in Creo Elements/Direct Drafting.

Creo Elements/Direct Drafting will automatically load the part into the position, relative to the origin, in which it was originally created.

Note

Creo Elements/Direct Drafting loads the file immediately below the top part.

Importing a Subpart

You can import a part file as a subpart under the active part.

To import a subpart:

1. Click **File** ► **Import/Export** ►  **Import Part**.
The **Load File** dialog box opens.
2. If the file that you want to import is not listed in the File pane of the File Browser:
 - a. In the **Directory** box, select the directory that contains the drawing file that you want to import.
 - b. Double-click the directory.

If the directory is specified in the File pane, double-click the directory name to list the files contained in it.
3. Click the **As Subpart** check box.
4. Click or specify the filename.

-
5. If a file is provided in a format other than MI (the default), then in the **Filter** box, select the applicable file format. Refer to [Opening a Drawing File Using the Creo Elements/Direct Drafting File Browser on page 532](#) for a list of supported formats.
 6. Click **Open** to add the file as a subpart to the current drawing in Creo Elements/Direct Drafting.

Creo Elements/Direct Drafting automatically loads the part into the position, relative to the origin, in which it was originally created.

 **Note**

Creo Elements/Direct Drafting loads the file as a part. The part belongs to the active part and is placed under it in the parts hierarchy.

Importing a Creo Elements/Direct Drafting Macro

To import a Creo Elements/Direct Drafting macro:

1. Click **File** ► **Import/Export** ►  **Import Macro**.
2. Click **Macro**.
3. Click one of the following:
 - Click **Input Macro** to import a macro provided in the ASCII format.
The **Input Macro** dialog box opens.
 - Click **Load Macro** to import a macro provided in the binary format.
The **Load Macro** dialog box opens.
4. Select the macro from the dialog box.
5. Click **Open** to open the macro in Creo Elements/Direct Drafting.

Importing a Font

To import a font provided in the binary format:

1. Click **File** ► **Import/Export** ►  **Import Font**.
2. Click **Font**.
3. Click **Load Font** to import a font provided in the binary format.
The **Load Font** dialog box opens.

-
4. Select a font and then click **Open** to load the font for use in Creo Elements/Direct Drafting.

To import a TrueType font provided in the ASCII format:

1. Click **File** ► **Import/Export** ►  **Import Font**.
2. Click **Font**.
3. Click **Convert From TT/T1** to import a TrueType font provided in the ASCII format.

The **Convert Font** dialog box opens.

4. In the **Name** box, select a font or fonts. Optionally, click the **All** check box to select all fonts.
5. Click **Convert** to convert and load the font for use in Creo Elements/Direct Drafting.

Specifying the Resolution of an Imported TrueType Font

To specify the resolution for a TrueType font that you want to convert to a Creo Elements/Direct Drafting font:

1. Type `CONVERT_TO_ME10_FONT` in the user input line and press ENTER.
2. Type the desired resolution in the user input line and press ENTER. Note that the maximum value is 255.
3. Type the font name in quotes and press ENTER.

Importing a Picture File

Creo Elements/Direct Drafting lets you import picture files in TIFF (*.tif, compressed and uncompressed), BMP (*.bmp), JPEG (*.jpg) and PNG (*.png) formats. The following TIFF compression methods are supported:

- Packbits
- CCITT Group 3 1-Dimensional (modified Huffman encoding)
- CCITT Group 4

Being able to import images is particularly useful when you want to place a scanned image onto your drawing or include a 3D model for better visualization.

To import a picture:

1. Click **File** ► **Import/Export** ►  **Import Picture**.

The **Import Raster Image** dialog box opens.

2. Select an image.

-
3. Click **Open** to open the image file in Creo Elements/Direct Drafting.
 4. Type a name for the picture in the user input line and press ENTER.
 5. Specify the position for the imported image file.

Refer to [Editing Imported Pictures on page 577](#) for further details on handling imported TIFF files in Creo Elements/Direct Drafting.

Importing an HPGL File

Creo Elements/Direct Drafting enables you to load HP-GL and HP-GL2 files. To import a HP-GL (Hewlett-Packard Graphics Language) or HP-GL2 file:

1. Click **File** ► **Import/Export** ►  **Import HPGL**.

The **Load HPGL/2 file** dialog box opens.

2. Select the HPGL/2 file.
3. Click **Open** to open the file in Creo Elements/Direct Drafting.
4. Specify the position for the imported HPGL/2 file.

Exporting Macros, Fonts, and Pictures

You can export the following from your current Creo Elements/Direct Drafting session:

- Parts
- Creo Elements/Direct Drafting Macros
- Fonts
- Pictures

Exporting a Part

To export the active (current) part in the current Creo Elements/Direct Drafting drawing:

1. Click **File** ► **Import/Export** ►  **Export Part**.

The **Store Part** dialog box opens.

2. If the filename that you want to use for saving the file is not listed in the File pane of the File Browser:
 - a. In the **Directory** box, select the directory for saving the file.
 - b. Double-click the directory.

-
3. In the **Name** box, type the name of the file in which you want to save the current part.
 4. If you want to save the part in a format other than MI (the default), then in the **Filter** box, select the applicable format. Refer to [Saving a Drawing File via the Creo Elements/Direct Drafting File Browser on page 536](#) for a list of supported formats.
 5. Click **Save** to save the part in the specified drawing file.

Exporting a Creo Elements/Direct Drafting Macro

To export a Creo Elements/Direct Drafting macro:

1. Click **File** ► **Import/Export** ►  **Export Macro**.
2. Click one of the following:
 - Click **Save Macro** to export and save a macro in the ASCII format.
 - Click **Store Macro** to export and save a macro in the binary format.

The **Select Macro** dialog box opens.

3. In the **Name** box, select a macro.
Optionally, click the **All** check box to select all macros.
4. Click **OK**.
The **Save Macro** dialog box opens.
5. In the **Name** box, type or select a macro file name (* .m).
6. Click **Save** to save the macro file.

Exporting a Font

When you have edited a font (or any character or characters included in a given font), you need to export this font for further use.

To export a font:

1. Click **File** ► **Import/Export** ►  **Export Font** .
2. Click one of the following:
 - Click **Save Font** to export and save a font in the ASCII format.
 - Click **Store Font** to export and save a font in the binary format.

The **Select Font** dialog box opens.

3. In the **Name** box, select a font.

Optionally, click the **All** check box to select all fonts.

4. Click **OK**.

The **Save Font** dialog box opens.

5. In the **Name** box, type or select a font file name (*.fnt).
6. Click **Save** to save the font file.

Exporting a Picture

Creo Elements/Direct Drafting allows you to export pictures in various formats.

To export your picture:

1. Click **File** ► **Import/Export** ►  **Export Picture**.
2. Select a single picture for export.
3. Type the name of the destination file with the desired file format (JPG, PNG, TIF, or PNG) and press ENTER.

Exporting as Image

Creo Elements/Direct Drafting allows you to export a drawing as an image in high resolution pixel format.

To export a drawing as an image:

1. Click **File** ► **Import/Export** ►  **Export as Image**.
2. Click one of the following:
 - **Export as BMP**
 - **Export as JPEG**
 - **Export as PNG**
 - **Export as TIFF**
3. To specify input, type `PORT`, `BOX`, or `DEL_OLD` in the user input line and press ENTER.

Alternatively, type the name and the destination path for the drawing image file in the user input line and press ENTER to directly obtain the drawing image at the specified destination path.
4. If you type `PORT` in the user input line and press ENTER:
 - a. Click a point, or type `CURRENT` or the viewport number in the user input line and press ENTER.

-
- b. Type `RESOLUTION` or destination for the drawing image file in the user input line and press `ENTER`.

To specify the resolution for the drawing image, type `RESOLUTION` in the user input line and press `ENTER`. Then perform the following steps:

- Type `PAPER_TO_PIXELS` in the user input line and press `ENTER`.
Creo Elements/Direct Drafting recalculates paper format and dpi to X, Y resolution. You are then prompted for a paper format (A-E, A4-A0) or size, and then for the dpi.
 - Type the width and height of the pixels. Type `-1` for either width or height to retain the ratio of the current viewport and specify the other dimension in pixels.
5. If you type `BOX` in the user input line and press `ENTER`:
 - a. Enclose the required drawing area in a box and press `ENTER`.
 - b. Type the name and the destination path for the drawing image file in the user input line and press `ENTER`.

Possible error messages are:

- Not enough memory available to perform the operation.
- ***The output image size is limited to a maximum of 4/3 billion pixels.

Copying Files and Directories

You can use Creo Elements/Direct Drafting to copy files and directories. You can select files, directories, and subdirectories by using the **Open** dialog box or by specifying their names in the user input line.

Copying Files

To copy a file:

1. Click **Miscellaneous** and then, in the **System** group, click  **Copy**.
2. Type the filename (between single quotes) in the user input line and press `ENTER`.

You can use the following four wildcard characters in the file name:

- `*`
- `[...]`
- `?`
- `\`

-
3. Type one of the following in the user input line:
 - `SELECT` and then press `ENTER` to select the specified file. You can further narrow the selection using catalog sorting and one of the following relational operators:
 - `=`
 - `<>`
 - `>`
 - `<`
 - `>=`
 - `<=`
 - `DEL_OLD` and then press `ENTER` to delete the earlier version of the specified file.
 - `SCREEN` and then press `ENTER` to view the source file name in a standard text editor.
 4. Type the destination path for the file in the user input line and press `ENTER`.

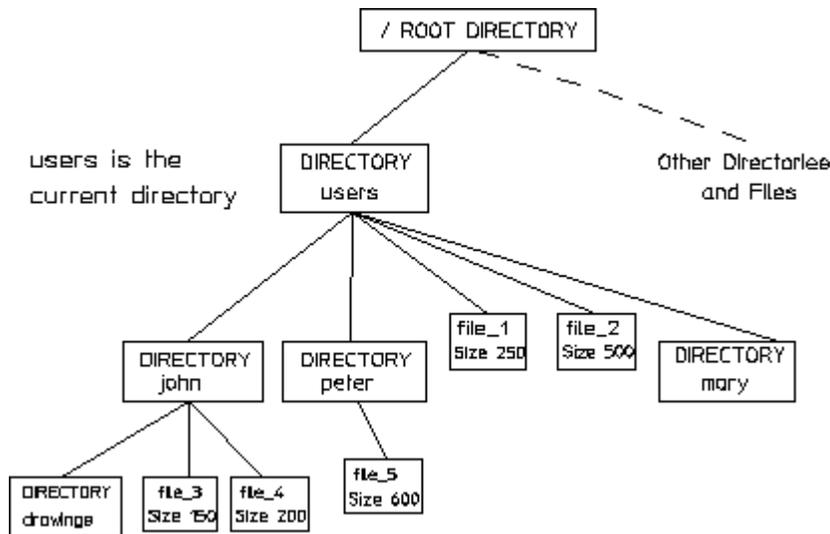
Copying a File Using the Selection Mechanism

The following example shows how to:

- Find files in the current directory (`users`) with a filename beginning with `f`.
- Select files greater than 250 bytes in size.
- Copy those files to the `mary` directory.

The following figure shows the file structure before using  **Copy**:

File Structure Before Copy

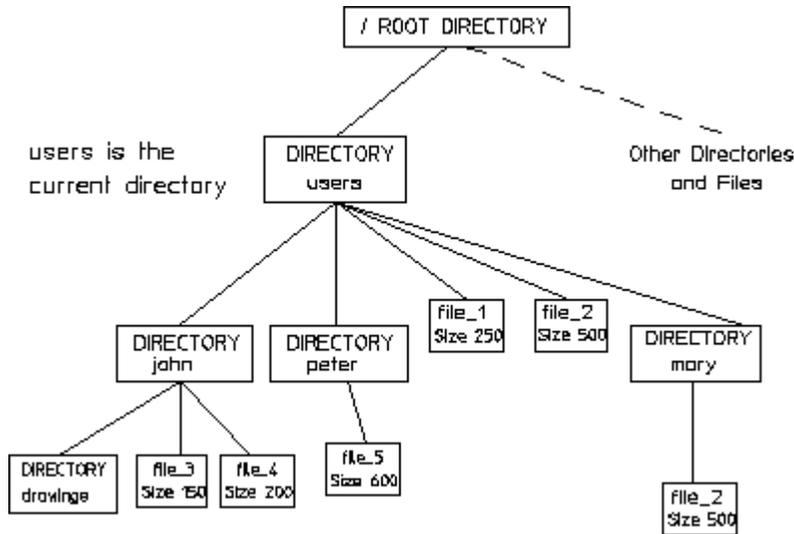


1. Click **Miscellaneous** and then, in the **System** group, click  **Copy**.
2. Type 'f*' in the user input line and press ENTER.
Creo Elements/Direct Drafting identifies the file_1 and file_2 files.
3. Type SELECT in the user input line and press ENTER.
4. Type 'SORT FILE_SIZE' in the user input line and press ENTER.
5. Type '>250' in the user input line and press ENTER.
Creo Elements/Direct Drafting searches for files that are larger than 250 bytes and identifies the file_2 file.
6. Type mary in the user input line and press ENTER.

Creo Elements/Direct Drafting copies the file_2 file to the mary directory.

The following figure shows the file structure after using  **Copy**:

File Structure After Copy



Copying Directories

You must use the `RECURSIVE` option to copy directories.

To copy directories you must specify the name of the directory as the source name (include a directory path if required). The entire directory with its files and subdirectories will then be copied to the directory specified as the destination file.

Note

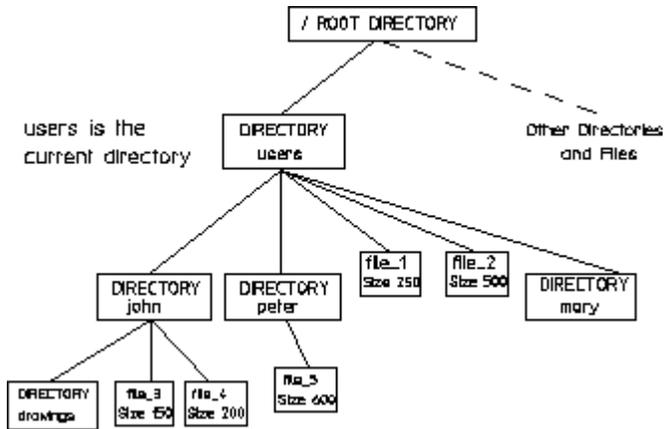
- While using  **COPY**, you can copy directories using the `RECURSIVE` option only.

Copying a Directory Using Recursive

The following example shows how to copy the `/users/peter` directory to the `/john/drawings` directory.

The following figure shows the directory structure before using  **Copy**:

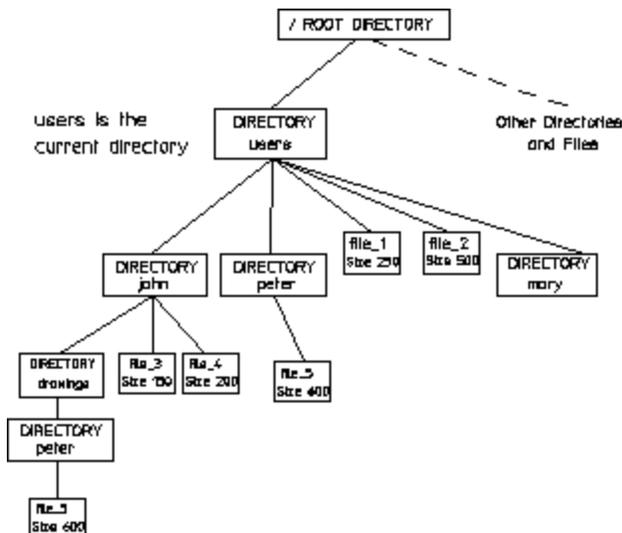
Directory Structure Before Copy



1. Click **Miscellaneous** and then, in the **System** group, click  **Copy**.
2. Type `/users/peter` in the user input line and press ENTER.
Creo Elements/Direct Drafting identifies the `peter` directory.
3. Type `RECURSIVE` in the user input line and press ENTER.
4. Type `/john/drawings` in the user input line and press ENTER.
Creo Elements/Direct Drafting copies the `peter` directory to the `drawings` directory.

The following figure shows the directory structure after using  **Copy**:

Directory Structure After Copy



Copying Options

When you are copying files or directories, you can specify the following options in the user input line before specifying the destination filename or directory name:

- Use `Delete Old` to overwrite an existing file.
- Use `Screen` to view the files and directories to be copied in the standard text editor.

Note

- If you have specified more than one file as the source name, you must specify a directory as the destination name. Else, the command does not operate.
 - You cannot specify more than one file or directory as the destination name.
 - You cannot copy device files, block special files, or char special files.
-

Purging Files or Directories

You can use Creo Elements/Direct Drafting to purge files or directories.

To purge a file or directory:

1. Click **Miscellaneous** and then, in the **System** group, click  **Purge**.
2. Type the name of the file or directory (between single quotes) in the user input line and press ENTER.

You can use the following wildcard characters in the directory path:

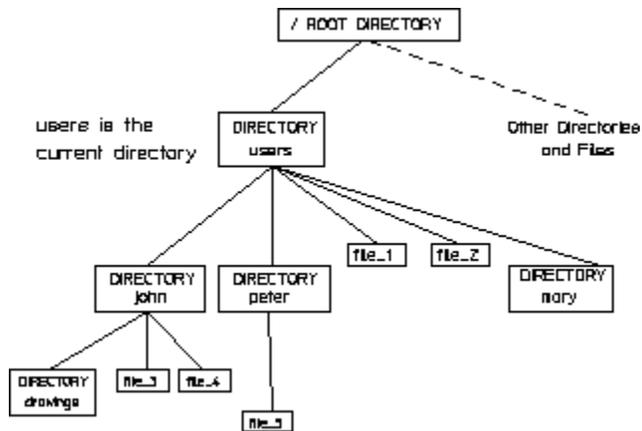
- *
 - [...]
 - ?
 - \
3. Type one of the following in the user input line and press ENTER:
 - `SCREEN` to view the files or directories for deletion in a standard text editor.
 - `SELECT` to use the catalog selection mechanism on the specified files or directories.
 - `CONFIRM` to directly purge the file.

Purging Examples

For the purpose of the following examples, assume the current directory is /users.

The following figure shows the file structure before purging:

File Structure Before Purging



Example 1

To purge the file_1 and file_2 files:

1. Click **Miscellaneous** and then, in the **System** group, click  **Purge**.
2. Type ' * ' in the user input line and press ENTER.

Creo Elements/Direct Drafting identifies the john and peter directories, the file_1 and file_2 files, and the mary directory.

3. Type SELECT in the user input line and press ENTER.
4. Type "FILE_TYPE" in the user input line and press ENTER.
5. Type < > 'DIR' in the user input line and press ENTER.

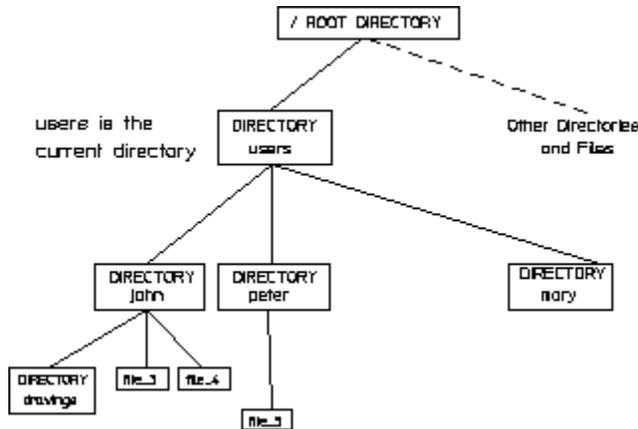
Creo Elements/Direct Drafting sorts through the three directories and two files and identifies the file_1 and file_2 files.

6. Click  **Confirm**.

The file_1 and file_2 files are purged.

The following figure shows the file structure after purging:

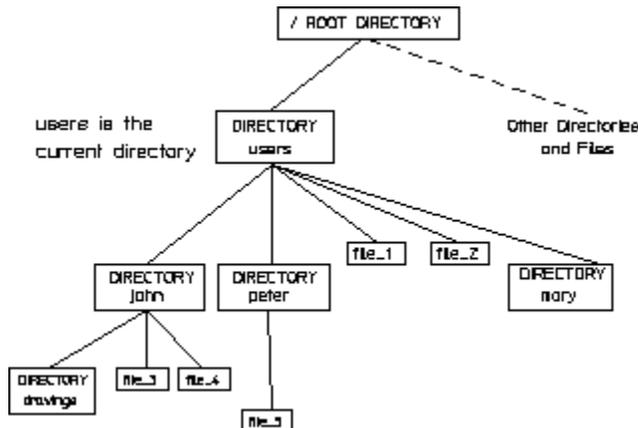
File Structure After Purging



Example 2

The following figure shows the directory structure before purging:

Directory Structure Before Purging



To purge the john, peter and mary directories:

1. Click **Miscellaneous** and then, in the **System** group, click  **Purge**.
2. Type ' * ' in the user input line and press ENTER.
Creo Elements/Direct Drafting identifies the john and peter directories, the file_1 and file_2 files, and finally the mary directory.
3. Type SELECT in the user input line and press ENTER.
4. Type 'FILE_TYPE' in the user input line and press ENTER.
5. Type 'DIR' in the user input line and press ENTER.

Creo Elements/Direct Drafting now sorts through the three directories and two files and identifies the `john`, `peter` and `mary` directories.

In the last case (example 2) nothing is purged. Creo Elements/Direct Drafting tries to purge the `john` directory. However, since this directory is not empty, an error message is displayed and the operation is terminated. If however, the empty `mary` directory had been the first directory to be found by Creo Elements/Direct Drafting, it would have been purged and Creo Elements/Direct Drafting would then have given an error message, since it would have then found either the `john` or `peter` directory, neither of which are empty.

To successfully delete `john`, `peter` and `mary` directories, you must use the `RECURSIVE` option.

Using the `RECURSIVE` option, you can purge files, non-empty directories and empty directories. The `RECURSIVE` option operates on the whole directory path tree of the directory you have already specified.

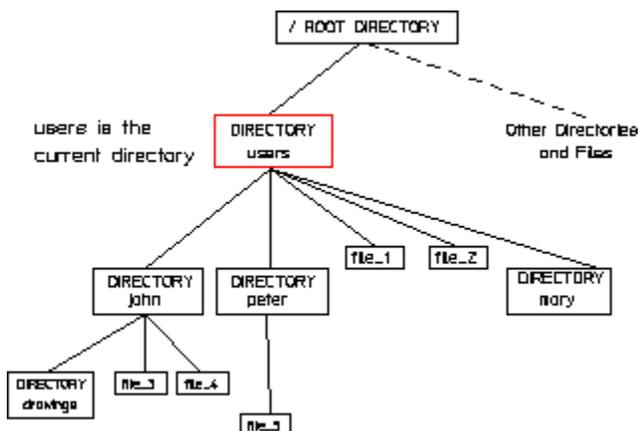
 **Note**

`RECURSIVE` is the only option that allows you to purge directories.

Example 3

The following example shows how to delete the `john` directory using the `RECURSIVE` option. The following figure shows the `john` directory before purging:

Directory Structure Before Purging



1. Click **Miscellaneous** and then, in the **System** group, click  **Purge**.
2. Type '`john`' in the user input line and press ENTER.

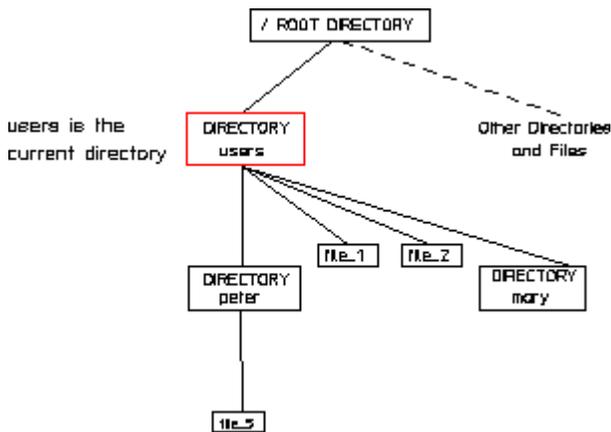
-
- Creo Elements/Direct Drafting identifies the john directory.
 - Type RECURSIVE in the user input line and press ENTER.
 - Type SCREEN in the user input line and press ENTER.

The contents of the john directory are displayed in a standard text editor. The john directory and its contents are deleted after you exit the standard text editor.

- To delete the john directory without viewing the directory contents, type CONFIRM in the user input line and press ENTER.

The following figure shows the directory structure after purging:

Directory Structure After Purging



Handling External File Formats

Creo Elements/Direct Drafting provides the following translators for external file formats:

- DWF Translator
- DXF DWG Translator
- IGES Translator
- VRML Translator

To run one of these translators, double-click the associated icon in the Creo Elements/Direct Drafting program group. Each of these translators is controlled via a DOS window.

Further Information

For further technical information about loading files, refer to the Creo Elements/Direct Drafting help system.

Type the following in the user input line and press ENTER:

- `help load`
- `help load_font`
- `help load_hpgl`
- `help load_macro`
- `help store`
- `help store_330`

Creo Elements/Direct Drafting displays further information about loading files.

18

Revision Checking

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This chapter describes how to use the buttons and options that allow you to compare similar drawings. This facility is particularly useful in design offices where successive drawings are revised and you need to identify the differences between revisions. For example, suppose you create a drawing of a plate early on in the development stage. A few weeks later you revise this drawing by stretching it, adding fillets, and adding holes. You can now compare the original drawing with the revised drawing to see where changes have been made. The differences between the original and revised drawings are shown in color.

Note

Revision checking is possible on drawings stored in MI-format 2.10 or higher. Incorrect results may occur if you attempt to compare drawings saved in MI-formats earlier than 2.10. For OLE objects, the revision checking facility compares the name of the native application, the class ID and the type of the OLE object.

Figure 198. Original Drawing and Revised Drawing on page 560 shows the original drawing (A) and the revised drawing (B). Figure 199. Both Files Loaded on page 560 shows A and B superimposed so that you can easily identify the modified elements.

Figure 198. Original Drawing and Revised Drawing

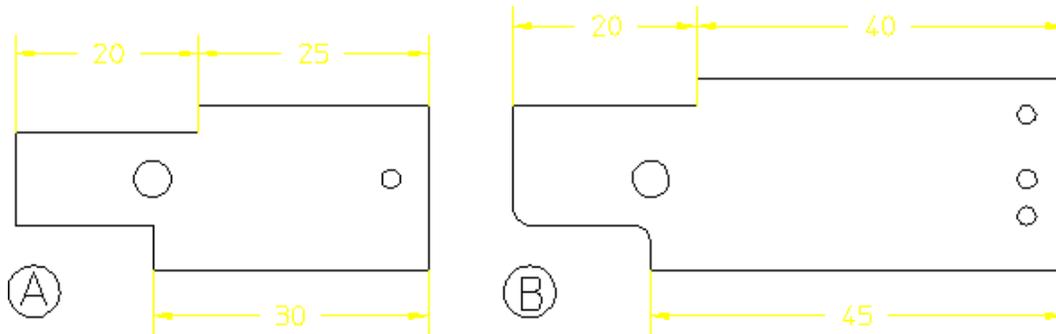
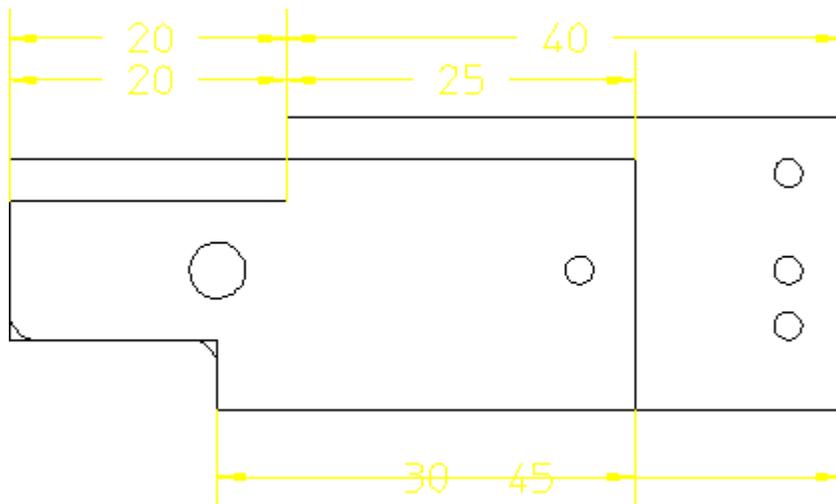


Figure 199. Both Files Loaded



Using Revision Checking

To use the revision checking facility:

1. Click **Compare** and then, in the **Setup** group, click  **Load Files**.

The **Load** dialog box opens.

2. In the **Original** box, type the name of the original drawing file.

For example, dt_cs_rev06.

Alternatively, click **Browse** to select the original drawing file.

3. In the **Revised** box, type the name of the revised drawing file.

For example, dt_cs_rev10.

Alternatively, click **Browse** to select to select the revised drawing file.

4. Click **OK**.

Creo Elements/Direct Drafting displays the two drawings overlapping.

5. Click **Compare** and then, in the **Compare** group, click  **Geometry**.

Creo Elements/Direct Drafting compares the location of the geometry and texts (without comparing the parts structure) in the two drawings and shows the result of the comparison in different colors according to the following defaults:

Blue	Identical geometry elements in both drawings
Green	Differences in the original drawing
Red	Differences in the modified drawing

6. Click **Compare** and then, in the **Compare** group, click  **Annotation**.

In addition to  **Geometry**,  **Annotation** also compares the properties of the geometry and texts — linetype, color, text attributes, info texts — and the dimensions and hatches in both drawings.

7. Click **Compare** and then, in the **Compare** group, click  **Parts**.

In addition to  **Annotation**,  **Parts** compares the parts structure in both drawings.

Note

If you modify a drawing solely by a scaling factor, clicking  **Parts** displays all elements in red (indicating a difference). However,  **Geometry** and  **Annotation** display elements and annotation in blue (indicating identical elements).

Note

Both drawings must have the same location compared to the drawing origin to get meaningful results from the comparison.

Be careful when working with scaling factors (Part Scale).

Viewing Single Drawings

When both drawings are loaded, you can toggle between them using the buttons in the **Show** group on the **Compare** tab.

To view the original drawing only:

- Click **Compare** and then, in the **Show** group, click  **Original**.

To view the revised drawing only:

- Click **Compare** and then, in the **Show** group, click  **Revised**.

To view both the drawings:

- Click **Compare** and then, in the **Show** group, click  **Both**.

Viewing and Logging Differences

When both the drawings are loaded and compared using the buttons in the **Compare** group, you can view the differences between them. You can also view the generated logs of both the drawing files in a standard text file.

To view the identical elements in the drawing:

1. Click **Miscellaneous** and then, in the **Rev Check** group, click  **Log Diff**.
2. Click **Identical**.

To view the original drawing:

1. Click **Miscellaneous** and then, in the **Rev Check** group, click  **Log Diff**.
2. Click **Drawing 1**.

Creo Elements/Direct Drafting shows the original drawing and generates a log of the original drawing in a standard text file.

To view the differences in the modified drawing:

1. Click **Miscellaneous** and then, in the **Rev Check** group, click  **Log Diff**.
2. Click **Drawing 2**.

Creo Elements/Direct Drafting shows the differences in the modified drawing and generates a log of the modified drawing in a standard text file.

Note

If you click the **Arrow** check box in the **Rev Check** group on the **Miscellaneous** tab, Creo Elements/Direct Drafting indicates the differences in the drawing with arrows.

Continuing with the Drawing

When you are ready to continue with your drawing:

- Click **Compare** and then, in the **Continue** group, click  **Original** or  **Revised**.

Creo Elements/Direct Drafting displays the selected drawing and deletes the other drawing. You can now continue working on this drawing in the usual manner.

Unlike the **Show** group buttons, the **Continue** group buttons do not let you toggle between the two drawings as it deletes the one you do not select.

Resetting Colors and Showing Differences

The colors generated after selecting a button in the **Compare** group can be reset to their original colors.

To view the drawings in the original colors:

- Click **Compare** and then, in the **Show** group, click  **Reset**.

To view the drawing elements in different colors depending on the result of the last comparison:

- Click **Compare** and then, in the **Show** group, click  **Differences**.

Changing the Color Scheme

To change the color scheme for changes in the original drawing:

1. Click **Compare** and then, in the **Setup** group, click the arrow next to  **Original**.
2. Click a color.

To change the color scheme for changes in the revised drawing:

1. Click **Compare** and then, in the **Setup** group, click the arrow next to  **Revised**.
2. Click a color.

To change the color scheme for identical elements:

1. Click **Compare** and then, in the **Setup** group, click the arrow next to  **Identical**.
2. Click a color.

Refer to [Defining Custom Colors on page 100](#) for instructions on defining custom colors.

The color changes take effect with the next redraw.

Further Information

For further technical information about revision checking, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- `help wui_rc_load_files`
- `help rc_view`
- `help rc_show_diff`
- `help rc_clean_up`
- `help wui_rc_color_settings`

Creo Elements/Direct Drafting displays further information about revision checking.

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Associative Documentation Update

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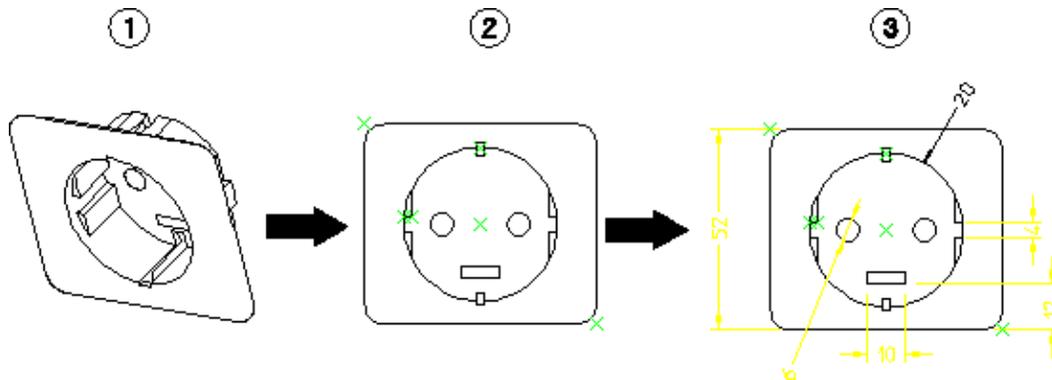
This chapter tells you how to use the Associative Documentation Update (ADU) functionality.

ADU provides a way of reflecting 3D design modifications from Creo Elements/Direct Modeling in related 2D technical drawings (layouts) in Creo Elements/Direct Drafting.

Elements of existing technical drawings (text, symbols, and hatches) are automatically transferred to the new 2D technical drawing after a change in a 3D design. 2D dimensions associated to 3D geometry are updated automatically. Geometry and annotation changes are highlighted in user-definable colors.

The following figure shows a 3D design (1) done with Creo Elements/Direct Modeling and its 2D layout (2) in Creo Elements/Direct Drafting. Modifications made to the 3D design are highlighted in color in the 2D layout. (Drawing 2 is the undocumented layout and drawing 3 is the documented layout.)

3D Model and 2D Layout



When modifications are made to a 3D drawing, ADU remembers identifiers and tries to relocate them in the new layout. Elements with identifiers that are found again in the new layout are shown in green and identifiers that are not found are shown in red at their original position.

Using Associative Documentation

To work with associative documentation in Creo Elements/Direct Drafting, you need to activate the ADU tab.

To activate the ADU tab:

1. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.

2. In the **Customize the Ribbon** list, click the **ADU** check box.

The **ADU** tab is displayed on the ribbon.

Loading Drawings

Before you begin, you must have two versions of a drawing available that were created by Creo Elements/Direct Modeling and passed to Creo Elements/Direct Drafting as 2D layouts. For example:

`layout_1`: Drawing that belongs to the original 3D model and that was completely documented in Creo Elements/Direct Drafting in an earlier session.

`layout_2`: Layout that belongs to the modified 3D model and that is undocumented.

1. Click **ADU** and then, in the **Setup** group, click  **Load & Update**.

The **Load** dialog box opens.

2. In the **Drawing** box, type the name of the drawing file (for example, `layout_1`).

Alternatively, click **Browse** to select the drawing file.

3. In the **Layout** box, type the name of the layout file (for example, `layout_2`).

Alternatively, click **Browse** to select the layout file.

4. Click **OK**.

Creo Elements/Direct Drafting updates `layout_2` with the 2D documentation that had been added to `layout_1`. The result of the ADU update is shown in colors according to the following color defaults:

- Annotations

Green	Transferred 2D	All elements of the existing technical drawing as texts, symbols and drawing frames without a reference to geometry coming from 3D. They are transferred to the new 2D technical drawing that is the result of the Associative Documentation Update (ADU). Additionally, all dimensions and hatches associated to geometry that comes from 3D and that was not changed during the 3D modification are transferred.
Blue	Updated 2D	All dimensions and hatches associated to geometry that was modified during the 3D modification but that could be unambiguously determined in the layout file. For example, if the radius of a bore-hole is changed in the 3D model with a local modification then the radius dimension associated to the bore-hole in the existing technical drawing will be updated to the new bore-hole as the modified bore-hole could be unambiguously determined in the layout file.
Red	Regenerated 2D	This means dimensions and hatches that referred to geometry coming from the old drawing that is no longer in the new layout. For example, if a bore-hole was deleted in the 3D model, the radius dimension associated to the bore-hole in the existing technical drawing will be regenerated.

- 3D Geometry

Magenta	Equal 3D	The same geometrical location and same unique name in both versions of the drawing.
Yellow	Updated 3D	The same unique name but not the same geometrical location in both versions of the drawing.
Cyan	New 3D	New geometry that has been added to the original design.

The second use of  **Load & Update** loads a Creo Elements/Direct Drafting drawing and a Creo Elements/Direct Modeling layout but updates the specified view only.

If no view is given, Creo Elements/Direct Drafting prompts you to specify a view to update. You can identify a view by entering the string (for example `front1`) or select a view by calling the file browser `FBROWSER`.

The `STORE CURR` command stores the currently loaded drawing to the file system.

Setting Colors

To change the color scheme for transferred 2D elements:

1. Click **ADU** and then, in the **Setup** group, click  **Transferred 2D**.
2. Select a color.

To change the color scheme for updated 2D elements:

1. Click **ADU** and then, in the **Setup** group, click  **Updated 2D**.
2. Select a color.

To change the color scheme for regenerated 2D elements:

1. Click **ADU** and then, in the **Setup** group, click  **Regenerated 2D**.
2. Select a color.

To change the color scheme for equal 3D drawing:

1. Click **ADU** and then, in the **Setup** group, click  **Equal 3D**.
2. Select a color.

To change the color scheme for updated 3D drawing:

1. Click **ADU** and then, in the **Setup** group, click  **Updated 3D**.
2. Select a color.

To change the color scheme for new 3D drawing:

1. Click **ADU** and then, in the **Setup** group, click  **New 3D**.
2. Select a color.

The color changes take effect with the next redraw.

Showing Drawing Differences

To temporarily view the results of the ADU in different colors (with specific colors for elements that are only in `layout_1` or `layout_2`):

- Click **ADU** and then, in the **Show** group, click **Diff**.

To view the updated drawing in the original colors:

- Click **ADU** and then, in the **Show** group, click **Reset**.

Note

- All texts and geometry that was added to complete the original 2D drawing is transferred to the new layout at its original position.
 - Associated geometry (center lines and symmetry lines) are treated in the same way as annotations. For example, a center line follows a moved hole.
 - Detail parts are not updated but are also transferred to the new layout at their original position and with their original geometry.
-

Accepting and Changing Dimensions

When ADU has finished processing, you must decide whether to accept or change the updated annotations. You do this by using **Accept** and **Change** in the **Continue** group on the **ADU** tab. You can also use **Accept** in conjunction with  **Select** to accept, for example, 100 dimensions in one step.

To accept a blue dimension:

1. Click **ADU** and then, in the **Continue** group, click **Accept**.
2. Select a blue dimension.

Once accepted, the dimension is shown in its original color (yellow if the default has not been changed) and cannot be undone. If you refresh the drawing, the original dimensioning color remains.

To accept a red dimension:

1. Click **ADU** and then, in the **Continue** group, click **Accept**.
2. Select a red dimension.

Once accepted, the dimension is deleted together with any regenerated reference elements.

There may be instances where you want to relocate a dimension that ADU proposes. You can do this using **Change** in the **Continue** group on the **ADU** tab.

To change a dimension vertex:

1. Click **ADU** and then in the **Continue** group, click **Change**.
2. Select the dimension.
3. Select the new reference element.
4. Click the point where you want to locate the dimension.

Creo Elements/Direct Drafting redraws the dimension and deletes the old vertex point.

You can also use **Change** to reposition dimensions and to regenerate hatches (keeping original attributes).

The following table summarizes the differences between the **Accept** and **Change** when combined with the different annotation colors.

Differences between Accept and Change

Button	Description
Accept	<p>Tells the system to accept a proposal of ADU.</p> <ul style="list-style-type: none"> Blue: Leave annotation as it is. After you accept the annotation, it appears in its original color. Red: Delete annotation together with regenerated reference points (also red).
Change	<p>Tells the system to exchange the elements to which annotations refer. For example, you can exchange a dimension reference-element without deleting the dimension. When changing recreated (red) reference points against valid reference elements, the red reference points are removed after a successful exchange.</p> <p>Exchanging the reference elements (the contour) of a hatched face means removing the old hatch and specifying a new contour by retaining the hatch attributes.</p> <p>After you change the annotation, it appears in its original color.</p>

Accept can be used on blue and red annotations only. **Change** can be used on all annotations, regardless of the color. It can also be used on annotations that are not related to the ADU process.

Associative Elements

When working with associative elements (symmetry lines, reference lines, and center lines), please note the following:

- If the associative elements have been moved but not modified (means, for example, not stretched), they will be displayed in green ( **Transferred 2D**).
- If the associative elements have been modified (like the circle radius having been increased) the center lines will be displayed in blue ( **Updated 2D**).

- If associative elements have been deleted (for example, the circle), the center lines will be displayed in red ( **Regenerated 2D**).

Accept and **Change** also work with associative elements. The following table shows the effects of combining the buttons with the different colors.

Color	Button	Effects and Actions
Red	Accept	Element is deleted.
Blue	Accept	Element will change color.
Red	Change	The old elements are deleted and you must select pick new associated elements.
Blue	Change	The old elements are deleted and you must select pick new associated elements.

Modifying Dimensions

To move a dimension:

1. Click **ADU** and then, in the **Modify** group, click **Move Dim**.
2. Select the dimension.
3. Click the location where you want to move the dimension.
1. Click **ADU** and then, in the **Modify** group, click **Reset Dim**.
2. Select the dimension.

Modifying Views

Once you have accepted or changed ADU proposals, you modify the position and scale of individual views.

When working with ADU, you should not use the buttons on the **Modify** tab because association may be lost.

By Reference Point

To move a view using a reference point:

1. Click **ADU** and then, in the **Modify** group, click **Move View**.
2. Click **2 Points**.
3. Click a source point.
4. Click a destination point for the view.
5. Select the view.

Horizontal Move

To move view to a new position in the horizontal direction:

1. Click **ADU** and then, in the **Modify** group, click **Move View**.
2. Click **Horizontal**.
3. Type the horizontal distance in the user input line and press ENTER.
4. Select the view.

Vertical Move

To move a view to a new position in the vertical direction:

1. Click **ADU** and then, in the **Modify** group, click **Move View**.
2. Click **Vertical**.
3. Type the vertical distance in the user input line and press ENTER.
4. Select the view.

Rotate a View

To rotate a view about a center:

1. Click **ADU** and then, in the **Modify** group, click **Rotate View**.
2. Type the angle of rotation in the user input line and press ENTER.
3. Select the view.

Scale a View

To scale a view about a center:

1. Click **ADU** and then, in the **Modify** group, click **Scale View**.
2. Type a scale factor in the user input line and press ENTER.
3. Select the view.

Example

The following example shows the steps that you must follow when working with ADU. For reasons of clarity only one view is considered and it is assumed that both layouts are available. Before you begin however, it may be helpful to think of the complete process as 9 steps:

1. Create the first design with Creo Elements/Direct Modeling.
2. Create the first undocumented layout with Creo Elements/Direct Modeling.
3. Document this first layout with Creo Elements/Direct Drafting.

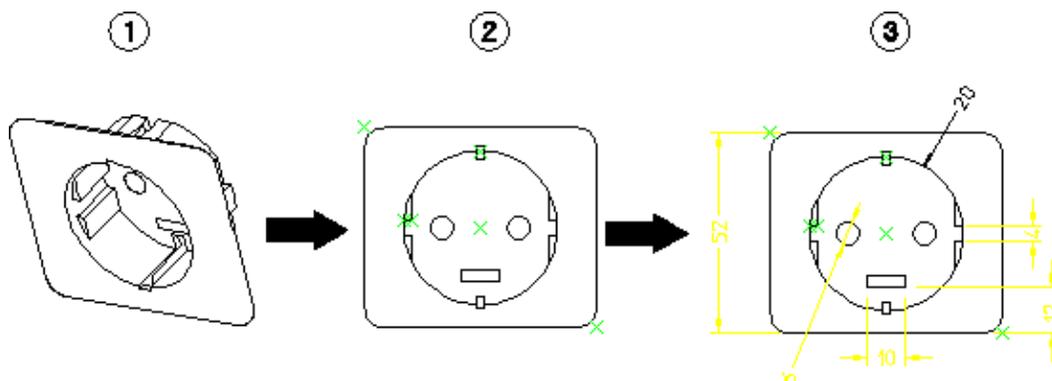
4. Make changes to the first design in Creo Elements/Direct Modeling and create the second design.
5. Create the second undocumented layout with Creo Elements/Direct Modeling.
6. Load the first layout which is documented into Creo Elements/Direct Drafting with ADU.
7. Load the second, undocumented layout into Creo Elements/Direct Drafting with ADU
8. Accept, reject, or modify the proposals from ADU.
9. Save the new, resulting layout to a file.

The example is a two-pin electrical socket designed using Creo Elements/Direct Modeling with layout-filename `socket_ly1`. This design is then modified according to the following:

- Added: Center hole for the retaining screw.
- Deleted: Punched slot.
- Moved: Left and right recesses.

The following figure shows the first 3D design, the undocumented first layout, and the documented first layout.

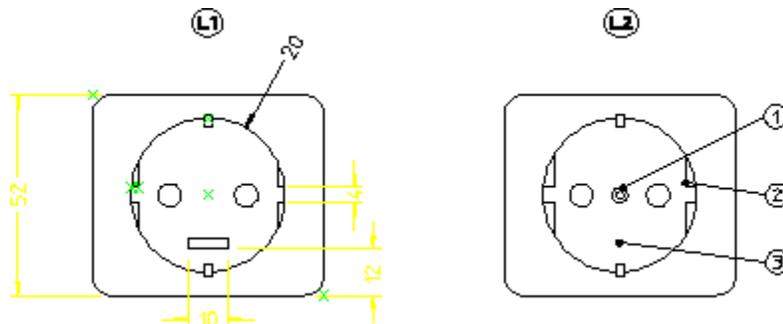
First Design



- 1 3D Design
- 2 Undocumented first layout
- 3 Documented first layout

The following figure shows the first layout and the second layout side-by-side. Layout 2 contains the new and modified geometry but is not documented. The numbered items referred to are:

First Layout



- 1 Center hole added
- 2 Left and right recesses modified
- 3 Punched slot deleted

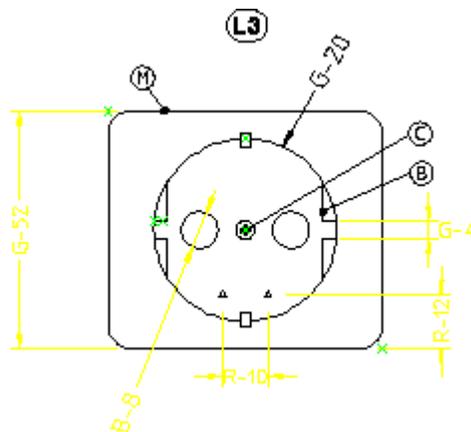
The filename for the new layout is `socket_ly2`. You are now ready to use ADU.

1. Click **ADU** and then, in the **Setup** group, click  **Load & Update**.
2. In the **Drawing** box, type `socket_ly1`.
3. In the **Layout** box, type `socket_ly2`.

Creo Elements/Direct Drafting superimposes the second layout on the first layout and assigns colors to elements and geometry.

In the following examples, colors are indicated by the following texts and dimension prefixes:

Layouts Superimposed



- G: Green
- R: Red
- M: Magenta
- C: Cyan

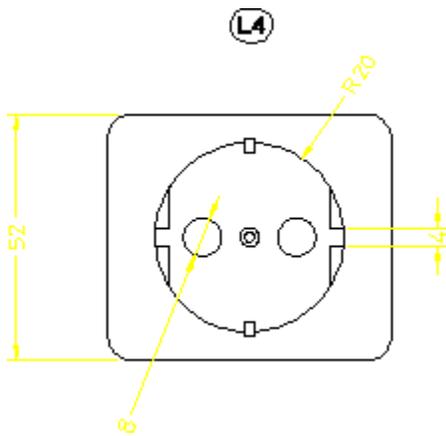
- B: Blue

You must now decide which of the proposals from ADU to accept or change.

1. Click **ADU** and then, in the **Continue** group, click **Accept**.
2. Select the R-10 dimension.
3. Select the R-12 dimension.

The following figure shows the cleaned-up drawing.

Updated Layout



20

Editing Imported Pictures

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Show - Image Options

Show commands allow you to control the tradeoff between image quality and processing speed for redraw.

Show or Hide

Images can be hidden to speed up processing. If the drawing includes several images, you can choose to hide individual images or all images.

Use **Show** or **Hide** to show or hide pictures.

To show or hide pictures:

1. Click **Modify**, and then in the **Picture** group, click  **Picture Browser**.
The **Picture Browser** opens.
2. Click the required picture name in the **Name** column.
3. Click **Show** to show the picture in the drawing.
Click **Hide** to hide the picture in the drawing.



Tip

While zooming, click the **Show Box** in the **Picture Browser** to hide the images.

Quality

When large drawings are placed on a small screen, some pixels from the original drawing cannot be shown. Two or more pixels from the large drawing must be mapped into one destination pixel on the screen. Traditionally, this operation meant that small lines and details could disappear.

Creo Elements/Direct Drafting provides aliasing to let you control quality vs time trade-offs of large drawing displays. Aliasing algorithms establish an area of investigation in the large drawing; formulas compute an average color tone for that area; and the system maps that color result to the destination pixel. Areas of investigation are based on one pixel and a pattern of its neighbors. Color tone result is grey-scale, an average of the original black and white pixels.

To control display quality using aliasing:

1. Click **Modify** and then, in the **Picture** group, click **More**.
2. Click  **Anti-alias Factor**.

3. Type the picture quality number (between -2 and +2) in the user input line and press ENTER.
4. Select the picture.

Positive input between 0 and 2 indicates the system to use a **block** of converging pixels in the aliasing formula. Input 2 to use the largest possible block of neighboring pixels. This results in the slowest possible operation.

Input 0 to produce the fastest operation. No aliasing.

Negative input between 0 and -2 tells the system to use pixels on a **diagonal** in the aliasing formula. This option is much faster than using blocks, and often produces a picture of good quality. Input -2 to use the largest number of neighboring diagonal pixels in the aliasing formula.

Editing Pictures

When you have imported a bitmap (BMP or TIFF or JPEG or PNG) file as described in [Importing a Picture File on page 544](#), you can edit it as an image in Creo Elements/Direct Drafting.

Arranging Images

To arrange images:

1. Click **Modify** and then, in the **Picture** group, click the arrow next to  **Level**.
2. Click the required  **Level** button:

Level Button	Purpose
 To Front	Move an image in front of all other objects.
 To Back	Move an image at back of all other objects.
 Forward	Move one level forward.
 Backward	Move one level backward.
 In Front Of	Move in front of a reference object.
 Behind	Move behind a reference object.

3. Select the image you want to move to the foreground or background.

When two pictures overlap, only one picture is shown on the screen or printed. All the pictures within the same part are ordered according to level. Similarly, subparts are ordered according to level. In other words, the level is the position of an object (picture or part) in the sequence of all the objects within the same part (without subparts).

The picture that is in front of another is shown when two pictures overlap.

 **Note**

-  **Level** changes the level of a single part or subpart as well as a picture.
 - If you have multiple subparts, you cannot have some pictures in one subpart in front of pictures in another subpart, and some behind. You can only have all pictures from one subpart in front of all pictures in another subpart.
-

Using an external editor

1. Click **Modify** and then, in the **Picture** group, click **More**.
2. Click  **External Edit**.
3. Click the picture to be edited.

An external editor opens the picture in a new window.

4. Edit the picture and close the external editor.

The default external editor is MS Paint. To specify a different external editor, set the environment variable `MEPICEDITOR` (first choice) or `SDTOOLBARICONEDIOR` (second choice).

Updating a picture

To update a single picture with the contents of the new bitmap picture file:

1. Click **Modify** and then in the **Picture** group, click **More**.
2. Click  **Update**.
3. Click the picture you want to update.
4. Type the file path and file name of the new picture file in the user input line.

To update the earlier uploaded file, type an empty string (two single quote marks - ' ') in the user input line.

5. Press ENTER.

Renaming a picture

To rename a picture in Creo Elements/Direct Drafting:

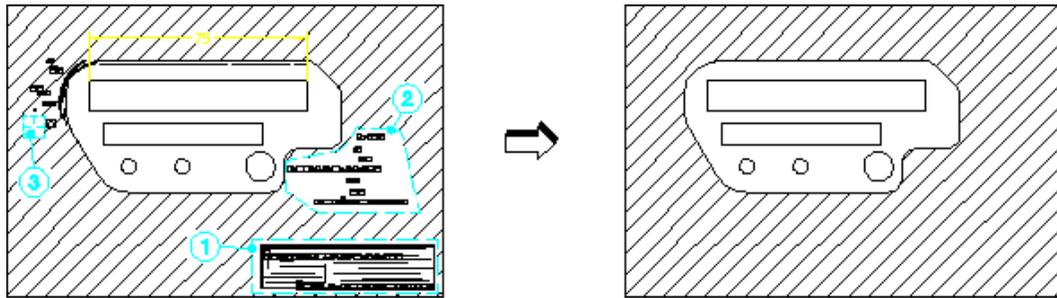
1. Click **Modify** and then, in the **Picture** group, click **More**.
2. Click  **Rename**.
3. Select the picture.
4. Type the new name for the picture (in quotes) in the user input line and press ENTER.

You can verify the change in name of the picture using the **Part Browser**.

Changing the Color of Pixels

After you have imported an image, you probably want to clear unwanted information or simply clean it of the debris usually associated with scanned images. This is done by changing the color of the pixel data. If the image is black and white, you can only reverse the color (black to white or white to black). Changing the color of individual pixels to the background color of the image appears to delete the pixel. However, you have only colored the pixel. It is not possible to delete pixels in order to see through to an underlying image.

Figure 200. Cleaning an Imported Image



- 1 **Box**
- 2 **Poly**
- 3 **Brush**

Creo Elements/Direct Drafting allows you to clean or edit an image using  **Color**.

To change the color of pixels:

1. Click **Modify** and then, in the **Picture** group, click  **Color**.
2. Click  **Change**.
3. Select a single picture.

4. Click any of the following **Color** button:

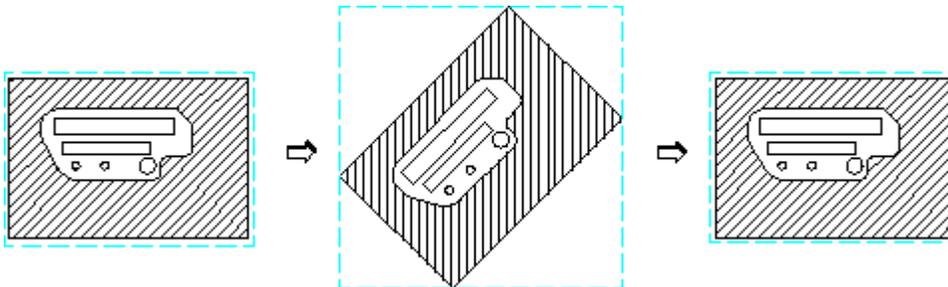
- Click  **Box** and enclose the required pixels in a box.
- Click  **Poly** and enclose the required pixels in a polygon.
- Click  **Brush** and press SHIFT while dragging the cursor over the pixels to be colored.

5. Click  **OK**.

Additional Information:

- If the **Brush** color is different from the background color, you can add or red-line information on the image.
- The image is always described by a rectangular frame. If you rotate the image, the frame increases or decreases according to the horizontal and vertical lines passing through the corners; see [Figure 201. Rotating an Imported Image on page 582](#). The swept area is only temporary — it disappears along with any pixel editing you may have done.

Figure 201. Rotating an Imported Image



Inverting Pictures on Black and White Displays

Creo Elements/Direct Drafting allows you to change between white-on-black and black-on-white displays using  **BW Invert**:

1. Click **Modify** and then, in the **Picture** group, click **More**.
2. Click  **BW Invert**.
3. Select the picture to invert.

-
4. Click  **OK**.

 **Note**



BW Invert only works on true black and white displays, not grey or color.

Transparency

You can select one color in your picture to become transparent when that picture overlays another picture. The color you select for transparency becomes invisible, allowing you to see through those pixels to the lower picture.

To set transparency attributes:

1. Click **Modify** and then, in the **Picture** group, click the arrow next to  **Color**.
2. Click **Transparency**.
3. Click any of the following:
 - Click **On** and select the picture.
If the picture has a transparency color, the color becomes invisible during overlay.
 - Click **Off** and select the picture.
If that picture has a transparency color, the color does not become invisible (default setting).
 - Click **Color**.
The **Select Color** dialog box opens. In the **Color** box, select a color and click **OK**. Select the picture to apply the selected transparency color to it.

 **Note**

Transparent picture attributes must be set by Creo Elements/Direct Drafting. Attributes from other programs such as those written in jpg, tiff, etc. are not supported. All imported files are opened as non transparent.

Using the Picture Browser

To use the **Picture Browser**:

- Click **Modify** and then, in the **Picture** group, click  **Browser**.

Creo Elements/Direct Drafting displays the **Picture Browser**.

The **Picture Browser** displays the following information about the currently loaded images:

Name	Filename of the image
Z-level	Numerical value for the "hidden" level assigned to an image in the drawing
Show	Status of the SHOW command: On, OFF, or BOX
X-Scale	Scaling factor in the x-direction
Y-Scale	Scaling factor in the y-direction
Angle	Angle of rotation
Anti-A	Anti-alias factor of the image
Trans On	Image transparency on
Trans Off	Image transparency off

Additional Information

- The imported figure is embedded in the MI file format. In the MI file, the pixel data will be handled in a special section as ASCII characters.
- The imported image maintains its aspect ratio.
- If two images overlap each other, you can use  **Level** in the **Picture** group on the **Modify** tab to move the image forwards or backwards. Click a non-overlapping area to select the image. Alternatively, type the name in the user input line and press ENTER.
- If drawing geometry overlaps an image, it is placed at the front of the image or images.

Tip

Change the color of the overlapping geometry.

- Plotting of images is supported by raster-based plotters only (for example, plotters supporting PostScript or HP-GL/2 plus RTL). Images are ignored on non-supported plotters.
- IGES and DXF/DWG translators do not translate pictures by default and report a warning message in the log file when pictures are skipped.

-
- The Store Pixel Data switch in the IGESo.conf file allows storage of picture to separate files.
 - When a picture is loaded, the size of the picture (in current units) is determined by the factor (current_viewport->factor which is number of current units per pixel). Hence the actual size of the image depends on the current value of the zoom factor.
 - When an image with more colors than supported by the hardware is loaded and subsequently saved, loss of color occurs. For example, if an image containing 1500 colors is loaded on a machine which can display only 256 colors and saved, only 256 out of the 1500 colors are saved in the MI file. (The pixels with the lost colors are mapped to the nearest available color in the color table.) This changes the size of the MI file once a load and save is done across different platforms (with different number of bitplanes) without any deletion of data (image contents) in the file.

Further Information

For further technical information about pictures, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- help picture
- help change_level
- help chg_pixel_color
- help resample_picture
- help change_picture_aa_factor
- help rename_picture
- help change_picture_transparency
- help wui_inq_picture_color

Creo Elements/Direct Drafting displays further information about pictures.

21

Linking and Embedding OLE Objects

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Files (such as spreadsheets, graphics or text pages) created in external applications can be included in Creo Elements/Direct Drafting by inserting associated Object Linking and Embedding (OLE) objects. Drawings containing OLE objects can be printed as PDF and PS files.

Please note the following:

- OLE functionality is supported in the Fluent user interface and Windows user interface only.
- A Creo Elements/Direct Drafting file containing OLE objects added in the Windows user interface can be loaded in the Classic user interface but the OLE information is not displayed. However, the existing OLE information is retained, even if the Creo Elements/Direct Drafting file is saved in the Classic user interface.

Note

Creo Elements/Direct Drafting can also be inserted as an OLE object in other applications that support OLE.

Inserting an OLE Object

To insert an OLE object in a Creo Elements/Direct Drafting drawing:

1. Click **File** ►  **Import/Export** ►  **Import OLE**.

The **Insert Object** dialog box opens. It lists all applications on your system that provide OLE capability.

2. In the **Object Type** box, select an application.
3. Select one of the following:

- **Create New**
- **Create from File**

Create New

Create New inserts a new OLE object of the selected application in your Creo Elements/Direct Drafting drawing.

To create a new OLE object:

1. In the **Object Type** box, select an object type.
2. Click the **Display As Icon** check box to represent the OLE object by the associated application icon in Creo Elements/Direct Drafting.
3. Optionally, click **Change Icon** to select a different icon for the OLE object.
4. Click **OK**.

The **Open** dialog box opens.

5. Using the File browser, select the OLE object.

The default OLE object name is displayed in the user input line.

6. Retain the default OLE object name or type a name for the OLE object in the user input line and press ENTER.
7. Specify the insertion position for the OLE object in Creo Elements/Direct Drafting.

The OLE object is displayed in the Creo Elements/Direct Drafting drawing.

Create from File

Create from File inserts the contents of a file as an object in your document so that you can activate the object using the application in which it was created.

To create an OLE object from an existing file:

1. Click **Browse** and browse to the file you want to link using the file browser.

-
2. Click **Open**.
 3. Optionally, click the **Link** check box to link the OLE object to its source application such that changes in the original file are reflected in the OLE object in Creo Elements/Direct Drafting.
 4. Optionally, click the **Display As Icon** check box to represent the OLE object by the associated application icon in Creo Elements/Direct Drafting.
 5. Optionally, click **Change Icon** to select a different icon for the OLE object.
 6. Click **OK**.
 7. Retain the default OLE object name or specify a name for the OLE object in the user input line and press ENTER.
 8. Specify the insertion position for the OLE object in Creo Elements/Direct Drafting.

The OLE object is displayed in the Creo Elements/Direct Drafting drawing.

 **Note**

If the inserted OLE Object is a Microsoft Excel sheet, the contents displayed may differ from the actual sheet depending on the font size, the row height, and the number of rows and columns used in the actual sheet. The size of the inserted OLE object remains the same even after you add rows or columns to the actual sheet. However, there is no loss of information in the Microsoft Excel sheet even if all the rows and columns are not displayed.

Editing an OLE Object in Creo Elements/ Direct Drafting

Once the OLE object is inserted in the Creo Elements/Direct Drafting drawing, you can edit it as described in the following sections.

Arranging OLE Objects

To arrange OLE objects:

1. Click **Modify** and then, in the **OLE** group, click the arrow next to  **Level**.
2. Click  **To Front**,  **To Back**,  **Forward** or  **Backward**, or  **In Front Of** or  **Behind**.

3. Select the **OLE** object you want to arrange.

Activating an OLE Object

By activating an OLE object, you open the underlying file in its source application for editing. The OLE objects created by using **Create from File** reflect the changes made to the underlying file.

To activate an OLE object:

1. Click **Modify** and then, in the **OLE** group, click **Activate**.
2. Click the OLE object you want to activate.

Renaming an OLE Object

You can rename an existing OLE object in a drawing.

To rename an OLE object:

1. Click **Modify** and then in the **OLE** group, click  **Rename**.
2. Select the OLE object to be renamed.
3. Type the new name of the OLE object in the user input line and press ENTER.

Using the OLE Browser

To open the **OLE Browser**:

- Click **Modify** and then, in the **OLE** group, click  **Browser**.

The **OLE Browser** displays the following information about the OLE objects currently included in your Creo Elements/Direct Drafting drawing:

Name	The name you specified for the OLE object
Z-level	Numerical value for the "hidden" level assigned to an image in the drawing
Show	Status of the SHOW command: ON, OFF, or ICON
X-Scale	Scaling factor in the x-direction
Y-Scale	Scaling factor in the y-direction
OLE Type	OLE object type
Program ID	Application object type
Class ID	OLE object class ID

Using the **OLE Browser**, you can perform the following actions on the OLE objects:

-
- Activate
 - Delete
 - Show
 - Hide
 - Box
 - Convert to icon
 - Rename
 - Save

Further Information

For further technical information about OLE objects, refer to the Creo Elements/Direct Drafting help system.

Type the following in the Creo Elements/Direct Drafting user input line and press ENTER:

- `help ole_list`
- `help activate_ole`
- `help rename_ole`

Creo Elements/Direct Drafting displays further information about OLE objects.

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Annotation

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This chapter discusses the functionality from the Creo Elements/Direct Drafting **Annotation** tab.

Introduction

The functionality from the **Annotation** tab in Creo Elements/Direct Drafting enables you to visualize, annotate and edit MI format 2D drawings generated by Creo Elements/Direct Annotation.

General Procedure

The functionality from the **Annotation** tab in Creo Elements/Direct Drafting Annotation comprises the following steps:

- Prepare the Creo Elements/Direct Drafting drawing environment for Annotation drawings.
- Load an MI file generated by Creo Elements/Direct Annotation in Creo Elements/Direct Drafting.
- View, annotate and edit this file as appropriate. For example, you can manipulate views and you can hide or delete elements. Also, you can modify and/or add geometry by means of regular Creo Elements/Direct Drafting commands.

Note

A warning message is displayed for any "structurally dangerous" operations that would impair association between 3D model or 2D drawing.

- After making changes to the underlying 3D model in Creo Elements/Direct Modeling, update the relevant views in Creo Elements/Direct Modeling and then review the resulting changes in 2D (Creo Elements/Direct Drafting).
- Edit the information in the 2D views as appropriate (for example, dimensioning elements coming from 3D, adding text, and so on).
- Check the part structure of the Annotation MI file by means of the Creo Elements/Direct Drafting Part Browser.
- Save the 2D drawing for further use in Creo Elements/Direct Drafting or in Creo Elements/Direct Annotation.

Annotation Rules

To ensure that the Annotation MI drawing file retains its 3D or 2D association for further use in Creo Elements/Direct Modeling, you must observe the following rules when using the functionality on the **Annotation** tab in Creo Elements/Direct Drafting:

- Do not copy one drawing in another drawing.
- Do not manually insert one or more sheets into another drawing.
- Do not manually insert one or more sheets in another drawing and do not manually insert views into a sheet (using the part definition commands of Creo Elements/Direct Drafting). Only use the buttons in the **General** group and the **Sheet** group provided by the Creo Elements/Direct Drafting **Annotation** tab.
- Do not change or delete info texts beginning with "SD_" or "DOCU" or "AM_".
- Do not modify view parts or parts below view parts with the buttons on the **Modify** tab of Creo Elements/Direct Drafting. Only use the buttons in the **View** group provided by the Creo Elements/Direct Drafting **Annotation** tab.
- Do not scale view parts or parts below view parts using the buttons in the **Parts** group on the **Structure** tab of Creo Elements/Direct Drafting. Only use **Scale View** in the **View** group on the **Annotation** tab of Creo Elements/Direct Drafting.
- Do not change the scale of the TOP part or the parts below the TOP part with the  **Scale** buttons in the **Parts** group on the **Structure** tab of Creo Elements/Direct Drafting.
- Do not change the owner of a part or subassembly which belongs to the drawing in Creo Elements/Direct Drafting.
- Do not use Creo Elements/Direct Drafting to smash or gather any part created by Creo Elements/Direct Annotation (sheets, frames, view parts, parts below view parts, sketches).
- Do not create detail parts in Creo Elements/Direct Drafting — these would not be associative to the 3D model. Use the **Detail** commands on the **Annotation** tab of Creo Elements/Direct Modeling to create detail views.

Preservation of Modifications

The following modifications done with the **Annotation** functionality in Creo Elements/Direct Drafting are preserved when an update is performed in Creo Elements/Direct Annotation:

- Attribute changes such as line type or line color, or addition of info texts to the geometrical elements coming from the 3D model.
- Center lines created in Creo Elements/Direct Drafting.
- Dimensions, texts, symbols and additional 2D geometry created in the appropriate view parts.

 **Note**

Symbols, text and additional 2D geometry can be added to each part except the TOP part.

- Geometrical elements belonging to subparts which are not owned by the TOP part named "SD_drawing".
- Hatches added in Creo Elements/Direct Drafting.

Limitations

Some operations performed with the **Annotation** functionality in Creo Elements/Direct Drafting will not deliver the expected results after reloading and updating in Creo Elements/Direct Annotation.

- Parts created in Creo Elements/Direct Drafting as sub-parts of the TOP part named "SD_drawing" will not be loaded into Creo Elements/Direct Annotation.
- Geometry or any other additional elements created in the TOP part named "SD_drawing" will not be loaded into Creo Elements/Direct Annotation.
- Subparts created in Creo Elements/Direct Drafting that are not owned by the TOP part are not displayed in the Drawing Browser of Creo Elements/Direct Annotation. However, their geometry is displayed in the viewport.
- Leader lines (or text with leaderlines) created in Creo Elements/Direct Drafting do not behave associatively in Creo Elements/Direct Annotation.
- Hatch holes automatically created in Creo Elements/Direct Drafting which have been forced by creation of dimensions or texts with Creo Elements/Direct Drafting option `TEXT_HOLE_INSERTION` being set to `ON` by default do not update associatively in Creo Elements/Direct Annotation.
- Hatch holes automatically created in Creo Elements/Direct Annotation for texts do not update associatively in Creo Elements/Direct Drafting when the text is modified in Creo Elements/Direct Drafting.
- Hatches that were automatically created in Creo Elements/Direct Annotation but deleted in Creo Elements/Direct Drafting reappear after the next update in Creo Elements/Direct Annotation.
- 2D view parts created in Creo Elements/Direct Annotation that are renamed in Creo Elements/Direct Drafting get their original name during the next view update in Creo Elements/Direct Annotation.

-
- If you split or merge 2D geometrical elements in Creo Elements/Direct Drafting which were derived from 3D geometrical elements during a view update in Creo Elements/Direct Annotation, then the geometrical elements look like the original ones after the next view update in Creo Elements/Direct Annotation.

Preparing the Creo Elements/Direct Drafting Drawing Environment

Before you load a MI file generated in Creo Elements/Direct Annotation, ensure you complete the following steps:

1. To ensure that the top part is empty:
 - a. Click **Miscellaneous** and then, in the **System** group, click **Reset System**.
 - b. Click  **Confirm**.

Creo Elements/Direct Drafting changes the active part to the top of the drawing and deletes the entire drawing. The top part of the drawing is renamed to **Top**.
2. Activate the Creo Elements/Direct Drafting **Annotation** tab:
 - a. Click **File** ► **Options**.

The **Options** dialog box opens. The **Customize Ribbon** pane is selected by default.
 - b. In **Customize the Ribbon** list, click the **Annotation** check box.

Loading an Annotation Drawing

To load a MI file generated in Creo Elements/Direct Annotation:

1. Click **File** ► **Open**.
2. Select the MI file from the Creo Elements/Direct Drafting file browser.

Using the Sheet Browser to Navigate among Sheets and Views

The **Sheet Browser** enables you to quickly and easily select a sheet or a view included in a sheet.

Displaying the Sheet Browser

To display the sheet browser:

1. Click **Annotation** and then, in the **General** group, click **Sheet Browser**.
The **SHEETS** browser opens.
2. Click the sheet browser and position it in the drawing area.

Selecting a Sheet or View from the Sheet Browser

To select a sheet from the sheet browser:

1. In the sheet browser, click **Curr Sheet**.
2. Select the sheet name in the sheet browser.
The selected sheet becomes the current sheet.

To select a view from the sheet browser:

1. In the sheet browser, select the sheet that contains the view.
2. Click **Curr View**.
3. Select the view in the sheet.

The selected view becomes the current view.

Views

The Creo Elements/Direct Drafting **Annotation** tab provides a number of buttons for selecting, changing and manipulating views.

Selecting the Current View

To select the current view in the current sheet:

1. Click **Annotation** and then, in the **General** group, click **Current**.
2. Click **View**.
3. Select the view.

You can also use the sheet browser to select the current view. See [Selecting a Sheet or View from the Sheet Browser on page 596](#),

Modifying Views

You can change the position of views on a sheet. You can also move views between sheets.

Note

A warning message is displayed for any "structurally dangerous" operations that would impair 3D or 2D association.

To change the position of one or multiple views:

1. Click **Annotation** and then, in the **View** group, click **Move**.
2. Click one of the following:
 - **2 Points** to move the selected view or views from the first point to the second point.
 - **Horizontal** to move the selected view or views by the specified horizontal distance.
 - **Vertical** to move the selected view or views by the specified vertical distance.
 - **Sheet to sheet** to move the selected view or views to another sheet.

To rotate a view or views by an angle:

1. Click **Annotation** and then, in the **View** group, click **Rotate View**.
2. Type the angle of rotation in the user input line and press ENTER.
3. Select the view.

To scale a view:

1. Click **Annotation** and then, in the **View** group, click **Scale View**.
2. Type the view scale in the user input line and press ENTER.
3. Select the view.

Showing Hidden Lines

To show hidden lines in the Annotation MI file:

1. Click **Annotation** and then, in the **Show** group, click the **Hidden Line** check box.
2. Select the view that contains the hidden lines.

Showing Tangent Lines

To show tangent lines in the Annotation MI file:

1. Click **Annotation** and then, in the **Show** group, click the **Tangent Line** check box.

-
2. Select the view that contains the tangent lines.

Hiding Hidden Lines

To hide hidden lines in the Annotation MI file:

1. Click **Annotation** and then, in the **Show** group, clear the **Hidden Line** check box.
2. Select the view that contains the hidden lines.

Hiding Tangent Lines

To hide tangent lines in the Annotation MI file:

1. Click **Annotation** and then, in the **Show** group, clear the **Tangent Line** check box.
2. Select the view that contains the tangent lines.

Showing Differences in a View Following a 3D Change

Any view you see using the Creo Elements/Direct Drafting **Annotation** tab is a 2D derivative of a 3D model view in Creo Elements/Direct Drafting. When the underlying 3D model changes in Creo Elements/Direct Modeling, you can update the associated view with the **Annotation** module in Creo Elements/Direct Modeling and check with the Annotation functionality in Creo Elements/Direct Drafting for differences in the view before and after the update.

Showing Differences

To show differences in a view:

- Click **Annotation** and then, in the **Show** group, click **Difference**.

Note

- By default, the Creo Elements/Direct Annotation update colors are used to mark differences as well as unchanged elements.
 - Any unchanged elements are marked by the equal info text flag.
 - Any changed elements are marked by the non-equal info text flag.
-

Hiding Differences

To hide the differences in a view before and after an update to the underlying 3D model:

- Click **Annotation** and then, in the **Show** group, click **Reset**.

Note

Reset returns the view to the state before **Difference**.

Handling Annotation Elements

Optionally, you can change or accept annotation elements.

Changing an Annotation

You can change an annotation, for example, by selecting a different reference element.

To change an annotation:

1. Click **Annotation** and then, in the **Modify** group, click **Annotation**.
2. Click **Change**.
3. Select the annotation.
4. Perform the steps displayed in the user input line.

Accepting All Annotation Elements

To accept all (current or changed) annotation elements:

1. Click **Annotation** and then, in the **Modify** group, click **Annotation**.
2. Click **Accept**.

Dimensions

Moving a Dimension

To move a dimension:

1. Click **Annotation** and then, in the **Modify** group, click **Dimension**.
2. Click **Move**.

-
3. Select the dimension and drag it to the new position.

Returning a Dimension to the Original Position

To move a dimension back to its original position:

1. Click **Annotation** and then, in the **Modify** group, click **Dimension**.
2. Click **Reset**.
3. Select the dimension.

Sheets

You can create, select, change, and modify sheets.

Creating a Sheet

To create a new sheet:

1. Click **Annotation** and then, in the **Sheet** group, click **New**.
2. Click one of the following:
 - **Frame from File**
 - **Frame from Database**
3. Type the number of the new sheet in the user input line and press ENTER.

The **Load Frame** dialog box opens.

Note

You can see the existing sheet numbers in the sheet browser.

4. Select a frame for the new sheet.

You can then add new views to this sheet, for example, by moving from an existing sheet or by loading a view from Creo Elements/Direct Drafting. See [Modifying Views on page 596](#) for more information.

Selecting the Current Sheet

To make a sheet current:

1. Click **Annotation** and then, in the **General** group, click **Current**.
2. Click **Sheet**.

3. Select the sheet in the **Part Browser**.

Alternatively, select the current sheet from the sheet browser (see [Selecting a Sheet or View from the Sheet Browser on page 596](#)).

Modifying a Sheet

You can change the position of the frame of a sheet. Also, you can replace the current frame of a sheet by a different sheet, or you can delete a sheet.

Note

A warning message is displayed for any "structurally dangerous" operations that would impair 3D or 2D association.

To modify a sheet:

- Click **Annotation** and then, in the **Sheet** group, click one of the following:
 - **Frame Type** to select a different frame from the **Load Frame** dialog box. The existing frame is replaced by the new frame.
 - **Frame Position** to move a frame to a new position.
 - **Renumber** to renumber a sheet.
 - **Delete** to delete a sheet.

Hiding, Deleting, Unhiding

The Creo Elements/Direct Drafting **Annotation** tab provides a number of commands for selectively hiding, deleting or unhiding elements in the drawing.

Hiding or Deleting Elements

Any elements you hide are simply "invisible" in the drawing. Any elements you delete are physically removed from the drawing.

Note

A warning message is displayed for any "structurally dangerous" operations that would impair 3D or 2D association.

To hide or delete elements:

-
1. Click **Annotation** and then, in the **Modify** group, click **Hide Geometry**.
 2. Click one of the following:
 - **No Delete** to hide elements without deleting them.

 **Note**

You can restore the hidden elements using the buttons in the **Unhide Geometry** group.

- **Previous Delete** to delete the elements that you have selected using **No Delete**.

 **Note**

You cannot restore the elements that you delete with **Previous Delete** using the buttons in the **Unhide Geometry** group.

- **Final Delete** to delete (remove) the elements.

 **Note**

You cannot restore the elements that you delete with **Final Delete** using the buttons in the **Unhide Geometry** group.

3. Select the elements or enclose the elements in a box to hide or delete them.

Unhiding Elements

You can unhide any elements that you have hidden using **No Delete**.

Unhiding Selected Elements

To selectively unhide the elements that are hidden (but not deleted):

1. Click **Annotation** and then, in the **Modify** group, click **Unhide Geometry**.
2. Click **Selected**.

The elements that you have hidden using **No Delete** are displayed in a special highlight color.

3. Select the elements that you want to unhide or enclose the elements in a box

-
4. Click  **Confirm**.

Sequentially Unhiding Elements

Optionally, you can sequentially unhide elements that have been hidden (but not deleted).

To sequentially unhide the elements:

1. Click **Annotation** and then, in the **Modify** group, click **Unhide Geometry**.
2. Click one of the following:
 - **Backward** to undo your **Hide Geometry** operations in reverse order (the last **Hide Geometry** operation is undone first).
Optionally, click **Backward**, type the number of backward steps in the user input line, and press ENTER.
 - **Forward** to undo your **Hide Geometry** operations in the order in which they were performed (the first **Hide Geometry** operation is undone first).
Optionally, click **Forward**, type the number of forward steps in the user input line, and press ENTER.

Any unhidden elements are now shown in a special highlight color.

Changing Colors, Linetypes and Hatches

On the **Annotation** tab, you can change the color and linetype of hidden lines and tangent lines. Also, you can change the color, angle, and distance of hatches.

Changing the Color and Linetype of Hidden Lines

To change the color of hidden lines:

1. Click **Annotation** and then, in the **View** group, click **Hidden Line**.
2. Click **Color** and select the view.
The **Select Color** dialog box opens.
3. In the **Color** box, select a color.
4. Click **OK**.

To change the linetype of hidden lines

1. Click **Annotation** and then, in the **View** group, click **Hidden Line**.
2. Click **Line Type** and select the view.

The **Change Line Type** dialog box opens.

-
3. In the **Type** box, select a linetype.
 4. Click **OK**.

Changing the Color and Linetype of Tangent Lines

To change the color of tangent lines:

1. Click **Annotation** and then, in the **View** group, click **Tangent Line**.
2. Click **Color** and select the view.

The **Select Color** dialog box opens.

3. In the **Color** box, select a color.
4. Click **OK**.

To change the linetype of tangent lines:

1. Click **Annotation** and then, in the **View** group, click **Tangent Line**.
2. Click **Line Type** and select the view.

The **Change Line Type** dialog box opens.

3. In the **Type** box, select a linetype.
4. Click **OK**.

Changing the Color, Angle, and Distance of Hatches

To change the color, angle, and distance of hatches:

1. Click **Annotation** and then, in the **View** group, click **Hatch**.
2. Click **Color** to change the color of hatches.
3. Click **Annotation** and then, in the **View** group, click **Hatch**.
4. Click **Angle** to change the angle of hatches.
5. Click **Annotation** and then, in the **View** group, click **Hatch**.
6. Click **Distance** to change the distance of hatches.

Refer the general instructions provided in [Changing the Drawing Environment and Making Measurements on page 90](#) to change the color angle, and distance of hatches.

Showing and Storing

When you have completed your work on the Annotation MI drawing file, you may want to see what this drawing would look like in **Annotation** or in **Creo Elements/Direct Drafting**.

Showing the Drawing in Annotation Mode

To show the drawing in Annotation mode:

- Click **Annotation** and then, in the **Show** group, click the **As Annotation** check box.

This mode displays your **Hide Geometry** operations.

Showing the Drawing in Creo Elements/Direct Drafting Mode

To show the drawing in Creo Elements/Direct Drafting mode:

- Click **Annotation** and then, in the **Show** group, clear the **As Annotation** check box.

This mode ignores any **Hide Geometry** operations but displays any **Prev Delete** and **Final Delete** operations.

Storing the Drawing for Further Use in Annotation

To store the drawing for further use in Creo Elements/Direct Annotation:

1. Click **File** ► **Save**.
2. Specify the path and the file name to save the current Annotation MI drawing.

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Printing and Plotting

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This chapter describes how to print and plot drawings and create screen dumps.

Using the Print Manager

Creo Elements/Direct Drafting spools all plots and screen dumps either to the Print Manager or to the Clipboard, and not directly to an output device.

The Print Manager controls device selection, configuration, and spooling for the printers and plotters on your system. Once you send a plot from Creo Elements/Direct Drafting to the Print Manager, the Print Manager takes over the process of producing the final output. By setting REQUEST_PRINT_SETUP, you can directly access the Print Manager's setup dialogs each time you plot or screen dump.

The Clipboard temporarily stores a plot or screen dump so that it can be electronically transferred or "pasted" in another application.

Note

Depending on the output device or devices on your system, the Print Manager may allow you to set several printer or plotter options in addition to those offered by Creo Elements/Direct Drafting. Refer to your Windows documentation and hardware manuals for information on configuring your printer or plotter to work with Windows.

Creating a Print Configuration

To facilitate the creation of consistent print or plot output, Creo Elements/Direct Drafting enables you to create any number of print configurations.

To create a print configuration:

1. Click **File** ►  **Print** ►  **Print Setup**.

Alternatively, click  **Print Preview and Print** on the Quick Access Toolbar or press CTRL+P.

The **Print** dialog box opens.

2. Define the settings for the new print configuration.

These settings are discussed in the following sections of this chapter.

3. In the **Print configuration** area, click **Settings**.

The **Print Configurations** dialog box opens.

4. Type a name for the print configuration in the text box and click **Add** to add this new print configuration name to the list of print configurations.

Creo Elements/Direct Drafting prompts for confirmation to add the print configuration to the list.

5. Click **OK**.
6. To save the print configuration for future use, select it and then, click **Store**.
The **Store Print Configurations** dialog box opens.
7. In the **Name** box, type a name for the print configuration file (For example, `my_print.m`) for your print configuration file.

 **Note**

This filename does not have to be the same as the print configuration name. However, ensure that the filename is readily identified.

8. Click **Save**.
9. Click **Close**.

Example of a Print Configuration

An example of a Creo Elements/Direct Drafting print configuration follows:

```
{ User specific customization }
```

```
WIN_PRT_MGR PAPER 'a3' ORIENTATION LANDSCAPE COPIES 1 END
```

```
PLOT_SCALE AUTO
```

```
PLOTTER_TYPE ADD MSWINDOW_GDI_PRINTER PRT_DRIVER 'HP LaserJet  
1100' 'department_1' END
```

```
WIN_PRT_MGR PAPER 'a4' ORIENTATION PORTRAIT COPIES 1 END
```

```
PLOT_SCALE AUTO
```

```
PLOTTER_TYPE ADD MSWINDOW_GDI_PRINTER PRT_DRIVER 'HP DeskJet'  
'department_2' END
```

```
WIN_PRT_MGR PAPER 'a4' ORIENTATION LANDSCAPE COPIES 1 END
```

```
PLOT_SCALE AUTO
```

```
PLOTTER_TYPE ADD MSWINDOW_GDI_PRINTER PRT_DRIVER 'HP LaserJet
1100' 'department_3' END
```

```
PLOTTER_TYPE 'department_3'
```

```
{ plot it }
```

```
PLOT 0,0 0 all BW_IMG NORM_VIDEO
```

The settings in this `customize_p.m` print configuration file imply the following:

- One copy of the drawing is printed to A4 paper in landscape orientation.
- The drawing is scaled to fit.
- The `PLOTTER_TYPE` variable is set to `department_3`. This is the printer that will be used as the output device.
- The entire drawing is printed in normal (as opposed to inverse) video.

Reusing a Saved Print Configuration

When you quit Creo Elements/Direct Drafting, the system automatically saves your current print settings. These settings are used when you start Creo Elements/Direct Drafting again.

You can see and modify your current configuration file by opening `<application data path>\PTC\Creo Elements Direct Drafting [version]\[version]\print_configurations\current_print_cfg.m`.

You can also use existing or create your own configuration files. If you use the name convention `(print_cfg_<configuration name>.m)` for your own configuration files and place them in the `print_configurations` subdirectory, these files appear in the **Print** dialog box as a **Print configuration**. The configurations that you create with the **Print** dialog box are automatically stored in this subdirectory when you store your configuration.

To reuse binary legacy print configuration files (with file name `<print configuration>.cfg`) from a CoCreate Drafting version `< 13.00`:

1. Click **File** ►  **Print** ►  **Print Setup**.

The **Print** dialog box opens.

Alternatively, press CTRL+P to open the **Print** dialog box.

2. In the **Print configuration** area, click **Settings**.

The **Print Configurations** dialog box opens.

3. Click **Load**.

The **Load Print Configurations** dialog box opens.

4. Select the print configuration file and click **Open**.

The print configurations appear in the **Print Configurations** list.

5. Select the required print configuration.
6. Click **Close**.

The **Print** dialog box settings change to the settings defined for the loaded print configuration.

Defining the Print or Plot Settings

Before printing or plotting a drawing, you should define exactly what you want to print or plot and how the drawing will be printed or plotted.

Setting the Plot or Print Destination

Creo Elements/Direct Drafting provides a wide range of printer or plotter output options. After you select an option, you can send plots and screen dumps either to the Windows Print Manager or to the Clipboard. Send plots to the Print Manager for output on the system's default printer or plotter. Send a plot to the Clipboard if you want to insert it in another Windows application.

Note

Creo Elements/Direct Drafting automatically derives the extension for the output file according to the selected printer or plotter.

Selecting a Printer or Plotter

To set a printer or plotter:

1. Click **File** ►  **Print** ►  **Print Setup**.

The **Print** dialog box opens. The **Printer** tabbed page opens by default.

Alternatively, press CTRL+P to open the **Print** dialog box.

2. In the **Printer** area, select the printer or plotter in the **Name** box.

Note the boxed plus sign beside some generic options.

-
- Expand the plus sign and select the model number, if required.
3. Click **Properties** and set the properties (such as the paper size and the resolution) of this printer or plotter.

If the printer or plotter you want to use is not included in this list, use standard Microsoft Windows techniques to install, set up, and configure this printer or plotter.

 **Note**

Refer to your Windows documentation and hardware manuals for information on configuring your printer or plotter to work with Windows.

Routing the Print or Plot Output to a File

To send the print or plot output to a file:

1. Click **File** ►  **Print** ►  **Print Setup**.

The **Print** dialog box opens. The **Printer** tabbed page opens by default.

Alternatively, press CTRL+P to open the **Print** dialog box.

2. In the **Printer** area, click the **Print to File** check box.
3. In the **Destination** box, type the file path and file name.

Ensure that you match the printer or plotter with the format of the output file. For example:

- If you want to create an HP-GL file, select an HP-GL compatible plotter.
- If you want to create a PostScript file, select a PostScript printer.
- If you want to create a PDF file, select the PDF Generic option.

After you click **Print**, the file is saved at the destination path with the specified name.

Stop on Error

If you want Creo Elements/Direct Drafting to warn you when drawings do not fit on the selected paper size, click the **Stop On Error** check box on the **Printer** tab of the **Print** dialog box. Creo Elements/Direct Drafting checks:

- Whether the size of the drawing fits on the selected paper format.
- When printing Annotation drawings, whether the largest one matches the chosen plot configuration. If not, Creo Elements/Direct Drafting warns you

with the Plot doesn't fit on paper message and you can stop the process.

Setting the Paper Size and Number of Copies

To set the paper size and number of copies for printing or plotting:

1. Click **File** ►  **Print** ►  **Print Setup**.
The **Print** dialog box opens. The **Printer** tabbed page opens by default.
Alternatively, press CTRL+P to open the **Print** dialog box.
2. In the **Paper** area, select the paper size in the **Format** box.
To set a custom paper size, select **Custom size** in the **Format** box and define the size of the paper.
3. If you have a multi-sheet drawing, click the **Use sheet format** check box to automatically set the format and orientation of each sheet based on the sheet size.
4. To set the number of copies, type or select the number of copies in the **Copies** box.

Setting Orientation

To set the paper orientation:

1. Click **File** ►  **Print** ►  **Print Setup**.
The **Print** dialog box opens. The **Printer** tabbed page opens by default.
Alternatively, press CTRL+P to open the **Print** dialog box.
2. In the **Orientation** area, click **Portrait** or **Landscape**.

Note

If you set the paper size such that the height and width are equal, the orientation is set to landscape by default.

Positioning Your Drawing

To position your drawing:

1. Click **File** ►  **Print** ►  **Print Setup**.

Alternatively, press CTRL+P to open the **Print** dialog box.

2. On the **Drawing** tab, in the **Position** area, click any of the following:
 - **Center** (default setting) to position your drawing at the center of the paper.
Creo Elements/Direct Drafting places the bounding box centered on the paper.
 - **Left** to position your drawing at the lower left corner prior to printing or plotting.
Creo Elements/Direct Drafting places the bounding box in the lower left corner of the paper.

You can specify an offset in the X direction and in the Y direction in the **X Offset** box and **Y Offset** box, respectively, for **Center** and **Left**. The print or plot output is moved accordingly in the X direction and/or Y direction.

These offsets are interpreted in terms of the current length unit (such as mm) set up for your Creo Elements/Direct Drafting environment.

Setting a Rotation Angle

To rotate the drawing output to the printer or plotter:

1. Click **File** ►  **Print** ►  **Print Setup**.

Alternatively, press CTRL+P to open the **Print** dialog box.

2. On the **Drawing** tab, in the **Position** area, type the angle of rotation in the **Rotation** box.

The angle of rotation is interpreted in terms of the current angular units set up for your Creo Elements/Direct Drafting environment.

Selecting the Drawing Range for Printing or Plotting

To select the drawing range for a print or plot:

1. Click **File** ►  **Print** ►  **Print Setup**.

Alternatively, press CTRL+P to open the **Print** dialog box.

2. On the **Drawing** tab, in the **Range** area, click any of the following:
 - **All** (default setting) to print or plot the entire drawing - irrespective of the current viewport contents.
 - Click **Sheet** to print each Annotation sheet on a separate page. See [Stop on Error on page 611](#) for information on a possible warning.
 - Click **Viewport** to print or plot the contents of the current viewport.

For example, if you are currently viewing a part, only the elements of that part contained within the window of the current viewport will be printed or plotted.

- Click **Box** to print or plot a specific area of a viewport.

Click **Pick** and then use your cursor in the drawing viewport to specify two opposite corners of a box. Only the contents of the boxed area will be printed or plotted. For example, suppose you are currently viewing a part, and the other parts or elements exist in the same area, but are not displayed. If you click **Box**, all the drawing elements existing in the boxed area (both displayed and hidden), are plotted.

Setting the Plot Scale

You can set a scale factor for output or automatically scale the drawing as large as possible within the limits of your paper.

To set the print or plot scale factor:

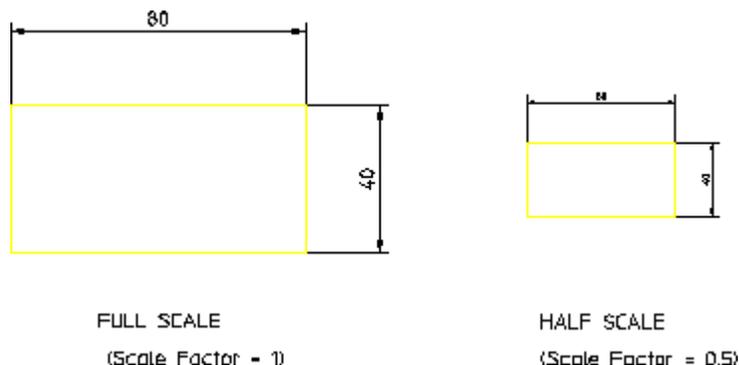
1. Click **File** ►  **Print** ►  **Print Setup**.

Alternatively, press CTRL+P to open the **Print** dialog box.

2. On the **Drawing** tab, in the **Scale** area, click any of the following:
 - Click **Fit** to scale the print or plot output (as large as possible) within the limits of the defined paper size.
 - Click **Factor** (default setting) and type the required scaling factor in the **Factor** box.

The entire print or plot output is scaled, but dimension values are not affected. [Figure 202. Operation of Print/Plot Scale on page 614](#) shows the operation of print or plot scale.

Figure 202. Operation of Print/Plot Scale



If you set the plotting scale such that the drawing is too large for the paper, Creo Elements/Direct Drafting warns you at the start of a plot by displaying the maximum scale possible. Pressing ESC aborts the plot and allows you to reset the plot scale. Otherwise, press SPACE BAR and the plot will continue, cutting off elements outside the paper format.

Setting the Content of the Print or Plot Output

Creo Elements/Direct Drafting enables you to specify what will be included in the print or plot output.

To set the content of the print or plot output:

1. Click **File** ►  **Print** ►  **Print Setup**.

Alternatively, press CTRL+P to open the **Print** dialog box.

2. On the **Drawing** tab, in the **Content** area, click any of the following:
 - **Drawing** to include all entities except construction lines in the print or plot output.

Note

Spotlights for the current part are omitted. The drawing appears as a normal drawing irrespective of the screen status (that is, the plot is unaffected by the **Show** functions).

- **Current Show** to print or plot the drawing as shown in the viewport.
 - a. Select a viewport in which you can set your own drawing conventions.
 - b. Use the **Show** functions to set the desired attributes.
 - c. Click **Current Show**.

The subsequent plots will contain whatever drawing conventions you have selected in the current viewport.

Note

Drawing and **Current Show** only determine how the plot appears. The plotting area is unaffected.

Defining Color Settings

Creo Elements/Direct Drafting allows you to define the **True Color**, **Plot Transformations** and **True Color & Screen Trans** settings for your printer or plotter environment.

To define the color settings of you drawing:

1. Click **File** ►  **Print** ►  **Print Setup**.
Alternatively, press CTRL+P to open the **Print** dialog box.
2. On the **Drawing** tab, in the **Colors** area, click **Pens**.
The **Plot Transformation** dialog box opens.
3. In the **Plot Transformation** list, click **True Color**, or **Plot Transformations**, or **True Color & Screen Trans**.

True Color

In the **Plot Transformations** list, select **True Color** to create a color plot of your drawing that is suitable for output to color output devices capable of producing color rastering.

This feature is available on PostScript Level 2 printers and HP-GL/2 printers. This means, for example, if you have access to an HP DesignJet 650C color printer you can print any number of colors up to the capability of the printer.

Note

If you have a drawing that contains elements in the color white and specify truecolor plotting, the elements are drawn in white. Normally, when sending drawings to a pen-plotter, the white elements on the screen are converted to black.

Plot Transformation

In the **Plot Transformation** list, select **Plot Transformations**.

The following settings are available:

Edit	Displays a separate Pen Format dialog box to set a custom plot transformation map.
Reset	Restores the default Plot Transformations settings.
Default Color	Applies a default color-to-pen assignment map.

Default B/W	Applies a monochrome color-to-pen assignment map.
Default GDI	Applies a 125 – color plot pen color-to-pen assignment map.

To set up the **Plot Transformations** map:

1. In the **Plot Transformations** map, click the row you want to change.
The selected row is highlighted.
2. Click **Edit**.
The **Pen Format** dialog box opens.
3. In the **Drawing** area of the **Pen Format** dialog box:
 - a. In the **Linetype** box, select a "from" linetype and then, in the **To** box, select a "to" linetype to which you want to map the "from" linetype. For example, if you want all dashed lines in the drawing to be plotted as dotted lines, map "from" dashed "to" dotted.

If you want to map all linetypes in the drawing to the associated linetypes on the plotter, click the **All Linetypes** check box.
 - b. Click the **Color Range** check box and then, in the **From (RGB)** box, type the RGB value for a "from" color. In addition, in the **To (RGB)** box, type the RGB value for a "to" color to which you want to map the "from" color.

If you want to map all colors in the drawing to the associated colors on the plotter, click the **All Colors** check box.
 - c. In the **Pen Size** box, type a "from" pensize and then, in the **To** box, type a "to" pensize to which you want to map the "from" pensize. For example, if you want pensize 0.3 in the drawing to be plotted by a 0.45 pen on the plotter, type 0 . 3 in the **Pen Size** box and 0 . 45 in the **To** box.

If you want to map all pensizes in the drawing to the associated pensizes on the plotter, click the **All Pen Sizes** check box.

Assigning a Specific Pen to a Given Linetype or Color Combination

To assign a specific plotter pen to a given linetype or color combination in the **Plot Transformations** map:

1. In the **Color** box of the **Drawing** area, select the color.
2. In the **Linetype** box of the **Drawing** area, select the linetype.
3. In the **Linetype** box of the **Plotted As** area, select the linetype to be plotted.

If you want to print or plot using the linetype that is used in the drawing, click the **Same Linetype** check box.

4. In the **Pen** box, type or select the pen number.
5. In the **Pen Width** box, type a pen width.

If you want to print or plot using the pen width that is used in the drawing, click the **Same Pen Width** check box.

6. Click **OK** to add the changed row to the **Plot Transformations** map.

For example, assume that you want to use a 0.3 mm blue pen to plot all dashed blue lines of the drawing. This pen is pen # 5 in your plotter.

1. In the **Linetype** box, select the dashed linetype and in the **Color** box, select **Blue** in the **Drawing** area of the **Plot Transformations** map.
2. In the **Plotted As** area, click the **Same Linetype** check box.
3. In the **Pen Width** box, type 0 . 3.
4. In the **Pen** box, type or select the pen number 5.

 **Note**

The difference between line width (pen size) and line size is explained in [Line Width vs. Line Size on page 95](#).

Repeat this procedure till the **Plot Transformations** map contains the required map of colors, linytypes, and associated plotter pens.

To restore the **Plot Transformations** default settings, click **Reset**.

To restore the default color settings, click **Default Color**.

Setting the Linetype Length

Creo Elements/Direct Drafting allows you to set the pattern length of the linetype for the plot or print using the **Plot Linetype Length** dialog box.

To open the **Plot Linetype Length** dialog box:

1. Click **File** >  **Print** >  **Print Setup**.

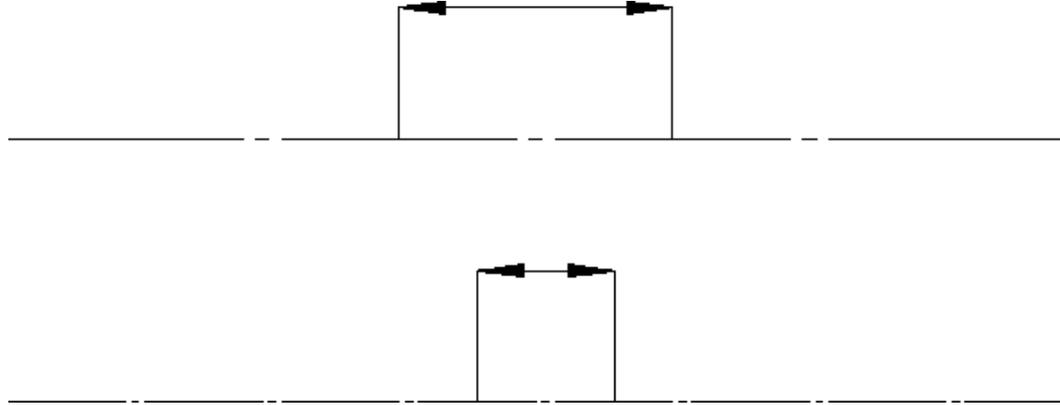
Alternatively, press CTRL+P to open the **Print** dialog box.

2. On the **Drawing** tab, in the **Colors** area, click **Pattern Length**.

The **Plot Linetype Length** dialog box opens.

You can set the pattern length for any of the seven broken linetypes. Each broken linetype is a pattern repeated many times to produce a line. You can specify the length of each pattern and thus alter the fineness of the line. [Figure 203. Different Linetype Lengths on page 619](#) shows two DASH_CENTER lines drawn with different linetype lengths. The arrow displays the linetype length.

Figure 203. Different Linetype Lengths



The seven broken linetypes that you can set are:

- DASHED
- LONG_DASHED
- DOT_CENTER
- DASH_CENTER
- PHANTOM
- CENTER_DASH_DASH
- DOTTED

To set the linetype length and minimum space for individual broken linetypes in the **Plot Linetype Length** dialog box:

1. In the **Linetype name** column, click the required linetype.
2. In the **Length** box, type or select a segment length for each broken linetype.
3. In the **Min. Dist.** box, type or select the minimum distance (the gap) between the broken line segments.
4. Click **Apply**.
5. Click **OK**.

Alternatively, to set all broken linetypes to the same segment length and the same minimum distance, specify the segment length and minimum distance and click **Apply to All** in the **Plot Linetype Length** dialog box.

Setting the Picture Color

Creo Elements/Direct Drafting allows you to set the output of the image to be either **Color** or **Black & White** when printing or plotting a drawing that includes embedded pictures.

To set the picture color:

1. Click **File** ►  **Print** ►  **Print Setup**.
Alternatively, press CTRL+P to open the **Print** dialog box.
2. On the **Pixmap** tab, in the **Pixmap** area, click **Color** or **Black & White**.

Retaining or Inverting Black and White Pictures

To reverse the black and white pictures:

1. Click **File** ►  **Print** ►  **Print Setup**.
Alternatively, press CTRL+P.
2. On the **Pixmap** tab, in the **B&W Pixmap** area, click **Reverse**.

Setting Picture Quality

To set the picture quality:

1. Click **File** ►  **Print** ►  **Print Setup**.
Alternatively, press CTRL+P.
2. On the **Pixmap**, in the **Pixmap Quality** area, click any of the following:
 - **Best** for high quality.
 - **Low** for draft quality.

Previewing a Print or Plot

Creo Elements/Direct Drafting allows you to view print or plot output on your screen using the **Plot Preview** system viewport. You can open the viewport after you select all the necessary print settings (as a last step before printing), or open it to view while working on your drawing.

1. Click **File** ►  **Print** ►  **Print Setup**.

The **Print** dialog box opens.

Alternatively, press CTRL+P to open the **Print** dialog box.

-
2. Click the **Show Preview** check box.

The **Plot Preview** system viewport appears.

3. Select a color setting for the **Plot Preview** viewport:
 - **Original** to preview color printer or plotter output.
 - **Grayscale** to preview black and white printer or plotter with dithering capability output.
 - **B&W** to preview black and white printer or plotter output.

The **Plot Preview** viewport is visible till you clear the **Show Preview** check box.

Starting a Print or Plot Job

When you have selected or specified all of the required print or plot output settings, click **Print** to send your drawing to either the Print Manager or to the Clipboard or to a file.

The Print Manager queues the drawing to the current default output device. The Clipboard temporarily stores the drawing so that it can be electronically pasted in another application.

Printing or Plotting Multiple Drawing Files

To plot or print multiple Creo Elements/Direct Drafting drawing files from the Creo Elements/Direct Drafting File Browser:

1. In the Windows file browser, select the drawing file to be printed or plotted and right-click.
2. Click **Print**.

Dumping the Creo Elements/Direct Drafting Screen

Creo Elements/Direct

To copy the entire Creo Elements/Direct Drafting screen to the Clipboard, press ALT & Print.

Then paste the screen dump in a graphics application of your choice for further processing. For example, in Microsoft® Paint, click **Edit ▶ Paste** to paste the Creo Elements/Direct Drafting screen dump to the drawing area.

Writing a Creo Elements/Direct Drafting Screen dump to an Output File

Optionally, you can write a screen dump of your current Creo Elements/Direct Drafting window directly to an output file.

See [Exporting a Picture on page 547](#) for instructions.

Further Information

For further technical information about plotting or dumping, refer to the Creo Elements/Direct Drafting help system.

Type `help plot`, or `help bitmap_to_file` in the Creo Elements/Direct Drafting user input line and press ENTER.

Creo Elements/Direct Drafting displays further information about plotting or dumping.

Parametric Design with Creo Elements/Direct Drafting Fluent User Interface

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Introduction

Roadmap

Read this chapter first. It provides an overview of the Parametric Design software, introduces several important terms, and directs you to the sections that describe how to start and use Parametric Design.

What is Parametric Design?

The Creo Elements/Direct Drafting Parametric Design module adds powerful parametric and constraint-based modeling capabilities to Creo Elements/Direct Drafting.

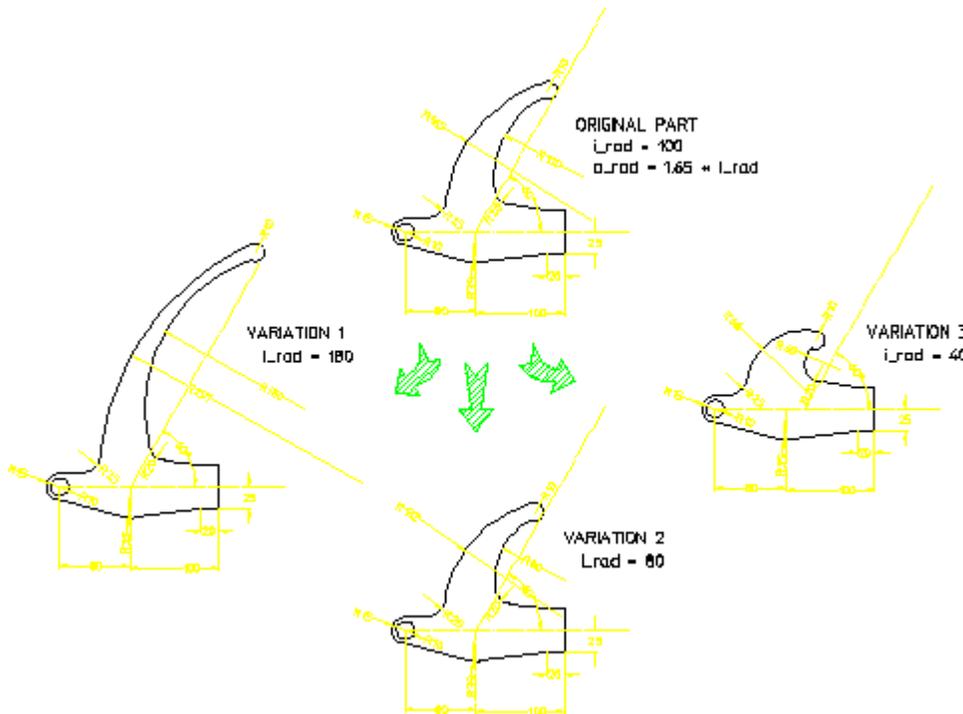
Parametric modeling allows you to attach "names" or parameters to drawing dimensions, such as radii or lengths, and then to change the drawing by changing the numerical values or expressions associated with these parameters.

Constraint-based modeling allows you to specify geometric properties and relationships. The software then modifies the part so that these constraints are satisfied. For example, you can say "these two elements should be tangent," or "this line should be horizontal," or "this dimension should have such-and-such a value."

Parametric Design combines both types of modeling into a module that allows you to quickly and easily perform modifications that are beyond the scope of the basic Creo Elements/Direct Drafting **Modify** toolset.

With Parametric Design module you can use any existing part as a template for generating variations or a family of parts that otherwise would have to be drawn from scratch. [Figure 204. Master Part Three Variations on page 625](#) below shows a "master" part and three variations that were generated with Parametric Design. Once constraints and parameters were defined for the master part with Parametric Design, each variation was generated by modifying only one dimension on the master.

Figure 204. Master Part & Three Variations



Parametric Design provides these capabilities without changing existing Creo Elements/Direct Drafting functionality or procedures. Loading the Parametric Design module does not change any existing Creo Elements/Direct Drafting menus or commands, so your drawing methods need not change to use it. You can use any existing Creo Elements/Direct Drafting part with Parametric Design, and parts generated by Parametric Design will be fully compatible with Creo Elements/Direct Drafting. The constraint and parametric information generated by the software is added directly to the master part (as info text), and is automatically stored/recovered along with the part.

Starting Parametric Design Module

To enable the **Parametric Design** module in Creo Elements/Direct Drafting:

- Click **File** ► **Modules** ►  **Parametric**.

Basic Working Methods

For the Draftsperson

Parametric Design gives the Draftsperson a simple, efficient interface for generating variations from pre-constrained parts. When a pre-constrained part is loaded in Creo Elements/Direct Drafting, all parameters that can accept value input appear in a table on this menu along with their current values. To generate variations of the part, the Draftsperson sets the appropriate values for any or all of these parameters and then uses the **Solve** group buttons on the **Parametric** tab. Parametric Design then generates the variation, which can be saved, copied, or modified further.

For the Designer

The Designer or Engineer uses Parametric Design to assign, evaluate, and modify constraints and parameters on that part. Once the constraints and parameters are in place, the **Solve** group buttons on the **Parametric** tab can be used to generate variations. The typical steps needed to constrain a part are as follows:

Preparation

Model inconsistencies can cause unexpected results in Parametric Design and should usually be resolved before assigning constraints to a part. The following buttons in the **Modify** group on the **Parametric** tab check parts for inconsistencies such as extra points and duplicated or stacked geometry; and optionally adjusts or removes the offending elements:

-  **Clean Pnt**
-  **Clean Duplicate**
-  **Clean Stacked**

Parametric Design operates on drawing elements within its zone. The zone may contain an entire part or selected elements of a part. The buttons under  **Zone** in the **Modify** group on the **Parametric** tab select elements for the zone.

Assign Constraints

Before you can use Parametric Design to generate variations from a part, constraints must be assigned to the part. These constraints must be sufficient to tell the parametric solver exactly what to do with each element in the zone to generate the variation. If elements are not constrained or are incorrectly constrained, the solver is not able to generate the variation.

Constraints may be assigned manually or automatically. Typically, the designer uses a combination of methods as follows:

1. Assign a full set of constraints that completely define the part's current configuration:
 - a. Manually assign a few reference constraints and perhaps symmetry line constraints to important elements in the part.
 - b. If large sections of the part's geometry can move together as a single unit, collect these sections into rigid bodies using  **Rigid** in the **Create** group on the **Parametric** tab. This can greatly reduce the number of constraints needed on the part.
 - c. Use **Complete** to automatically assign a reasonable set of constraints to the remaining elements.
2. If you used the **Solve** group buttons on the **Parametric** tab to generate variations, check the potential result or results by means of  **Preview**. You may need to modify certain constraints to better reflect your design intentions.
3. Modify constraints as needed to generate the desired variation from the current configuration. You may add or delete constraints or change the values associated with existing constraints to come up with the variation you want.

To manually assign or modify constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select a constraint type (for example, **Angle**, **Dimension**, **Size**).
4. Click **Apply**.
5. Select the drawing elements to receive the constraint.

Iconic call-outs are used to label constraints as they are assigned.

Angle, distance, and dimensional constraints, among others, may be given parametric as well as numerical values when you assign them. A parametric value may be a text label, a Creo Elements/Direct Drafting expression, or a macro name which is resolved at solve time to generate a numeric value for the constraint. Parameters can perform math functions and can reference other parameters when they are resolved. You are not required to use parametric constraints, but they are an especially powerful feature of the software.

Solve for the Variation

After you have constrained your part, use the **Solve** group buttons on the **Parametric** tab to solve the set of constraints and generate the variation. Parametric Design attempts to create new geometry that fits the constraints by adjusting sizes, locations, and relationships of elements in the part. If the current set of constraints is inconsistent or incomplete, the solver can selectively highlight the offending areas. A variation can be previewed before actually being generated, and it can be generated either as a copy of or as a replacement for the original geometry.

Learning to Use Parametric Design

The best way to learn Parametric Design is to work with the software and consult this guide as needed for clarification. The [Examples on page 715](#) chapter provides several work-along examples that we recommend you to try. To encourage you in this, we have included the parts used in the examples along with the Parametric Design software. You need only load the parts into Creo Elements/Direct Drafting to begin working with the examples.

Generating Variations

Roadmap

Read this chapter to learn how to generate variations from parts that have already been constrained and parameterized with Parametric Design. To learn how to assign constraints and parameters, go on to [Working with Parametric Design on page 633](#) after you read this chapter.

Parametric Design provides an efficient interface for generating new parts (variations) from a master part that has already been processed through Parametric Design. Once you load a master part into Creo Elements/Direct Drafting, you need only supply a few parameter values and use the **Solve** group buttons on the **Parametric** tab to generate a particular variation. Again, the basic working method for generating a variation of a pre-processed master part is:

1. Load the master part.
2. Click **Parametric** and then, in the **Show** group, click **Current**.

The **Current Constraints** window opens.

3. Change parameter values as required in the parameter value table.
4. Use the **Solve** group buttons on the **Parametric** tab to generate the variation.

The following sections describe each step.

Load the Master Part

A master part can be any Creo Elements/Direct Drafting part that has Parametric Design constraint and parameter data associated with it. This data is generated by the Designer using Parametric Design and is automatically stored along with the part when it is saved to a file. Parametric Design data is loaded back into Creo Elements/Direct Drafting whenever you load the master part with the standard Creo Elements/Direct Drafting **Load** functions.

Parametric Design operates only on the current Creo Elements/Direct Drafting part. If you load a drawing that contains several parts, ensure that the part you wish to use as the master is the current Creo Elements/Direct Drafting part. Use

the Creo Elements/Direct Drafting  **Part** button under  **Edit** in the **Parts** group on the **Structure** tab to select a particular part to be current.

To confirm that the part you have loaded has the necessary Parametric Design data associated with it to be a master part, check the parameter value table in the **Current Constraints** dialog box. If no entries appear in the table after you load a part or make it the current part, it cannot be used as a Parametric Design master from this menu.

List the Current Constraints

To create variations from a master part, open the **Current Constraints** dialog box

by clicking  **Current** in the **Show** group on the **Parametric** tab. Then click **Advanced** to open the **Advanced** window that displays the parameter value table. When you load a master part into Creo Elements/Direct Drafting, one or more entries appear in this table. Each entry contains a parameter name and a value. The parameter names correspond to elements in the master part that you can modify. The value entries represent the values that will be assigned to the corresponding drawing elements when you next use the **Solve** group buttons on the **Parametric** tab.

Change Parameter Values

Use the parameter value table to specify how variations of the master part should be generated. When you change the value for a parameter listed in the table, you indicate Parametric Design to modify all elements in the master part that are constrained to depend on this parameter. The constraints on the part determine exactly what modifications will occur. The changes are not actually made until you click the required button in the **Solve** group on the **Parametric** tab.

To change the value for a parameter, simply select its entry from the table and enter the new value from the keyboard. If there are too many entries to view at once, use the scroll bar beside the table. The type of value you give (linear, angular, or point) depends on the type of the parameter. Usually, the parameter

name is descriptive enough to indicate what type of value is expected. It is easy to tell if a point value is expected, because point parameters take up two slots in the table. The upper slot contains the x coordinate of the point and the lower slot contains the y coordinate. Note that you still enter point values using the standard Creo Elements/Direct Drafting x,y format.

If you need more specific information about the correspondence between parameter entries in the table and the elements in the master part, click **Show** in the **Current Constraints** dialog box. When you click **Show**, elements in the master that have parameters assigned to them are highlighted or marked with icons as described below.

If a parameter is associated with a dimension in the part, the parameter name is appended to the corresponding dimension text and both are highlighted (in CYAN by default). By examining the dimension, you can tell whether the parameter requires linear or angular input.

Parameter names may also be associated with several kinds of geometric constraints that have been assigned to the master part by a designer. If any of the parameters in the value table are associated with geometric constraints and you click **Show**, these constraints are marked with icons as well as with the parameter name. Each type of geometric constraint has a unique icon to identify it. Icons and parameter labels are drawn in CYAN by default. Your current text font is used to display the parameter names. Consult [Constraint Types on page 655](#) for complete information on geometric constraints, their icons, and the types of values to be supplied to each.

The **Show** highlighting is removed whenever you click the **Solve** group buttons to generate a variation.

Alternatively, to remove the highlighting:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. Click **Clear**.
3. In the **Type** box, select **All Types**.

Note

If the icons produced by **Show** are too large or too small, you can resize them using the **Icon Setting** dialog box.

Solve for the Variation

Once you have filled the parameter value table, use  **Preview**,  **Keep**, or  **No Keep** in the **Solve** group on the **Parametric** tab to create a variation with the desired values.

Preview

By default, the solver works in  **Preview** mode. The variation is computed, but is not actually created. Instead, an outline of the variation is superimposed over the current part. This gives you a good idea of what the new variation will look like without creating any new geometry or overwriting any existing geometry. If the preview indicates that the new variation is acceptable, you can actually create the new variation by re-solving with  **Keep** or  **No Keep**. The preview lines drawn by  **Preview** are removed when you click  **OK**.

Keep

 **Keep** generates the variation and inserts it into the current part as a modified copy of the original part. When you click  **Keep**, the solver computes the variation and temporarily overlays it onto the current part so that you can see what it looks like. If the variation is not what you want, click  **OK** or  **Undo** immediately to restore the original part. If the variation is OK, click a reference point for moving the new part, and then drag it to the desired location. After you've inserted as many copies as you want, click  **OK**. The overlaid variation is removed, and your original part is now restored.

Depending on how the master part was originally constrained, elements of the master such as construction geometry and dimensioning may not appear in copies made with  **No Keep**. Also, the copies are not normally affected by subsequent use of the **Solve** group buttons on the **Parametric** tab. To change any of these behaviors, you will most likely need to modify the zone memberships and/or constraints in the master part. See [Working with Parametric Design on page 633](#) for details.

No Keep

Use  **No Keep** if you want the solver to replace your current part with the variation. If the result of the solve is not what you expect, click  **Undo** immediately to restore your original part. You can continue to modify the part by entering new values into the parameter table and re-solving.

Saving and Restoring the Parameter Value Table

In the **Current Constraints** dialog box, click **Save** and **Input** to write out the current parameter value table to a file and later restore it. This feature lets you build a "library" of value tables that can be used to drive the master part. The advantage of this is that a series of standard parts based on a single master part can be stored as a single MI file (the constrained master part) and a collection of parameter value files. This arrangement takes up much less disk space than if an individual MI file were stored for each part in the series. To recreate any member of the series, you need only load the master part, **Input** the desired parameter value table, and **Solve**.

Note

When the parameter value table is written out, all Parametric Design parameters for the master part are written to the file. A number of these parameters may be expressions, which are used by the software, but do not appear in the table. Don't modify or remove these parameters from the output file!

Save the Parameter Value Table

To save the current parameter value table to a file:

1. Click **Save** in the **Current Constraints** dialog box.
The **Store** dialog box opens.
2. In the **Name** box, type the name of the file to write out at the prompt.
3. Click **Save**.

Restore a Parameter Value Table

To input a parameter value table:

1. If necessary, **Load** the master part that the value table file refers to.

To ensure that the master is the current part:

-
- a. Click **Structure** and then, in the **Parts** group, click  **Edit**.
 - b. Click  **Part**.
2. Click **Input** in the **Current Constraints** dialog box.
The **Load** dialog box opens.
 3. In the **Name** box, type or select the name of the parameter value file to input.
 4. Click **Open**.
 5. To apply the values from this table to the part, click **Solve**.

Creating a Parameter Value Table by Hand

When you save a parameter value table, the file generated is an ASCII macro file that contains one `PD_PARAM_FIX` command for each entry in the table, e.g., `PD_PARAM_FIX 'Width' 37.5`. It is relatively easy, therefore, to build your own parameter value tables outside of Creo Elements/Direct Drafting using an ordinary text editor. When writing a value table file by hand, keep the following points in mind:

- It may be very helpful later on if you include comment lines in your file that identify the master part this file applies to!
- Make sure that each `PD_PARAM_FIX` command in your file corresponds to a value parameter in the master part. If in doubt, save a parameter value table with Parametric Design and use this file as a template for your own files.
- When Parametric Design saves a parameter value table, it includes several macros in the output file which ensure that the values are restored in the correct units. When you create this file by hand, you are responsible for ensuring that the units used in the file correspond to the current Creo Elements/Direct Drafting units when you restore it. One way to do this is to include the macros used by Parametric Design in your own files (use any system-created value table file as a template).

Working with Parametric Design

Roadmap

Read this chapter to learn how to assign constraints to a Creo Elements/Direct Drafting part so that you or others can generate variations from it. If you only need to know how to generate variations from a pre-constrained part, skip this chapter and read [Generating Variations on page 628](#) instead.

This chapter is for Designers or Engineers who wants to incorporate Parametric Design capabilities in their Creo Elements/Direct Drafting design environment. We begin by describing the Parametric Design tools that the Designer should be familiar with:

- Part preparation tools
- Constraint assignment tools
- Solving tools

The chapter concludes with several sections that cover the major applications for these tools: developing master parts for repeated use by yourself or others, and using Parametric Design as an extension of the Creo Elements/Direct Drafting **Modify** commands to do one-time adjustments of complex parts.

Part Preparation

Before you assign constraints to a part, two preparation steps should be considered. The first is to check the part for inconsistencies that can interfere with constraint generation. The second is to select or restrict the portions of the part to work with.

The following buttons on the **Parametric** tab handle preparation tasks:

-  **Clean Pnt** (in the **Modify** group)
-  **Clean Duplicate** (in the **Modify** group)
-  **Clean Stacked** (in the **Modify** group)
-  **Rigid** (in the **Create** group)
- **Zone** group buttons
-  **Zone** (in the **Show** group)

Cleanup

Inconsistencies in a part (lines not quite parallel, small gaps between "tangent" objects, duplicate elements, etc.) are relatively common, but can cause problems for any program that must scan the part to extract information. To help eliminate drawing inconsistencies, Parametric Design provides several tools that can do

limited "cleaning" of parts. Use  **Clean Pnt**,  **Clean Duplicate**, or  **Clean Stacked** to clean the parts.

These buttons are functionally independent from the remainder of the Parametric Design software, and should be thought of as a "preprocess" for parts that are submitted for parametric design. The **Complete** command and the **Solve** group

buttons have additional part cleaning capabilities which are described in detail in [Advanced Topics and Tips on page 698](#). Each of the buttons described below affects all geometry in the current part.

Points

Model points within a given distance of each other are merged, and the elements attached to the merged points are adjusted (moved, stretched, or rotated) as necessary to reflect the new positions of the points. For each set of points to be

merged,  **Clean Pnt** chooses a location within the given tolerance and moves the points there. In general, there is no way to accurately predict or control which elements will be adjusted as a result of the merge. This should not be a limitation

in most cases, however, because the tolerance used is likely to be small.  **Clean Pnt** is most useful for removing small gaps between elements.

To use  **Clean Pnt**:

1. Click **Parametric** and then, in the **Modify** group, click  **Clean Pnt**.
2. Type a distance tolerance (current length units) in the user input line and press ENTER.

Points within the given tolerance are marked with a *, and will be merged. Elements that will be modified as a result of the merge are highlighted.

3. Click  **Confirm** to merge the marked points and modify the highlighted elements.

Duplicate

 **Clean Duplicate** scans the part for duplicate elements and deletes them. The button selects which element of a duplicate pair to delete. Elements are considered to be duplicates if their model points are identical. An exception is made for concentric circles (only): if the radii of concentric circles are within a given

tolerance, they are treated as duplicates. Normally,  **Clean Duplicate** should be

used after  **Clean Pnt** to clean up duplicate elements that may result from merging model points.

To use  **Clean Duplicate**:

1. Click **Parametric** and then, in the **Modify** group, click  **Clean Duplicate**.
2. Type a distance tolerance (current length units) for concentric circles in the user input line and press ENTER.

Elements that will be deleted are highlighted.

3. Click  **Confirm** to remove duplicate elements.

Stacked

 **Clean Stacked** finds stacked lines and arcs in the current part and calls  **Split** to split them.

To use  **Clean Stacked**:

1. Click **Parametric** and then, in the **Modify** group, click  **Clean Stacked**.

Elements to be split are highlighted and the points to be used for the splits are marked with a *.

2. Click  **Confirm** to split elements.

Note

To clean up most common problems in a part, click Clean buttons in the following sequence:

 Clean Pnt	Merge disconnected points.
 Clean Duplicate	Remove existing duplicates and those caused by merging points.
 Clean Stacked	Split overlapping elements.
 Clean Duplicate	Remove duplicates caused by splitting.

Setting the Zone

The Parametric Design **Solve** commands and  **Automatic** operate only on the current Creo Elements/Direct Drafting part, and only on the portion of that part which is included in a zone that you define. The zone mechanism allows you to apply parametric design to sections of a part, leaving the remainder untouched. By including only those sections that require parametric design in the zone, you can often significantly reduce the number of constraints you need to assign as well as shorten **Solve** times.

The zone is implemented via Creo Elements/Direct Drafting info text. The buttons under  **Zone** in the **Modify** group on the **Parametric** tab are used to add or remove the info text PD_ZONE to geometry elements in a part. Parametric Design will only operate on elements that have the PD_ZONE info text. Sections of the part that do not have this info fall outside the zone and are never modified by the **Solve** group buttons, remaining fixed in place.

When you start Creo Elements/Direct Drafting, the zone is empty. By default, elements you **Create** during an Creo Elements/Direct Drafting session are automatically added to the zone, but geometry you **Load** from unconstrained files is not included. The buttons under  **Zone** in the **Modify** group on the **Parametric** tab add and remove elements in the zone.

Adding Elements to the Zone

To add elements to the zone:

1. Click **Parametric** and then, in the **Modify** group, click  **Zone**.
2. Click  **Add Single** or  **Add Multiple**.
3. Select the geometry to be included in the zone.

 **Add Multiple** extends your selection to include all elements that share model points with the elements you select. The Creo Elements/Direct Drafting **Select** mechanism can be used with either button.

4. Click  **OK** to conclude.

Elements are highlighted as you add them to the zone.

Note

 **Add Multiple** and  **Add Single** do not add dimensioning or subparts in the master part to the zone. See [Zone Gymnastics on page 703](#) for additional information on adding to the zone.

Showing Contents of the Zone

To display the current contents of the zone:

1. Click **Parametric** and then, in the **Show** group, click  **Zone**.



Zone redraws all elements in the zone with the current highlight color and linetype.

2. Click  **OK** to remove the highlighting.

Removing Elements from the Zone

To remove elements from the zone:

1. Click **Parametric** and then, in the **Modify** group, click  **Zone**.
2. Click  **Delete**.
3. Select the elements that you want to remove.

If the elements have constraints attached to them, the constraints are not removed, but the elements will not be modified by subsequent use of the **Solve** group buttons on the **Parametric** tab.



Note



Delete only removes the elements from the zone that you click directly and not elements that share model points with those you select. In other words,



Delete is analogous to  **Add Single** rather than  **Add Multiple**.

Defining Rigid Bodies

A final preparation step that can often help simplify the design process is to identify areas of the part in which the geometry will not change in size or relative position, but which should be able to move (translate, rotate) as a unit. By defining these areas as rigid bodies, you can significantly reduce the number of constraints needed, and make the design process more predictable.

A rigid body is a named collection of geometry that acts as if it is sufficiently constrained to prohibit any relative change of size or position among the elements collected into it. The geometry collected into a rigid body can only be translated and/or rotated as a whole. To link a rigid body with other elements in a part, you can assign constraints between these elements and any element inside the rigid body.

Most Creo Elements/Direct Drafting geometry elements as well as hatching and subparts can be included in a rigid body. Like zones, rigid bodies are implemented via Creo Elements/Direct Drafting infos; entities with the info text `PD_RIGID` are linked into rigid bodies. For more in-depth information on rigid body applications see [Advanced Topics and Tips on page 698](#).



Rigid allows you to define and manipulate an unlimited number of rigid bodies inside a given part. The options in the **Generate Rigids** dialog box are:

- **Assign**
- **Show**
- **Clear**

Creating a Rigid Body

To create a new rigid body:

1. Click **Parametric** and then, in the **Create** group, click  **Rigid**.
The **Generate Rigids** dialog box opens.
2. Type a name for the new rigid body in the text box.
Ensure that you select a name that is not already in the rigid body list in the **Generate Rigids** dialog box if you want to create a new rigid body.
3. Click **Assign**.
4. Select the geometry to be included in the rigid body.
Elements are highlighted as you add them to the rigid body. When you make the first selection, the name of the new rigid body is added to the table.
5. Type the name of another rigid body in the user input line and press ENTER or click  **OK** to conclude.

Showing and Clearing Rigid Bodies

To show or clear an existing rigid body:

1. In the **Generate Rigids** dialog box, select the rigid body you want to show or clear.
If there are too many entries in this table to view at once, scroll the table to bring the correct entry into view.
2. Click **Show** or **Clear** to highlight or remove the rigid body from the list, respectively.
3. Click  **OK**.

Assigning Constraints

Constraints can be assigned manually or automatically to elements in the zone. The constraint assignment tools include:

- Commands:

Command	Function
Assign	Manually assign constraints to geometric elements.
Free	Remove constraints from geometric elements.
Show	Show existing constraints by displaying their icons.
Clear	Remove selected constraint icons (but not the constraints themselves) from the display.
Ban	Remove selected constraints and prevent them from being re-assigned by Complete .
Complete	Automatically generates a reasonable set of constraints to all remaining elements in the zone.

- Options:

Options limit the actions of **Show**, **Clear**, **Free** and **Ban**. Available options are:

Option	Limiting Action
New	Act only on constraints added by the most recent Complete command. New must be used immediately after Complete . Any intervening command converts new constraints to "system" constraints. New only modifies the actions of Show and Clear .
User	Act only on constraints assigned manually by the user. If the user modifies a system-generated constraint, it converts to a "user" constraint.
System	Act only on constraints generated by Complete and unchanged by the user.
Used	Act only on constraints actually used by the solver during the most recent Complete or Solve command.
Unused	Act only on constraints that were not used by the solver during the most recent Solve command.
Banned	Act only on constraints that have been banned by the user; i.e., constraints that cannot be assigned by Complete .
Violated	Act only on constraints marked "violated" by the most recent Solve .

- Types:

These blocks specify the available constraint types. You must always specify a type for **Assign**, **Show** and **Clear**. Types can also be used to limit the actions of **Free** and **Ban**. A generic type, **All Types**, refers to all the available constraint types. See [Constraint Types on page 655](#) for a complete description of each type.

The following sections describe each of the constraint assignment commands.

 **Note**

The default size and positioning of constraint icons may not be ideal. You can resize or change the color of icons via the **Icon Setting** dialog box and reposition them with  **Move Icon**. To update the icon display after changing their size or position, you can use the Creo Elements/Direct Drafting  **Redraw** button. The text labels that appear on some icons are drawn in the current Creo Elements/Direct Drafting text font.

Assign

Use **Assign** to manually assign constraints to elements.

To assign a constraint:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select a constraint type (for example, **Angle**).
4. Click **Apply**.
5. Select the required geometry for the constraint type.
6. If applicable, type a value or parameter name for the constraint in the user input line and press ENTER.
7. Continue to select geometry, select a new constraint type, or click  **OK**.

When you successfully assign a constraint, one or more icons appear on the part to identify the type and identification number of that constraint. The leader line of the icon is drawn in the same linestyle as the element it references.

 **Note**

When you use **Assign** to assign a constraint to a piece of geometry, the constraint is bound to that element and goes with it whenever it is moved. Thus, you can move elements in a part after assigning constraints without invalidating the constraints. On the other hand, if you delete a constrained element, all constraints on that element are removed along with it.

Some constraint types can be given numeric or parametric values when they are assigned. If you give a numeric value, the constraint will resolve to that value at solve time. If you give a parameter name or expression, the constraint resolves to the output of that parameter or expression at solve time. If you choose not to supply either a value or parameter, the constraint resolves to the corresponding value that is currently on the element at solve time. If a value or parameter name is assigned to a constraint, that data will appear along with the constraint's icon whenever it is displayed. See [Parameters on page 679](#) for a full discussion of how numeric values and parameters affect constraints.

Show and Clear

To assign and free constraints efficiently, it is important to know what constraints are currently assigned. **Show** and **Clear** work with the limit-options and constraint-type settings to selectively display and hide the icons and parameter labels of constraints currently on the part. Neither function affects the constraints themselves, just the display of their icons. Icons and parameter labels are displayed in the current show color (`PD_PREVIEW_COLOR`) and size. Parameter labels will use each dimension's text font for dimensional constraints and the current text font for geometric constraints.

Show and **Clear** operate identically. To use either:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Show** or **Clear**.
3. In the **Act On** box, select an option to limit the function's effect, if desired.
4. In the **Type** box, select the constraint type to show or clear.

Show options are additive; subsequent calls to **Show** do not erase the icons that are already on the screen, allowing you to build up the display. From time to time, as the display gets crowded, you can erase some or all of the icons with **Clear**.

The following table provides some **Show** examples:

Show all constraints:	Show, All Types
Show constraints set by Complete :	Show, System, All Types
Show reference elements set by the user:	Show, User, Refelem
Show angle and distance constraints:	Show, Angle, Show, Distance
Erase all constraint icons:	Clear, All Types

Free

Free removes constraints from elements. To remove constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Free**.
3. Use one of these methods to select constraints to free:
 - Click the icon of the constraint to be freed.
 - In the **Type** box, select a constraint type and then click the element itself.
 - Select a limiting option in the **Act On** box and then a constraint type in the **Type** box to remove the selected constraint from all elements in the zone. For example, **Free**, **User** and **Dimension** removes all user-assigned dimensional constraints.
4. Click  **OK** or continue to select constraints for removal.

Whenever you free a constraint, the corresponding constraint icon is removed from the display. If you free a constraint by accident, an immediate  **Undo** restores it.

Freeing a constraint does not prevent it from being re-assigned to the master by **Complete**. If you want to prevent a constraint from being re-assigned by **Complete**, use **Ban** instead.

To remove all constraints generated by **Complete**, click **Free**, select **System**, and select **All Types** in the **Action** area, **Act On** box and **Type**, respectively.

To quickly remove all constraint and parametric data from the current part, use click **Free** in the **Action** area and select **All Types** in the **Type** box. This command requires a  **Confirm**.

Complete

In order for the **Solve** group buttons to be able to generate a variation of a part, every element in the part must be completely constrained. To manually assign a complete set of constraints to a part of even moderate complexity would be time consuming and prone to error. The **Complete** command provides an efficient alternative. **Complete** scans the geometry currently in the zone and generates a reasonable set of constraints that is sufficient to completely constrain the geometry in its current configuration. In many situations, only a few constraints need be set manually before running **Complete**, and a few more changed after **Complete** is run to generate the variation. The [Strategy for Assigning Constraints on page 644](#) section below explains how to best integrate the automatic and manual options.

To use **Complete**:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.

The **Generate Constraints** dialog box opens.

2. In the **Action** area, click **Complete**.
3. Click **Apply**.

Parametric Design generates constraints. A series of messages is displayed to inform you about the command's progress.

While generating constraints, **Complete** may add a number of point elements to your part. These are needed to fix point-on or reference-point constraints, and should not be removed from the part. By default, these points are drawn as green ×s, but you can easily change this to make them nearly invisible. See [Parametric Design Defaults on page 761](#) for details.

Complete takes into account any constraints that you may have manually assigned to the part. If these constraints are inconsistent with one another, **Complete** may not be able to successfully generate a complete set of valid constraints. In this case, error messages and highlighting warn you about the problem and help you track the cause. The error-trapping mechanism used by **Complete** is also used by the buttons in the **Solve** group. See the [Solver Errors on page 648](#) section later in this Chapter for more information.

Ban

Ban works exactly like **Free** except that in addition to removing a constraint, **Ban** prevents that constraint from being re-assigned by subsequent **Complete** commands. Essentially, **Ban** forces a constraint to remain unset. To re-assign a banned constraint, you must assign it manually with **Assign**.

Strategy for Assigning Constraints

Complete and the manual constraint tools are designed to be used together to efficiently generate a workable set of constraints that lends itself to modification. The following strategy will produce the best results in most situations:

1. Before running **Complete**, manually assign sufficient constraints to:
 - "Anchor" the part. Do this by assigning **Refelem** or **Refpoint** constraints to one or more elements that you know should not change position in the variation. Anchoring helps to give **Complete** a frame of reference for generating new constraints.
 - Identify symmetry lines. Assign **Symmline** constraints to any elements that serve as symmetry lines in the part. **Complete** uses these lines to generate mirrored-element constraints. Parametric Design can also be configured to

automatically recognize all lines with a particular color and linetype as symmetry lines. You enable this capability by modifying the `pd_def.mac` file. See [Parametric Design Defaults on page 761](#) for details.

- If the part has several distinct components or views, make sure to anchor each view sufficiently to prevent them from interfering with each other in subsequent operations. You can do this either by assigning absolute constraints (**Refelem**, **Refpoint**) to each view, or by constraining the relative distance and/or orientation among all views with **Distance** or **Angle** constraints.
- Collect suitable elements into rigid bodies whenever possible.
- Manually constrain any elements whose position or orientation is critical in subsequent variations. For example, you should fix in place any elements that definitely should not move in the variation. Alternately, you could remove these elements from the zone before running **Complete**.
- In general, the more constraints you can assign by hand, the better for **Complete**. This is especially true if the part is not fully dimensioned.

 **Note**

Do not manually assign constraints designed to move elements away from their current location before running **Complete**. For example, don't assign a **Vertical** constraint to horizontal lines in the part before you use **Complete**. Such constraints make the extraction process much more difficult.

2. Run **Complete** to generate the remaining constraints needed to fully constrain the master part in its current configuration.
3. When **Complete** finishes, examine the constraints on the part to make sure that **Complete** hasn't made any poor decisions about assigning constraints. In particular:
 - Check dimensions. Make sure that constraints were actually assigned to any dimensions that you might have to change. If not, add new constraints. Also, be aware that different types of dimensions can cause **Complete** to behave in different ways. For example, a horizontal dimension between two points is quite different from a parallel dimension between the same two points, even if both have the same value. The first case implies that the second point is on a vertical line a given distance away from the first point. The second case implies that the second point is on a circle of a given radius centered at the first point. **Complete** takes these distinctions into

account when generating constraints. When in doubt about the type of linear dimension, redefine it.

- Check distance and size constraints. Distance constraints are usually assigned as a last resort to position disconnected geometry that is insufficiently dimensioned. The elements that receive distance constraints may not be the ones you would normally choose to receive them. Similarly, size constraints are normally only assigned to geometry that is connected to other elements, but is still under-dimensioned. Again, a size constraint may not be an appropriate choice.
- Check angle and slope constraints. Are elements with these constraints supposed to be fixed at the given angle?
- Check the constraints on rigid bodies. Ensure that **Complete** hasn't over-constrained each rigid body with regard to the rest of the part.

 **Inq Elem** and  **Inq Icon** in the **Show** group on the **Parametric** tab are especially useful for examining constraints generated by **Complete**.

 **Inq Elem** shows how the most recent **Complete** command or the buttons in the **Solve** group determined the size and position of an element that you select.

When you click  **Inq Elem** and then click an element, the icons for all the constraints that determined the size and position of that element are displayed. In addition, all geometry on which those constraints rely have their geometry highlighted.

 **Inq Icon** identifies the element or elements associated with a given constraint by highlighting the element or elements. You identify the constraint by clicking its icon.

4. If you make changes to geometry or constraints at this stage, re-run **Complete**. Check the following points any time you re-run:
 - If you removed any constraints, did you **Free** or **Ban** them? **Complete** is likely to re-assign the same constraints unless you used **Ban**.
 - If you added dimensioning to your part to eliminate the need for size or distance or angle constraints, did you **Free** the old system constraints? **Complete** will not automatically free existing system constraints and use the dimensioning to generate new constraints. If you **Free** the system constraints first, then the new dimensioning is taken into account.
5. Once you are satisfied that the part is completely and correctly constrained in its current configuration, start to modify those constraints that will be used to generate the variation or variation to ensure that the part responds as you intend.

Solve

All access to the Parametric Design solver is through the  **Preview**,  **Keep**, and  **No Keep** buttons in the **Solve** group on the **Parametric** tab.

When you click a button in the **Solve** group, the solver finds all the geometry currently in the zone, and evaluates all constraints on that geometry. If all the constraints can be successfully evaluated, the solver generates the variation specified by the constraints. Depending on the option selected, the solver can either create geometry for a new variation or simply preview it. The sections below describe the buttons in the **Solve** group.

Preview

By default, **Solve** works in  **Preview** mode. The variation is computed, but is not actually created. Instead, an outline of the variation is superimposed over the current part. This gives you a good idea of what the new variation will look like without creating any new geometry or overwriting any existing geometry. If the preview indicates that the new variation is acceptable, you can actually create the new variation by re-solving with the  **Keep** or  **No Keep** buttons. The preview lines are drawn in the current PD_PREVIEW_COLOR color (MAGENTA by default). Preview lines are removed from the viewport when you use the next instance by clicking the buttons in the **Solve** group or when you click  **OK**.

Keep

 **Keep** generates the variation and inserts it in the current part as a modified copy of the original part. When you click  **Keep**, the solver computes the variation, temporarily overlays it on the current part, and executes an operation as conducted by  **Multiple** (under  **Move** in the **General** group on the **Modify** tab) to allow you to copy the variation.

To use  **Keep**:

- Click **Parametric** and then, in the **Solve** group, click  **Keep**.

When the solver generates the variation and overlays it on the current part, evaluate the results. If the variation is not what you want, click  **OK** or  **Undo** immediately to restore the original part. If the variation is OK, click or specify a reference point for moving the new part, and then drag it to the desired

location. Repeat this step until you have inserted as many copies as you want.

Finally, click  **OK**. The overlaid variation is removed, and your original part is restored.

Only elements in the zone (that is, elements with the PD_ZONE info text) at solve time are copied by  **Keep**. Dimensioning and hatching elements are never put into the zone by Parametric Design commands. Therefore, these elements are not normally included in the copies you make with  **Keep**.

Copies made with  **Keep** are not put in the current zone and do not have constraints assigned to them. The original part from which copies are generated is unaffected by  **Keep**.

No Keep

Use  **No Keep** if you want the solver to replace your current part with the variation.

To use  **No Keep**:

- Click **Parametric** and then, in the **Solve** group, click  **No Keep**.

If the result of the solve is not what you expect, click  **Undo** immediately to restore your original part. You can continue to modify the part after solving with  **No Keep** by assigning new values to the constraints and re-solving.

Solver Errors

The solver or automatic constraint generator will fail without generating a complete solution if incomplete or inconsistent constraints exist on a part. Solver errors can result from any of the following conditions:

- The solver has successfully evaluated all the constraints it could find, but some elements cannot be resolved. More constraints are needed in order to completely constrain the part and generate a variation.
- A constraint has been violated while determining the new geometry. This is usually due to contradictory constraints on one or more elements in the part. For example, if two lines are constrained with different **Slope** constraints and are also constrained to be parallel, one of these constraints cannot be satisfied.

- An expression has been violated while determining new geometry. As with a violated constraint, a violated expression occurs when the result of a parametric expression contradicts an existing constraint.

When the solver or automatic constraint generator fails due to one of the above problems, a descriptive message is written to the prompt line and the solver goes into preview mode to show the status of the solution at the time the error was encountered. Elements that were successfully solved are displayed in their new

positions using the  **Preview** color (MAGENTA by default). Elements that were still unsolved when the solver exited are displayed in their original position, color and linetype. This often gives a good indication of the elements that caused the failure.

The **Display** dialog box options usually provide the clearest indication of problem sources. After every solve or constraint-extraction command, one or more info strings are always written to each drawing element that was affected by the command. These info strings give the current solve status of each element. **Show All** and **Hide All** allow you to selectively display or hide elements based on their solve status. These status displays are very effective for pointing out problem areas, especially in complex drawings, where highlighting may not clearly identify problem elements.

To open the **Display** dialog box:

- Click **File** ► **Settings** ►  **Parametric** ►  **Parametric Display**.

The **Display** options are described below.

Display Option	Function
Show All	Display all elements in the model.
Hide All	Hide all elements in the model.
Solved	Display or hide elements that were completely determined by the last solve or constraint extraction command. These elements have the PD_STATUS SOLVED info text.
Unsolved	Display or hide elements that were completely undetermined by the last solve or constraint extraction command. Unsolved elements either had no constraints on them or the solver exited before reaching them. These elements have the PD_STATUS UNSOLVED info text.

Display Option	Function
Partially Determined	Display or hide elements that were partially determined by the last solve or constraint extraction command. Some, but not all degrees of freedom have been determined for these elements. These elements have the PD_STATUS PARTIAL info text.
Inconsistent	Display or hide elements whose constraints have caused an inconsistency in the most recent solve or constraint-extraction process. These elements have the PD_STATUS INCONSISTENT info text. Inconsistent elements always fall into one of the three previous categories as well.

Debugging Solver Errors

When you're confronted with a solver error, first look at the error message posted to the prompt line. It will usually tell you whether the problem occurred because of too few constraints or because of conflicting constraints.

If the error message is something like:

```
Not enough constraints to solve for all elements
```

your drawing does not have sufficient constraints for the solver to generate a variation.

To quickly correct this situation, use **Complete** in the **Generate Constraints** dialog box to extract the missing constraints. If for some reason **Complete** is not applicable to your situation, use the **Display** options to highlight the problem areas and manually assign additional constraints to the offending elements. Start by highlighting all partially solved elements (click **Hide All** and **Show All** and then click the **Partially Determined** check box in the **Display** dialog box), and then use



Inq Elem to see what constraints are there. You may also want to examine the constraints between the partially solved elements and any completely solved elements they border.

If the error is due to conflicting constraints, the error message may resemble one of the following:

```
Encountered violated constraints
```

```
Could not constrain without violating n constraints
```

These messages indicate a violated user or system constraint.

Use the **Display** options along with  **Inq Elem** and **Show Violated** to help identify the cause of a constraint violation. Whenever a constraint is violated and the reason for the violation is not clear, first concentrate on those entities that are tagged as completely solved. The solver preview mode shows how these entities

are getting modified. Are they being modified as you expected? If so, next look at the solved and partially solved entities involved in the constraint that is being violated. Remove or re-assign the problem constraints to eliminate the conflict.

Another possibility is that the error message will resemble:

```
Inconsistently constrained elements found
```

```
Unable to constrain all elements
```

These messages indicate that an internal constraint has been violated. Examples of an internal constraint violation include:

- The implicit point that resides at the intersection of two or more elements constrained so as to move off of one or more of those elements.
- The endpoints of an arc constrained so as not to be equidistant from the arc center.

In this case, you need to identify and re-constrain the elements that are causing the conflict. For example, to isolate all elements that caused an inconsistency in the last solve, click **Hide All** to hide the entire drawing, then click **Show All** and click the **Inconsistent** check box to display only the elements that caused the inconsistency.

If the solver problem is due to a violated expression, the displays described above will be used, plus the entry for the offending expression in the parameter definition table is highlighted. To correct this situation, examine the highlighted expression carefully. You may need to rewrite it to avoid the conflict. It is also possible that the expression conflicts with geometric constraints on the part that can be removed or re-assigned.

A final class of error messages indicates problems with rigid bodies. A message of the form:

```
Unable to keep rigid body 'name' rigid
```

```
Unable to constrain while keeping rigid body 'name' rigid
```

indicates that competing constraints on the named rigid body are attempting to change its shape. This generally indicates that there are too many constraints assigned to the rigid body. Concentrate your debugging efforts on the named rigid body and eliminate constraints that are not required.

Application

There are two basic uses that the Designers have for Parametric Design. The first is to create master parts that will be used as templates for families of parts. The second is to perform modifications that are beyond the normal scope of the Creo Elements/Direct Drafting **Modify** buttons. The sections below describe each application.

Designing Master Parts

A "master part" is any constrained part that is designed to be used as a template for generating variations. For this application, the constraint information is just as important as the geometry of the part, and should be considered an integral part of the part. In order for a master part to be used successfully, the constraints must be robust enough to correctly generate the full range of required variations, and they should be set up to be accessible and easy to use, especially if the master is to be used by several people over an extended period of time.

To achieve these goals, the following procedure should be followed to prepare a master part:

1. Clean the input part.
2. Constrain the part.
3. Parameterize constraints.
4. Evaluate and debug the part.

Clean the Input Part

To consistently generate variations from a part, it is important that the original be free of geometric inconsistencies that could be carried over into the variations.

You should always look out for inconsistent geometry in any part that you intend to use as a master part. If you find inconsistencies, either remove them by using

 **Clean Pnt**,  **Clean Duplicate**, and  **Clean Stacked** in the **Modify** group on the **Parametric** tab, or isolate problem areas from consideration by the solver by collecting these entities into rigid bodies.

Constrain the Part

Constrain your part following the procedures described in the [Strategy for Assigning Constraints on page 644](#) section above. After you assign the initial set

of constraints, consider running  **No Keep** in the **Solve** group on the **Parametric** tab once. This serves two purposes. First, it quickly indicates whether your initial constraints are valid. Second, it provides an extra level of geometry cleaning by adjusting the sizes and positions of elements that fall within the **Complete** length and angle tolerances. [Advanced Topics and Tips on page 698](#) explains the **Complete** tolerance capabilities.

Parameterize Constraints

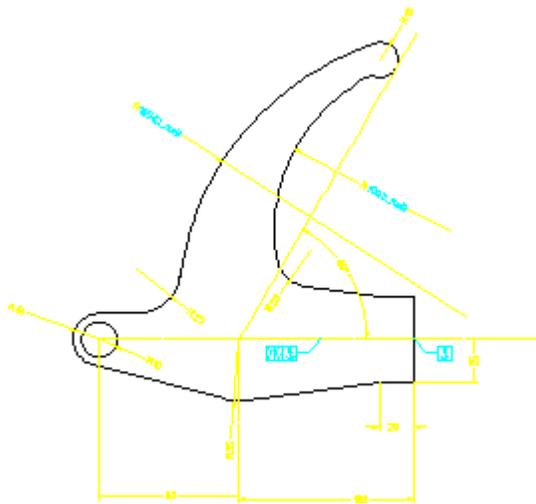
Two basic design goals for any master part should be robustness and ease of use. Parameters (described fully in [Parameters on page 679](#)) help you achieve both goals. By defining parametric values for some or all of the appropriate constraints on your part, you can reduce the number of constraints that the end user needs to modify directly in order to produce the variation. Other constraints can be

parameterized to depend on the values supplied by the user. In complex situations, expressions and macros can be supplied to check or filter the user's input before acting on it.

If the constraints that must be modified by the user are themselves parameterized, the user need not reassign these constraints. Instead, they will appear in the parameter value table, which provides a much simpler interface for the occasional user.

For example, consider the following master part, which was used to generate the variations shown in [Figure 205. master Part for rocker Arm on page 653](#):

Figure 205. Master Part for Rocker Arm



For this part, we want to generate variations in which only the configuration of the large arm changes. In all variations, we know that the outer radius of the arm should depend on the inner radius. Rather than require the end user to calculate this relationship and manually **Assign** both radii for each variation, we use parameters to do most of the work.

After constraining the part normally, we assign a dimensional constraint to the inner radius and give it the parameter name `I_rad`. The outer radius also gets a dimensional constraint, named `O_rad`. [Figure 205. master Part for rocker Arm on page 653](#) shows the constraints we supplied for this master part. The remainders were generated by **Complete**.

Using the parameter definition table, we define the constraint represented by the parameter `O_rad` to be $(1.65 * I_rad)$. This defines the relationship between the two dimensions. As a result, the user only needs to give a value for `I_rad` to generate the variation. The `I_rad` parameter is available in the parameter value table, so the user can easily change it and then **Solve** for variations.

Another advantage of parameterization is that the parameter values can be stored and recovered for later use. The **Save** and **Input** functions in the **Current Constraints** dialog box allow you to write out the current parameter value table to a file and later restore it. This feature lets you build a "library" of value tables that can be used to drive the master part. The advantage of this is that a series of standard parts based on a single master part can be stored as a single MI file (the constrained master part) and a collection of parameter value files. This arrangement takes up much less disk space than if an individual MI file were stored for each part in the series, and is much easier to update if the geometry of the master part needs to be changed. To recreate any member of the series, the user need only load the master part, **Input** the desired parameter value table, and **Solve**. [Generating Variations on page 628](#) describes this functionality.

Evaluate or Debug the Part

Once you have finished constraining a master part, you must test it to ensure that it generates variations as intended. The extent of the testing you do depends on the complexity of the part and the intended users. Obviously, you should never release a master part for use by others till you have thoroughly checked it to ensure that it operates as intended.

The easiest way to test a master part is simply to modify constraints and **Solve** for the variations you expect to be needed.  **Preview** mode is especially useful for this type of checking because it never modifies the original part. If you need to examine the variation in more detail than  **Preview** allows, you can use  **No Keep** in the **Solve** group on the **Parametric** tab, but ensure you write your original master part out to a file first so that you can recover it.

Here are some tips to keep in mind while you test:

- Try small changes to the part first, until you get a feel for how it is behaving. As you become more familiar with it, increase the size of the modifications until you reach the maximum expected modification.
- Try modifying a single constraint or parameter at a time. In complex parts, the interactions among different constraints are not always obvious until you separate them out this way.
- Try supplying nonsense values or values beyond the expected maximum. Do these break the part badly? If so, you may want to consider implementing macros or expressions that limit the values that the part will accept. These expressions can be incorporated directly into the constraint structure of the master part.

Advanced Modification

A second application for Parametric Design is to do modifications that cannot be easily done with the standard Creo Elements/Direct Drafting **Modify** buttons.

The procedure for doing modifications is very similar to that for generating master parts, but since you are generally working for yourself on a part that you have designed yourself, the process can be less formal. Here are a few differences:

- Geometry cleaning may not be as important for this application, especially if you've designed the original part yourself and know that it is correct.
- Constraints can be assigned more quickly. In many cases, it is sufficient to anchor a few elements with **Refelem** constraints, run **Complete**, and immediately modify for the variation you want.
- When you have the modification you want, make sure to free all constraints from the variation and remove it from the zone. The zone and constraint data for a complex part can take up significant amounts of memory while the part is loaded, and will increase the size of the part's file when you store it.

Constraint Types

This chapter describes the usage and function for each of the available constraint types.

Constraint Types

Each constraint type is identified by a unique icon as you assign it to the master. [Figure 206. Constraint Icons on page 656](#) shows these icons. You should take a few minutes to ensure you recognize these icons and the constraint types they represent.

Figure 206. Constraint Icons

	Angle		Ref Point
	Collinear		Ref Elem
	Dimension [*]		Same Dist
	Distance		Same Size
	Fillet		Size
	Horizontal		Slope
	Mirror		Symm Line
	Parallel		Tangent
	Perpendic		Vertical
	Point On		

^{*} Dimension constraints are indicated by color change and a postfix or change to the main dimension text.

Angle

You assign **Angle** constraints to pairs of linear elements. At solve time, the relative angle between the elements in each angle-constrained pair is adjusted as needed to match the angle specified by the constraint. If this angle cannot be achieved due to other constraints, the solve fails.

To assign an angle constraint:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Angle**.
4. Click **Apply**.
5. Click the two linear elements to be constrained.

6. Choose the angle for the constraint by any of the following methods:

- Click  **OK** or select another angle pair. The constraint value is set to the actual angle between the lines at solve time, thus maintaining this angle in the variation.
- Type a specific angle value in the user input line and press ENTER. The elements will be adjusted to this angle.
- Type a parameter name or a Creo Elements/Direct Drafting expression that defines the angle (string input). The value returned in the parameter or the result of the expression is used to adjust the angle.

Each element in an angle-constrained pair is labeled with an angle icon, [Figure 206. Constraint Icons on page 656](#), and a reference number that is unique to the pair. If the constraint is set to a specific value or parameter name, that information is appended to the icon.

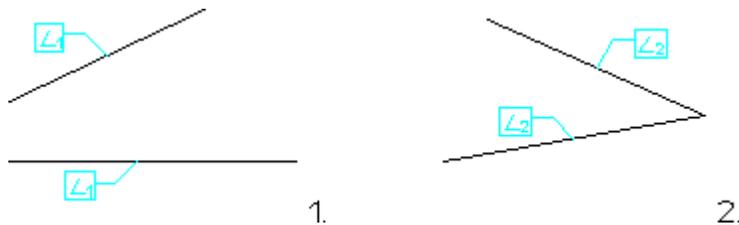
Application Notes

An **Angle** constraint will replace an existing **Parallel** or **Perpendic** constraint on a given pair of linear elements. This is because **Parallel** and **Perpendic** are simply special cases of the **Angle** constraint.

The angle between the elements in an **Angle** pair is given as the difference between the slopes of the two elements (see [Slope on page 675](#)). If the **Angle** constraint is parameterized, this angle can be accessed via expression by referencing the parameter name.

Complete will only extract angle constraints for line elements that share an endpoint or fillet. If you are assigning angle constraints manually, the members of the **Angle** pair need not be connected, [Figure 207. Angle-Constrained Lines on page 657](#).

Figure 207. Angle-Constrained Lines



Collinear

The **Collinear** constraint forces a pair of linear elements to become or remain collinear during the solve process. If other constraints on the part prevent the necessary adjustments to bring the pair together, the solve fails.

To assign **Collinear** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Collinear**.
4. Click **Apply**.
5. Click the two linear elements to be constrained.
6. Click  **OK** or assign the next constraint.

Each linear element in a collinear pair is labeled with the collinear icon, [Figure 206. Constraint Icons on page 656](#), and a reference number that is unique to the pair.

Application Notes

Lines need not be connected in order to assign or extract a **Collinear** constraint. [Figure 208. Collinear Lines on page 658](#) shows a common application for **Collinear** constraints: "gluing" a line element to a construction line.

Figure 208. Collinear Lines



When adjusting a pair of lines to make them collinear, the **Solve** group buttons try to use the smallest adjustment possible to bring the pair into line.

Dimension

Dimension constraints set the dimensions of elements in the master part to specific values. The **Solve** group buttons on the **Parametric** tab attempt to adjust constrained dimensions to the values given by the constraint. If this proves impossible, the solve fails.

To assign **Dimension** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.

-
3. In the **Type** box, select **Dimension**.
 4. Click **Apply**.
 5. Click the text of the dimension to be constrained.
 6. Choose a value for the constraint. You have three options:
 - Click  **OK** or select another dimension. The actual value of the constrained dimension at solve time will be assigned to the constraint. This dimension will not change in the variation.
 - Type a specific value for the dimension in the user input line and press ENTER. The dimension is adjusted to this value at solve time.
 - Type a parameter name or Creo Elements/Direct Drafting expression that resolves to the dimension value (string input) in the user input line and press ENTER. The dimension is adjusted to the result of the expression.

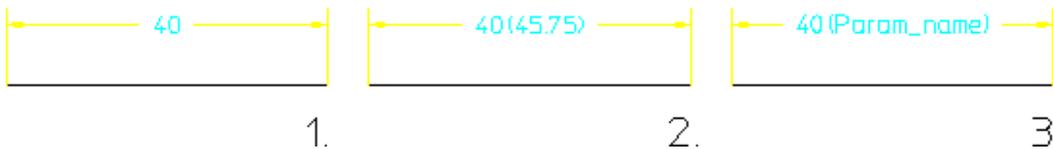
Dimension constraints are not identified by icons, but rather by a change to the dimension text and color. Depending on how you define the dimensional constraint, one of three displays will result at definition time and when you select **Show** and **Dimension** or **Show** and **All Types** in the **Generate Constraints** dialog box:

- Dimensions that are constrained to keep their current value during the solve are redrawn in the current icon color (CYAN by default).
- Dimensions that are constrained to a given value are redrawn in the current icon color (CYAN) and the main dimension text is replaced by the value given to the constraint.
- Dimensions that are given a parametric value are redrawn in the current icon color (CYAN) and the main dimension text is replaced by the parameter name, which is drawn using the dimension's font.

Note

An alternate display style for dimensional constraints can be specified by calling `PD_SHOW_USE_POSTFIX YES`, either from the Creo Elements/Direct Drafting command line or via the `pd_def.mac` file. This option causes the parameter value to be written as a postfix to the main dimension text, which remains unchanged, [Figure 209. Dimension Constraints, Alternate Style on page 660](#). Both the dimension text and postfix are redrawn in the current icon color (CYAN), to indicate that a dimensional constraint has been applied. The Parametric Design postfixes do not interfere with Creo Elements/Direct Drafting dimension postfixes, if present.

Figure 209. Dimension Constraints, Alternate Style



You cannot assign a dimension constraint to symmetric single or symmetric datum long dimensions.

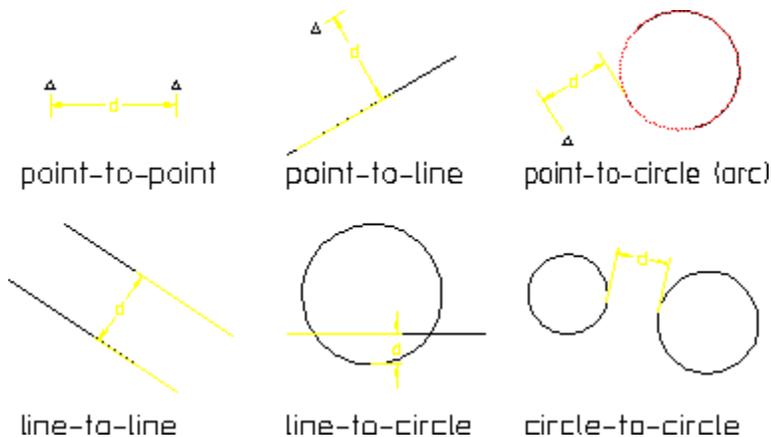
Distance

You can assign a **Distance** constraint to any pair of geometric elements except for spline-based entities (rational splines, bsplines, and ellipses). One or both elements in a distance-constrained pair may also be dimension text, as described below. At solve time, the relative distance between the extensions of the elements in the pair is adjusted to the distance specified by the constraint. If this adjustment cannot be made, the solve fails.

The extension of an element is its construction-geometry equivalent. For example, the extension of a line segment encompasses any point that would fall on a collinear construction line. The extension of an arc element encompasses any point that would fall on a construction circle that shares the arc's center and radius.

Elements in a distance-constrained pair need not be of the same type; for example, the distance between a circle and a linear element can be constrained, as can the distance between a point element and an arc. [Figure 210. Distance Determination on page 661](#) illustrates how distance is defined for different combinations of elements.

Figure 210. Distance Determination



To assign a distance constraint:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Distance**.
4. Click **Apply**.
5. Click each of the two elements to be constrained.
6. Choose the distance for the constraint. You have three options:
 - Click  **OK** or select another distance pair. The constraint distance will be set to the actual distance between the elements at solve time.
 - Type a specific distance value in the user input line and press ENTER. The distance will be adjusted to this value at solve time.
 - Type a parameter name or Creo Elements/Direct Drafting expression that defines the distance (string input) in the user input line and press ENTER. The distance will be adjusted to the result of this expression or parametric value at solve time.

Each element in a distance-constrained pair is labeled with the distance icon, [Figure 206. Constrain Icons on page 656](#), and a reference number that is unique to the pair. If the constraint is set to a specific value or parameter name, that information is appended to the icons.

Distance Constraints and Dimension Text

By default, when a dimensioned element moves or resizes as a result of a Parametric Design **Solve**, a new position for its dimension text is automatically determined. You can override this automatic placement by assigning **Distance** constraints directly to the text of the dimension(s) you want to control. This mechanism allows you to control the position of the center of the dimension text relative to the dimensioned element and relative to other elements in your drawing.

Note

Only **Distance** constraints may be assigned to dimension text; no other constraint types are allowed.

Application of distance constraints to dimension text must be done manually. The automatic constraint generator will never attempt to add distance constraints to dimensions. Because of this, the preferred method for applying distance constraints to dimensions is to first completely constrain the part without any constraints on the dimension position and then constrain only those dimension positions which need to be controlled. To assign the constraint to dimension text, click on the centerpoint of the main text element. Two constraints are generally needed to fully constrain the dimension position; one for each degree of freedom. In most cases, if you add only a single distance constraint, subsequent solves will fail. Construction geometry can often be helpful for providing additional "distance markers" in this application.

Application Notes

Several powerful parametric applications are made possible by distance-constraining strategically located point elements with regard to other elements in a drawing. In both of the following examples, point elements are created specifically for use by Parametric Design. The point elements are necessary because distance constraints can only be assigned to geometric elements, not model points.

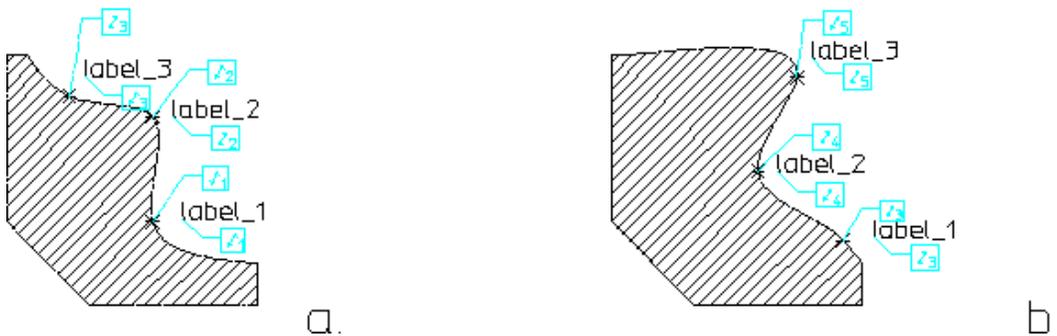
[Figure 211. Distance Constraints to Control Line Length on page 663](#) shows how distance constraints can be used to force a centerline to grow along with the rest of a part as it is resized. A point element is first assigned to each end of the centerline. Each point element is then distance-constrained to the nearest end of the part, (a). This distance will be maintained as the part is resized, (b).

Figure 211. Distance Constraints to Control Line Length



Figure 212. Distance Constraints to Control Text Position on page 663 shows how distance constraints can be used to fix text labels to a given distance from a spline. The distance is maintained as the spline is reshaped. Point elements are assigned to the interpolation points of the spline and then distance-constrained to the reference points of the text elements, (a). This distance will now be maintained as the spline is changed, (b).

Figure 212. Distance Constraints to Control Text Position



Fillet

The **Fillet** constraint can be assigned to any arc that qualifies as a fillet in the model. To qualify as a fillet, an arc must have at least one line segment or arc element at each of its ends. When an arc is constrained to be a fillet, **Solve** will ensure that it remains tangent to both the elements at its endpoints. The fillet may change radius, length or orientation as a result of the solve.

To assign **Fillet** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Fillet**.
4. Click **Apply**.
5. Click the arc to be constrained.
6. Click an element at each end of the arc to complete the fillet.

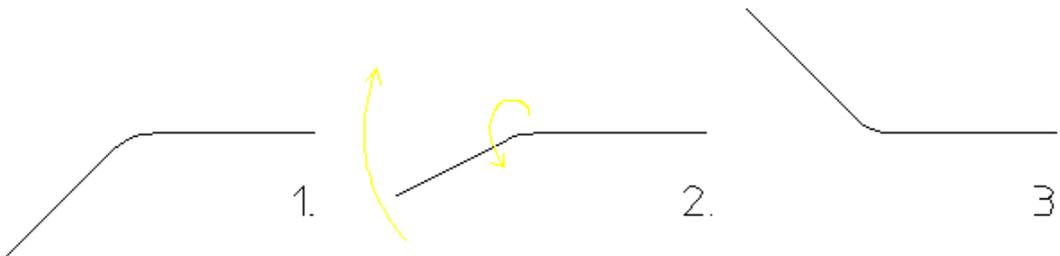
7. Click  **OK** or assign the next constraint.

Fillet-constrained arcs are represented by the fillet icon, [Figure 206. Constrain Icons on page 656](#).

Application Notes

If you assign a **Fillet** constraint to an arc that has a line segment at each end, Parametric Design ensures that the angle subtended by the arc will never become greater than 180° by "flipping" the arc whenever this angle exceeds 180° in a variation, [Figure 213. Adjustment Made for Fillet Constraints on page 664](#). This adjustment is only made in the case of an arc with a line segment at each end.

Figure 213. Adjustment Made for Fillet Constraints



If you intend for the angle subtended by the arc to be greater than 180° in the generated part, do not use a fillet constraint; instead, use two **Tangent** constraints to glue the ends of the arc to the appropriate elements.

Parametric Design fillet constraints give you better control than the Creo

Elements/Direct Drafting  **Modify** button (under  **Fillet** in the **Engineering** group on the **Drafting** tab) for changing fillet radii. By applying other constraints along with the fillet constraints, you can control exactly how elements tangent to a fillet adjust to accommodate the new radius.

Horizontal

Line elements that are assigned a **Horizontal** constraint will be adjusted as necessary at solve time to become horizontal. If **Solve** cannot make this adjustment due to other constraints on the part, the solve fails. A **Horizontal** constraint will not prevent a linear element from moving or changing length in the variation; it only adjusts the slope of that element to be zero.

To assign **Horizontal** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.

-
3. In the **Type** box, select **Horizontal**.
 4. Click **Apply**.
 5. Click the linear element to be constrained.
 6. Click  **OK** or assign the next constraint.

The horizontal icon, [Figure 206. Constrain Icons on page 656](#), identifies horizontally constrained lines.

A given element may have only one of the **Horizontal**, **Vertical**, or **Slope** constraints assigned to it. If you attempt to assign a **Horizontal** constraint to an element that is already assigned a **Vertical** constraint, the **Horizontal** constraint will replace it.

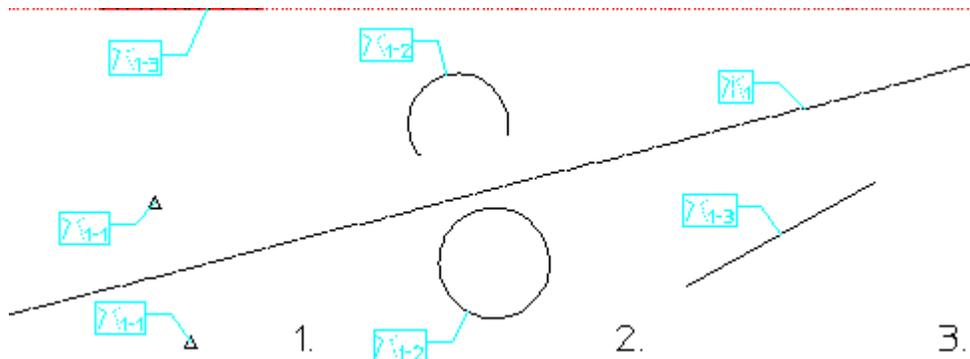
The action taken when you assign a **Horizontal** constraint to an element that is already assigned a **Slope** constraint depends on whether or not the **Slope** constraint is parameterized. If it is not, then the **Horizontal** constraint simply replaces it. If the **Slope** constraint is parameterized, however, the **Horizontal** constraint is not assigned. Instead, the value of the parameter is adjusted to 0° or 180° (effectively horizontal). This is done so as not to invalidate any expressions that may reference the existing slope parameter.

Mirror

When a pair of elements are constrained to be a **Mirror** pair, the buttons in the **Solve** group on the **Parametric** tab will attempt to move and/or resize both elements as needed so that their extensions mirror each other across a specified line of symmetry. The extension of an element is defined as the element's construction-geometry equivalent. For example, the extension of an arc is the construction circle that shares the centerpoint and radius of the arc. The extension of a line element is a collinear construction line.

Both elements in the mirrored pair must be of the same type (e.g., both circles, both lines, or both points), but need not be the same size. Mirrored lines may be of different lengths, and an arc and a circle can mirror each other. Spline and ellipsoid elements are not eligible for mirror constraints, though some measure of control over these entities can be had by assigning point elements to their control/interpolation points and then fixing those point elements. [Figure 214. Mirrored Element Pairs on page 666](#) shows several examples of mirrored element-pairs.

Figure 214. Mirrored Element Pairs



To assign a mirror constraint:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Mirror**.
4. Click **Apply**.
5. Click a linear element to be the symmetry line for the mirrored pair.
If this element does not already have a symmetry-line constraint assigned to it, one will be added.
6. Select a pair of elements to be mirrored across this line.
7. Click  **OK** or select another pair of elements to be mirrored about the original symmetry line.

When you successfully assign a mirror constraint, both elements in the mirrored pair will be labeled with the mirror icon, and the line of symmetry will be labeled with a symmetry line icon, [Figure 206. Constrain Icons on page 656](#). The icon for each symmetry line in the model is given a unique number, and the icon for each member in the mirrored pair contains a number in the form s-p, where s is the number of the symmetry line that this pair is bound to and p is the unique number for this mirrored pair.

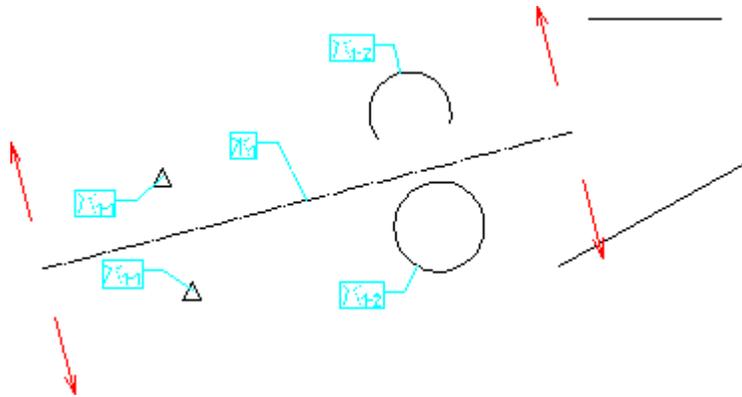
Application Notes

The symmetry line for a mirror constraint may be any linear element.

Complete only extracts mirrored pairs from areas that are orthogonal to defined symmetry lines. If your symmetry lines have infinite length (e.g., construction lines) this is no limitation, but if you choose a finite-length element as a symmetry line, items must lie completely within the orthogonal boundary in order to be

selected for mirroring. The two line segments on the right of [Figure 215. Auto-selection of Mirrored Pairs on page 667](#), for example, would not be extracted because neither falls inside the boundary. If you are assigning mirrored pairs manually, this limitation does not apply.

Figure 215. Auto-selection of Mirrored Pairs



Parallel

The **Parallel** constraint is assigned to pairs of linear elements. A pair of parallel-constrained elements will be adjusted to be parallel at solve time. If other constraints on the part prevent the pair from being made parallel, the solve fails. The positions, lengths, and absolute slopes of parallel elements are not affected by this constraint and may or may not change during the solve, but the slopes of the lines will be the same.

To assign **Parallel** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Parallel**.
4. Click **Apply**.
5. Click the linear elements to be constrained.
6. Click  **OK** or assign the next constraint.

Each linear element in a parallel pair is labeled with the parallel icon, [Figure 206. Constrain icons on page 656](#), and a reference number that is unique to the pair.

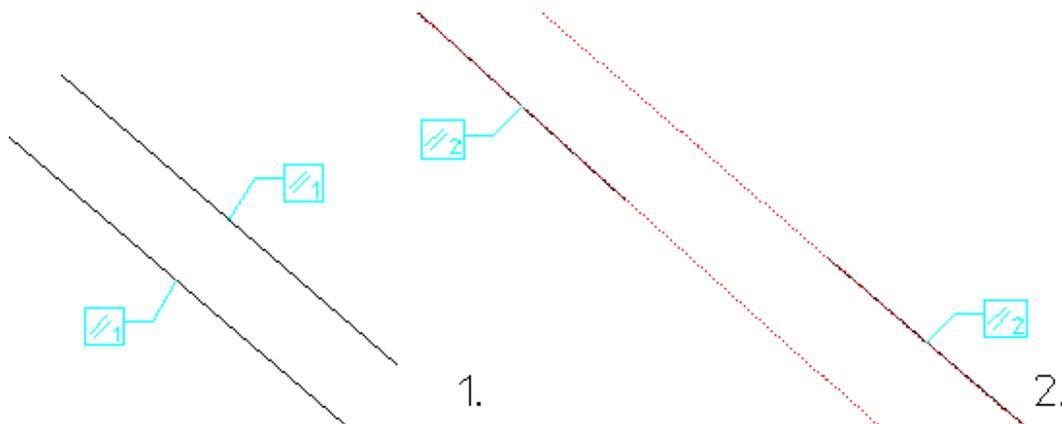
Application Notes

A given pair of linear elements may have only one of the **Angle**, **Parallel**, or **Perpendic** constraints assigned to it. This is because **Parallel** and **Perpendic** are simply special cases of the **Angle** constraint. If you attempt to assign a **Parallel** constraint to a pair of elements that is already assigned a **Perpendic** constraint, the **Parallel** constraint will replace it.

The action taken when you assign a **Parallel** constraint to a pair of elements that is already assigned an **Angle** constraint depends on whether or not the **Angle** constraint is parameterized. If it is not, then the **Parallel** constraint simply replaces it. If the **Angle** constraint is parameterized, however, the **Parallel** constraint is not assigned. Instead, the value of the parameter is adjusted to 0° or 180° (effectively parallel). This is done so as not to invalidate any expressions that may reference the existing angle parameter.

Parallel lines need not describe an orthogonal area in the part; [Figure 216. Parallel Lines on page 668](#) shows two sets of parallel lines.

Figure 216. Parallel Lines



Perpendicular

The **Perpendic** constraint is assigned to pairs of linear elements. A pair of perpendicular-constrained elements will be adjusted to be perpendicular at solve time. If other constraints on the part prevent this adjustment, **Solve** fails. The positions, lengths, and absolute slopes of perpendicular elements are not affected by this constraint and may change during the solve, but the slopes of the lines will be at right angles to one another.

To assign **Perpendic** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.

The **Generate Constraints** dialog box opens.

2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Perpendic**.
4. Click **Apply**.
5. Click the linear elements to be constrained.
6. Click  **OK** or assign the next constraint.

Each linear element in a perpendicular pair is labeled with the perpendicular icon, [Figure 206. Constrain Icons on page 656](#), and a reference number that is unique to the pair.

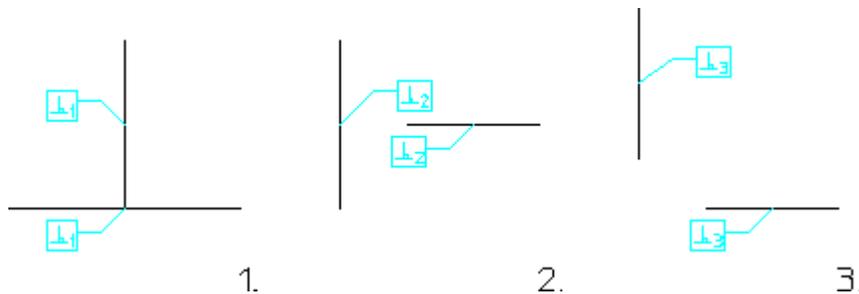
Application Notes

A given pair of linear elements may have only one of the **Angle**, **Parallel**, or **Perpendic** constraints assigned to it. This is because **Parallel** and **Perpendic** are simply special cases of the **Angle** constraint. If you attempt to assign a **Perpendic** constraint to a pair of elements that is already assigned a **Parallel** constraint, the **Perpendic** constraint will replace it.

The action taken when you assign a **Perpendic** constraint to an element pair that is already assigned an **Angle** constraint depends on whether or not the **Angle** constraint is parameterized. If it is not, then the **Perpendic** constraint simply replaces it. If the **Angle** constraint is parameterized, however, the **Perpendic** constraint is not assigned. Instead, the value of the parameter is adjusted to 90° or 270° (effectively perpendicular). This is done so as not to invalidate any expressions that may reference the existing angle parameter.

Lines need not be in contact with one another to be constrained perpendicular. [Figure 217. Perpendicular Lines on page 669](#) shows three different pairs of perpendicular lines.

Figure 217. Perpendicular Lines



Point on

Point on constrains a model point to remain on a given geometric element or its extension.

The point part of this constraint may be any visible model point. Line endpoints, circle centers, spline control points, and text reference points may all be used if visible. If necessary, click the **Vertex** check box in the Creo Elements/Direct Drafting **Show** dialog box to make these elements visible.

The extension of an element is its construction-geometry equivalent. For example, the extension of a line segment encompasses any point that would fall on a collinear construction line. The extension of an arc element encompasses any point that would fall on a construction circle that shares the arc's center and radius. All Creo Elements/Direct Drafting geometric elements, including splines, B-splines, and ellipses, are eligible to receive **Point on** constraints.

The constrained point may slide anywhere along the extension of the element it is attached to, but will always remain in contact with the extension. If other constraints prevent this contact, **Solve** fails.

To assign **Point on** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Point on**.
4. Click **Apply**.
5. Click the point to be constrained. You can select any visible model point (use the Creo Elements/Direct Drafting **Show** dialog box to display model points if needed). If no point element exists at that location, one will be created.
6. Click the geometry to be attached to the point.
7. Click  **OK** or assign the next constraint.

The **Point on** icon, [Figure 206. Constrain Icons on page 656](#), identifies this constraint.

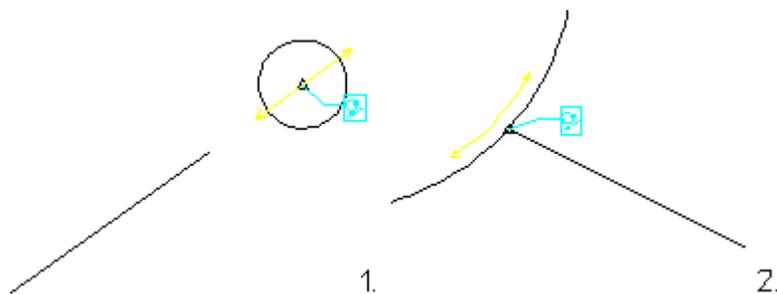
Application Notes

Two common applications for **Point on** constraints are shown in [Figure 218. Point-On Applications on page 671](#). Circles can be constrained to stay on a linear element by fixing the centerpoint on the line. Conversely, elements can be constrained to remain attached to the perimeter of a circle, ellipse or spline.

 **Note**

To assign a **Point on** constraint to the centerpoint of a circle or arc, you must first display centerpoints (with the **Vertex** check box clicked in the Creo Elements/Direct Drafting **Show** dialog box) and then click the centerpoint directly.

Figure 218. Point-On Applications



Reference Element

Any element in a part (except ellipses) can be fixed in place with a **Refelem** constraint. Reference elements are not changed in any way by **Solve**, so they will appear in the same position in the variation as they do in the master part.

To assign **Refelem** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Refelem**.
4. Click **Apply**.
5. Click the element to be constrained.
6. Click  **OK** or assign the next constraint.

Reference elements are labeled with the anchor icon, [Figure 206. Constrain Icons on page 656](#).

Application Notes

You should almost always constrain one or two elements in the master part as reference elements (construction geometry is good for this) so that **Solve** has a frame of reference for generating the variation. Otherwise, the variation may not be placed where you expect.

Reference Point

Any visible model point can be constrained to be a reference point. **Solve** places reference points in the variation at the position specified by the constraint.

To assign a reference point:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Refpoint**.
4. Click **Apply**.
5. Click the point to be constrained. You may select any visible model point (use the Creo Elements/Direct Drafting **Show** dialog box to display model points if needed) If a point element does not exist at the location you click, one will be created.
6. Specify a position for the reference point. You have two options:
 - Click  **OK** or select another point. The reference point will be placed into the variation at its current position at solve time.
 - Type a parameter name or Creo Elements/Direct Drafting expression that resolves to the point position (string input) in the user input line and press ENTER.

The reference point will be labeled with the position symbol, [Figure 206](#). [Constrain Icons on page 656](#). If the reference point is set to a parameter name, that information is appended to its icon.

Application Notes

Reference points, like reference elements, can be used to "anchor" portions of the master part in place, so that they maintain their positions in the variation.

To assign a reference point constraint to a model point that is not usually visible, for example, circle centers, spline control points, or text centers, you must first display them (with **Vertex** check box clicked in the Creo Elements/Direct Drafting **Show** dialog box) and then click them directly.

A reference point can also be used to supply data to macros or expressions when given a parametric value. If you supply a parameter label when assigning a reference point, the coordinate value of the point can be accessed in an expression by referencing the parameter name.

Same Distance

Samedist is assigned to two pairs of elements. All geometric elements except for spline-based entities (rational splines, bsplines, and ellipses) are eligible. You can assign constraints to the interpolation or control points of these entities, just not the spline itself. **Samedist** forces the distance between the members of the first pair of elements to be the same as the distance between the members of the second pair. If other constraints prevent the two distances from being made equal, **Solve** fails.

To assign **Samedist** constraints:

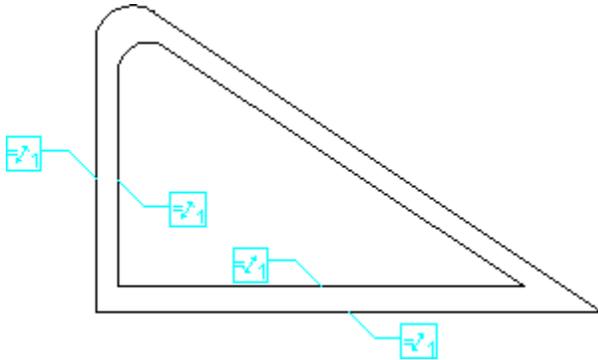
1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Samedist**.
4. Click **Apply**.
5. Click the two elements for the first pair.
6. Click the two elements for the second pair.
7. Click  **OK** or assign the next constraint.

Each element in a same-distance set is labeled with the same-distance icon, [Figure 206. Constrain Icons on page 656](#), and a reference number that is unique to the set.

Application Notes

Samedist constraints are often useful for forcing two contours to become/remain equidistant, [Figure 219. Same Distance Application on page 674](#).

Figure 219. Same Distance Application



Same Size

The **Same Size** constraint is applied to pairs of geometric elements, which must be of the same geometric type. All elements except for spline-based entities (rational splines, bsplines, and ellipses) and construction lines are eligible for this constraint. The members of a same-size pair will be adjusted as needed by **Solve** to become the same size. Specifically, the radii of circular elements and the length of linear elements is adjusted. If the members of a same-size pair are not in fact the same size at solve time, **Solve** will look at other constraints on the members of the pair to determine which member should be changed to match the size of the other.

To assign **Same Size** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Same Size**.
4. Click **Apply**.
5. Click the two elements to be constrained.
6. Click  **OK** or assign the next constraint.

Each element in a same-size pair is labeled with the same-size icon, [Figure 206. Constrain Icons on page 656](#), and a reference number that is unique to the pair.

Size

You can assign **Size** constraints to lines and all circular elements. At solve time, these elements will be adjusted to the size specified by the constraint. For circular elements, the adjustment is made to the radius; for lines, the length is adjusted. If the adjustment cannot be achieved, **Solve** fails.

To assign a size constraint:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Size**.
4. Click the element to be constrained.
5. Choose the size for the constraint by any of the following methods:
 - Click  **OK** or select another element. The element will be held at its current size at solve time. This is the default.
 - Type a specific size (radius for circular elements, length for lines) in the user input line and press ENTER.
 - Type a parameter name or Creo Elements/Direct Drafting expression that defines the size (string input) in the user input line and press ENTER.

Each size-constrained element pair is labeled with the size icon, [Figure 206. Constrain Icons on page 656](#). If the constraint is set to a specific value or parameter name, that information is appended to the icon.

Slope

Slope constraints are assigned to linear elements. At solve time, the slope of these elements will be adjusted to match the angle specified by the constraint. If the adjustment cannot be made, **Solve** fails.

Note

Slope constraints do not fix the position or length of a line, which may change unless otherwise constrained.

To assign a slope constraint:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Slope**.
4. Click **Apply**.
5. Click the linear element to be constrained.

The current slope of this element is displayed in the Creo Elements/Direct Drafting user input line.

6. Choose the angle for the constraint by any of the following methods:
 - Click  **OK** or select another element. The slope will be set to the actual slope of the line at solve time.
 - Type a specific angle value for the slope in the user input line and press ENTER.
 - Type a parameter name or Creo Elements/Direct Drafting expression that defines the slope angle (string input) in the user input line and press ENTER.

Each slope-constrained element is labeled with the slope icon, [Figure 206. Constrain Icons on page 656](#). If the constraint is set to a specific value or parameter name, that information is appended to the icon.

Application Notes

A given element may have only one of the **Horizontal**, **Vertical** or **Slope** constraints assigned to it. If you attempt to assign a **Slope** constraint to an element that is already constrained **Horizontal** or **Vertical**, the **Slope** constraint will replace the existing constraint.

Parametric Design defines the slope of a linear element as the angle which is swept by rotating the vector (1,0) to the intrinsic vector of the element. This intrinsic vector is the line that runs from the first to the second endpoint of the element. The intrinsic vector of a construction line depends on how it was created. This method is used so that rotations greater than 90° can be specified.

As a result of the method used to measure slope, it is not possible to visually determine the slope of a linear element from the graphics display. For example, a vertical line may have a slope of 90° or 270°, depending on whether it was constructed from "top to bottom" or "bottom to top." You can, however, read the current slope from the prompt line in step 3 of the assignment procedure above.

Symmetry Line

The **Symmline** constraint identifies a line of symmetry that can be used by **Complete** to assign mirror constraints. If you want to assign mirror constraints by hand, you should use **Assign Mirror** instead. You assign **Symmline** constraints to linear elements, which are then labeled with the symmetry line icon, [Figure 206. Constrain Icons on page 656](#).

To assign **Symmline** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.

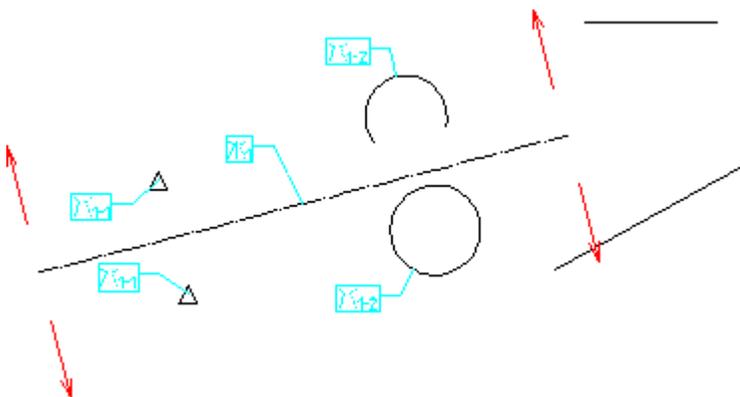
-
- The **Generate Constraints** dialog box opens.
 - In the **Action** area, click **Assign**.
 - In the **Type** box, select **Symmline**.
 - Click **Apply**.
 - Click the linear element to be constrained.
 - Click **OK** or assign the next constraint.

Application Notes

Parametric Design can automatically assign symmetry-line constraints to lines having a particular color and linestyle. You enable this behavior by calling the function `PD_AUTO_SYMMETRY YES` from the Creo Elements/Direct Drafting command line, or by including it in your `pd_def.mac` file. The functions `PD_AUTO_SYMMETRY_COLOR` and `PD_AUTO_SYMMETRY_LINETYPE` specify the color and linestyle. See [Parametric Design Defaults on page 761](#) for more information.

Complete only extracts mirrored pairs from areas that are orthogonal to defined symmetry lines. If your symmetry lines have infinite length (e.g., construction lines) this is no limitation, but if you choose a finite-length element as a symmetry line, items must lie completely within the orthogonal boundary in order to be selected for mirroring. The two line segments on the right of [Figure 220. Auto-selection of Mirrored Pairs on page 677](#), for example, would not be extracted because neither falls inside the boundary. If you are assigning mirrored pairs manually, this limitation does not apply.

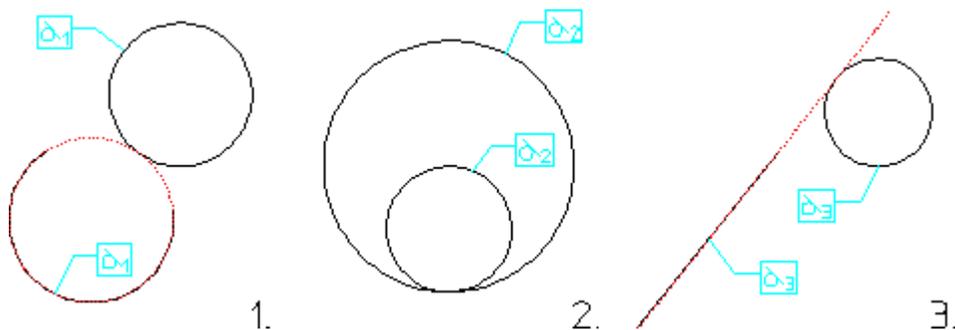
Figure 220. Auto-selection of Mirrored Pairs



Tangent

Tangent constraints are applied to pairs of geometric elements, at least one of which must be circular (circle, construction circle, arc, or fillet). At solve time, the elements are adjusted to ensure that their extensions are tangent. See [Point on on page 655](#) for an explanation of extensions. If this adjustment cannot be achieved, **Solve** fails. Two elements are considered tangent if their extensions touch at one point only, [Figure 221. Tangent Pairs on page 678](#).

Figure 221. Tangent Pairs



To assign **Tangent** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Tangent**.
4. Click **Apply**.
5. Click the two elements to be constrained. One element must be circular.
6. Click  **OK** or assign the next constraint.

Each element in a tangent pair is labeled with the tangent icon (see [Figure 206. Constrain Icons on page 656](#)) and a reference number that is unique to the pair.

Application Notes

Complete will not extract a tangent constraint for a pair of elements unless the elements themselves (not their extensions) are tangent. **Solve**, however, recognizes linear and circular extensions when adjusting for tangency.

Solve attempts to maintain as much of the current configuration as possible when adjusting to satisfy tangency constraints. When adjusting a circular and a linear element to be tangent, **Solve** attempts to keep the circle on the same side of the

line as it was originally. When adjusting two circles to be tangent, effort is taken to ensure that if one circle is inside the other, it remains inside, and that circles which are outside of each other stay that way.

Vertical

Linear elements that are assigned a **Vertical** constraint will be adjusted as necessary at solve time to become vertical. If **Solve** cannot make this adjustment due to other constraints on the element, **Solve** fails. A vertical constraint will not prevent a linear element from moving or changing length in the variation; it only adjusts the slope of that line to 90° or 270°.

To assign **Vertical** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Vertical**.
4. Click **Apply**.
5. Click the linear element to be constrained.
6. Click  **OK** or assign the next constraint.

The vertical icon, [Figure 221. Tangent Pairs on page 656](#), identifies vertically constrained lines.

Application Notes

A given element may have only one of the **Horizontal**, **Vertical** or **Slope** constraints assigned to it. If you attempt to assign a **Vertical** constraint to an element that is already assigned a **Horizontal** constraint, the **Vertical** constraint will replace it.

The action taken when you assign a **Vertical** constraint to an element that is already assigned a **Slope** constraint depends on whether or not the **Slope** constraint is parameterized. If it is not, then the **Vertical** constraint simply replaces it. If the **Slope** constraint is parameterized, however, the **Vertical** constraint is not assigned. Instead, the value of the parameter is adjusted to 90° or 270° (effectively vertical). This is done so as not to invalidate any expressions that may reference the existing slope parameter.

Parameters

A parameter is a named variable that you create with Parametric Design. You can create a parameter implicitly by giving a parameter name when you assign one of the several constraint types that support parameters; or you can create a parameter

explicitly by adding a new entry to the parameter definition table, which you access by clicking **Advanced** in the **Current Constraints** dialog box. Regardless of how you create a parameter, it is given an entry in this table. By editing the entry, you can assign a value to the corresponding parameter.

The value you assign to a parameter may be either a discrete numeric value or an expression. An expression is any valid Creo Elements/Direct Drafting expression. This line can reference other Parametric Design parameters, Creo Elements/Direct Drafting macros, and any of the arithmetic and logical constructs supported by the Creo Elements/Direct Drafting macro language.

The most important feature of parameters is that they can be assigned to constraints. After you assign a parameter to a constraint, you can modify the constraint by changing the parameter's value. Conversely, you can access the current value of the constraint by inquiring the parameter. This capability makes parameters an especially powerful tool for defining relationships among elements in a part.

Creating Parameters

Creating Parameters Implicitly

Parametric Design implicitly creates a new parameter whenever you **Assign** one of the following constraint-types and supply an unused parameter name (string) when prompted to do so by the software:

- **Angle**
- **Dimension**
- **Distance**
- **Refpoint**
- **Size**
- **Slope**

The name you give must conform to the naming rules given in the [Assigning Parameters on page 682](#), section and must not be the same as an existing parameter name. If you give an existing parameter name, that parameter is simply assigned to the constraint and no new parameter is created.

When you create a new parameter with this procedure, Parametric Design implicitly creates a default definition for the parameter and then binds it to the constraint you're assigning.

By default, the new parameter is given the current value of the constraint. For example, if you assign a size constraint to a 10 cm line and give the label 'Len_3' at the prompt, Parametric Design will create a new parameter, Len_3, and

give it a value of 100, assuming your current length units are mm. Parameters always use the units they were created with. `Len_3`, for example, will always be in mm even if you change your length units to `inches`.

Creating Parameters Explicitly

You can also create new parameters directly by adding entries to the parameter definition table. Parameters that you create this way are not automatically assigned to constraints on your part. For this reason, explicit creation is often used to create parameters that will be used only to supply constants, sub-expressions, or other values that have no relationship to the part.

To explicitly create a parameter:

1. Click **Parametric** and then, in the **Show** group, click  **Current**.

The **Current Constraints** dialog box opens.

2. Click **Advanced**.

The **Advanced** dialog box opens. It contains the parameter definition table.

The parameter definition table provides the following commands for the creation of new parameters:

Command	Action
Length	Create a new length parameter
Angle	Create a new angle parameter
Point	Create a new point parameter
User	Create a new user (unitless) parameter

Each of these commands adds a new parameter to the table. These parameters are not associated with constraints, but you can name and define them like any other parameter. The new parameters can be referenced in other expressions and can reference other parameters in their own expressions. Once defined, you may use them for reference or assign them to constraints in your part.

3. To create a new parameter:
 - a. Click the required command (for example, **Angle**).

The **Add Constrain** dialog box opens.
 - b. Type a name for the new parameter (for example, `'Angle_4'`).
 - c. Click **OK**.

The name you give must be a suitable parameter label as described in [Assigning Parameters on page 682](#). An explicitly created parameter is given an initial value of 0 and a value type of "unknown." Its value remains unknown until you assign it a value or until you assign it to a drawing element.

Assigning Parameters

Only the following constraint-types may be parameterized:

- **Angle**
- **Dimension**
- **Distance**
- **Ref Point**
- **Size**
- **Slope**

To assign a parameter to any of these constraint-types, you simply **Assign** the constraint to the desired element or elements, and enter the parameter name (enclosed in quotes) when prompted to do so.

Only those constraint types that support parameters will prompt you for them. If the named parameter does not exist, then Parametric Design creates a new parameter with this name, as explained above. If a parameter with this name does exist and matches the unit requirements of the constraint, it becomes associated with the constraint.

Parameter names are text strings. They must adhere to the naming rules for Creo Elements/Direct Drafting macros. Parameters cannot have the same name as a Creo Elements/Direct Drafting keyword, (for example, `rad` or `CONFIRM`). To avoid such conflicts, parameter names should begin with an uppercase letter, and contain only lowercase letters, integers, and/or the underscore (`_`) character in the remaining positions. The names `Distance`, `Rad`, `Dim_3`, and `My_line` are examples of valid parameter names. The software always displays parameter names with the first letter uppercase and remaining characters lowercase, regardless of how you enter them.

It is always a good idea to choose parameter names that are as descriptive as possible - especially if you are parameterizing a part that other people will use as a master part. A drafts person is much more likely to understand the purpose of the parameter `Circle3_radius` than the parameter `Wwxyz`.

Once you assign a parameter name to a constraint, the parameter remains assigned until you assign a different parameter name to this constraint, or until you **Free** the constraint.

Editing Parameter Definitions

The parameter definition table can be used to display and edit the definitions of all existing parameters on the current part. The parameter definition table is located in the **Advanced** dialog box.

To open the **Advanced** dialog box:

-
1. Click **Parametric** and then, in the **Show** group, click  **Current**.

The **Current Constraints** dialog box opens.

2. Click **Advanced**.

The **Advanced** dialog box opens. It contains the parameter definition table.

The parameter definition table contains one entry for each parameter. Each entry has five columns, which define the parameter:

- **Name**
- **Kind**
- **Value**
- **New Value Type**
- **New Value or Expression**

You can only edit the **New Value or Expression** column. The first four columns just supply information. The following sections describe each column:

Name

The **Name** column gives the parameter name as you assigned it. This is simply the label that identifies the parameter.

Kind

The **Kind** column declares the units for this parameter. Units are determined by the type of constraint this parameter is assigned to. The following "kinds" of units are recognized:

Kind	Units
Angle	An angle in the angular units in force when the parameter was created (degrees, radians, and so on).
Length	A linear distance in the linear units in force when the parameter was created (mm, inches, miles, and so on).
Point	An XY coordinate pair in the linear units in force when the parameter was created; for example, 20,30.5.
User	Unitless. A user parameter is never associated with a geometric constraint, so the solver never has to convert this parameter into system units. Only parameters created with the User option can be user parameters.

Value

The **Value** column shows the value that is currently associated with this parameter. This value may change when you next **Solve** for a variation. The value for a parameter is always given in the same units that were in force when the parameter was first created.

Parameters that resolve to point values use two lines in the parameter table. The value on the first line is the X coordinate of the point, while the second line contains the Y coordinate.

New Value Type

The **New Value Type** column specifies the method that **Solve** will use to evaluate this parameter. **New Value Type** must be one of the following:

Value Type	Meaning
Value	Set the value for this parameter to the specific value in the New Value or Expression column. Assume that the given value is in the correct units. Other parameters can depend on this parameter via expressions. This value is available to any geometric constraints that reference this parameter to solve for the geometry.
Expression	Generate a value for this parameter by resolving the expression given in the New Value or Expression column. Assumes that the expression is a valid Creo Elements/Direct Drafting expression which resolves to a value of the correct type. If the expression contains references to other parameters then the expression is only evaluated after all referenced parameters have a value. Conversely, other parameters can depend on this parameter via expressions. The value produced by this expression is available to all geometric constraints that reference this parameter.

Value Type	Meaning
Geometry	<p>The value for this parameter is taken from geometry that is solved for using constraints that are not associated with this parameter. Other parameters can depend on this parameter via expressions.</p> <p>For example, take a part in which there are two circles whose radii are parameterized as Radius_1 and Radius_2. If the first circle is tangent to three fixed lines then the value of Radius_1 is completely determined by the "geometry" of the three lines. An expression for Radius_2 such as (2 * Radius_1) is then possible. This works because Solve recognizes that the expression for Radius_2 should not be evaluated until Radius_1 is known. Radius_1 can only be known after the three tangent line constraints are evaluated.</p>
Unknown	<p>This parameter has been named, but has no association to an existing constraint and has no value or expression assigned. This parameter type is not evaluated by the solver.</p>

New Value or Expression

Use this column to assign new values for parameters. The value or expression contained in this column at **Solve** time will be interpreted and assigned to the corresponding parameter. If you leave this column blank, then the parameter value is extracted from the geometry of the part.

Assigning a Specific Value to a Parameter

To assign a specific value to a parameter:

1. Click the parameter's **New Value or Expression** column.
2. Type a numeric value (in appropriate units) in the text box.

The number you provide is displayed in the **New Value or Expression** column and the **New Value Type** column is set to "Value". This value will be assigned to this parameter by the next **Solve** command.

Assigning an Expression to a Parameter

To assign an expression to a parameter:

1. Click the parameter's **New Value or Expression** column.
2. Type a valid Creo Elements/Direct Drafting expression. All expression input must be enclosed in quotes. For example:

(57 + Dist)	Assign the sum of 57 and 'Dist' to the parameter.
(IF (A < 8) 57 ELSE 45 END_IF)	Assign 57 if the value of 'A' is less than 8; otherwise assign 45
(PNT_XY (X_OF 'PtA') 10)	Assign to a point parameter the x-value of 'PtA' and a y-value of 10.
My_macro	Evaluate 'My_macro' and assign the result.
My_macro2 Rad	Evaluate 'My_macro2', which needs a parameter that passes in the value of 'Rad'

The expression you provide is displayed in the **New Value or Expression** column and **New Value Type** becomes "Expression." The next use of the **Solve** group buttons causes this expression to be evaluated by Parametric Design and the result assigned to this parameter.

Extracting a Value from the Geometry

To extract a value from the geometry:

1. Click the parameter's **New Value or Expression** column.
2. Type an empty string ("") to clear the contents of the column.

The **New Value or Expression** column is now blank and **New Value Type** becomes "Geometry." Subsequent use of the **Solve** group buttons causes Parametric Design to extract a value for this parameter from the constrained geometry.

Geometry-type constraints are often useful for determining the locations of elements in the part. Suppose for example that you have defined several expressions that need to know the location of a particular point in the part. An easy way to pass this data is with a parameterized **Refpoint** constraint:

- Assign the **Refpoint** constraint to the model point you are interested in. Give the parameter name Pt1 to this constraint as you assign it.
- Use the parameter definition table to convert Pt1 to a "Geometry" type parameter.
- Now, whenever you **Solve**, the new location of the model point will be extracted and assigned to Pt1. Your expressions can access the location by referencing Pt1.

Removing Parameters

To remove parameters from the part:

1. Click **Parametric** and then, in the **Show** group, click  **Current**.

The **Current Constraints** dialog box opens.

-
2. Click **Advanced**.

The **Advanced** dialog box opens. It contains the parameter definition table.

3. In the **Name** column, click the parameter that you want to remove.
4. Click **Remove**.

Parametric Design does not allow you to remove parameters that are currently assigned to a constraint or that are referenced by another parameter via expression. To remove parameters that are assigned to constraints, first either **Free** the constraint that corresponds to the parameter or assign a new value to it, then **Remove** the parameter entry. To remove a parameter that is referenced by other parameters, you must first remove the offending references by editing the expressions that contain them.

Showing Parameters

The **Show** command in the parameter definition table highlights all elements in the part that currently have parameters assigned to them. Highlighting is done by displaying the constraint icons of these elements along with the corresponding parameter names. Parameter labels will be displayed using the current text font. Highlighting is in CYAN by default.

The **Show** highlighting is removed whenever the buttons in the **Solve** group are invoked. Alternatively, you can click **Clear** and select **All Types** in the **Generate Constraints** dialog box to remove the highlighting.

Setting Parameter Values

A common strategy in Parametric Design is to define your parameters so that the entire constraint structure of a part depends on several "Value"-type parameters. This strategy has several advantages:

- It is often possible to generate new variations by simply changing a few key value-type parameters.
- In situations where a Designer or Engineer is responsible for designing constraints for a master part and a Draftsperson is responsible for generating variations, this strategy "isolates" most of the constraint structure from the Draftsperson, who only has to modify the values of a few parameters to generate variations.

Parametric Design facilitates this strategy by giving easy access to all the currently set "Value" parameters in the parameter value table in the **Current Constraints** dialog box. Rather than display the entire parameter definition table whenever you want to change a "Value" parameter, you can display the **Current Constraints** dialog box, which contains only "Value" parameters.

Each entry in the parameter value table contains the name of a currently defined "Value" parameter along with the value that will be assigned to that parameter by the next **Solve** command. This value corresponds to the **New Value or Expression** column of the parameter definition table. Only those parameters with a type of "Value" appear on this column. If there are too many entries to view at once, use the scroll bar beside the table. To change the value of a parameter, simply select its entry and enter the new value. A full description of the parameter value table can be found in [Generating Variations on page 628](#).

 **Note**

While you can enter an expression rather than a value into the parameter value table, it is not recommended that you do so, because this changes the type of the parameter from "Value" to "Expression," and thus removes its entry from the parameter value table (but not from the parameter definition table).

Dimension Driven Modification

Roadmap

Read this chapter to learn how to modify geometry by modifying parametric dimensions. To learn how to assign constraints explicitly read [Working with Parametric Design on page 633](#) instead. To learn how to create, assign, and modify parameters read [Parameters on page 679](#).

Overview

Dimension Driven Modification simplifies access to some of the power of parametric modification. Geometry can be modified by simply selecting dimensions and entering new values.

Constraints associated with a number or a parameter (angle, dimension, distance, size, slope) can be considered parametric dimensions. Regardless of how they were created, all parametric dimensions can be modified by using the buttons

under  **Dimension** in the **Modify** group on the **Parametric** tab.

Viewing Parametric Dimensions

Display

To display parametric dimensions associated with selected geometry:

-
1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
 2. Click  **Show**.

Displaying parametric dimensions may require the automatic creation of temporary geometry (which is not visible). For example, dimensioning the radius of a construction circle is not allowed, so a real circle will be created temporarily for the display of parametric dimensions. Any temporary geometry is deleted by the Parametric Design module as soon as parametric dimensions are removed from the display.

Parametric dimensions can be displayed only when a part is fully and properly constrained. If the part is under-constrained when you click **Show** and then **Dimensions**, you are asked to confirm the running of **Complete**. **Complete** can be executed reliably when the part is mostly constrained — if the geometry was created with Design Intent Capture enabled, for example. But if the part does not contain any constraints, **Complete** should not be run till some initial constraints have been assigned. Refer to [Working with Parametric Design on page 633](#) for information on constraint assignment.

 **Note**

Parametric dimensions are only associated with constraints that contain a number or a parameter. Constraints such as parallel, horizontal, same size, reference element, and so on will not have parametric dimensions associated with them. A combination of constraint editing and dimension creation can be used to construct any desired parametric dimensions.

Hide

To hide parametric dimensions associated with selected geometry:

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Hide**.

Especially with large parts, the ability to selectively display and hide parametric dimensions can reduce screen clutter. A designer can quickly focus on the parametric dimension of interest.

Display Attributes of Parametric Dimensions

Color

All parametric dimensions are displayed with the same color. To set the color value use `PD_DEFAULT_DIM_COLOR`. For example, to specify green as the color for parametric dimensions type `PD_DEFAULT_DIM_COLOR 0, 1, 0` in the user input line and press ENTER.

All other attributes are controlled by the current settings of the dimension module.

Location

Parametric dimensions — unlike other dimensions — are displayed in default locations. You can move them for easier viewing.

To move a parametric dimension:

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Move**.

Note

If you have the Dimension Advisor Module enabled, the parametric dimensions are placed according to the selection made in the **Strategy** box in the **Options** settings of the **Dimension Settings** dialog box (the **Automatic** check box should be clicked in the **Options** settings).

Modifying Parametric Dimension Values

Once parametric dimensions are displayed, their values can be modified. Changing the value of a parametric dimension results in a corresponding change to the geometry of a drawing. The drawing can be updated after every modification, or the drawing update can be deferred until many parametric dimensions have been modified.

Immediate Modification

 **Immediate** specifies that all geometry should be updated as soon as the value of a parametric dimension is modified. This method can be useful with small drawings — where the speed of updating the geometry and redrawing the screen is not a concern. It can also be useful for experimenting with several different values for one parametric dimension.

To use  **Immediate**:

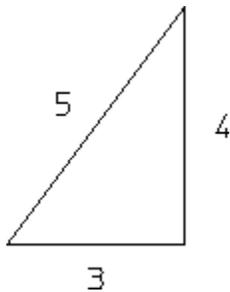
1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Immediate**.
3. Type the new value in the user input line and press ENTER.

Deferred Modification

 **Deferred** specifies that geometry should not be modified until the user makes an explicit request. This method improves performance, because the drawing is updated only once — even if the values of several parametric dimensions have been modified.

 **Deferred** is actually necessary when a set of parametric dimension modifications do not have valid intermediate geometry. For example, consider a triangle with sides of lengths 3, 4, and 5:

Figure 222. Triangle Requiring Deferred Modification



It would not be possible to change the side lengths to 30, 40, and 50 using  **Immediate**, because no valid triangle can be created with a combination of old and new lengths.

Note

Often a combination of  **Immediate** and  **Deferred** is useful when several parametric dimensions need modification.

To use  **Deferred**:

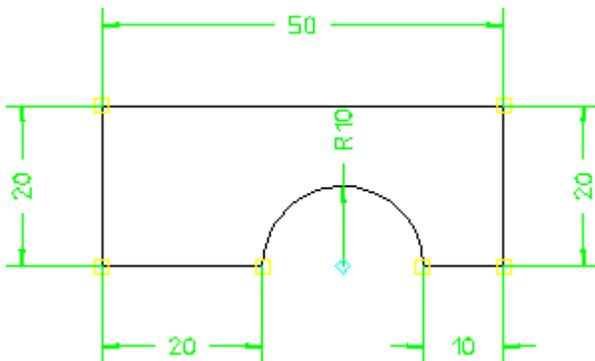
1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.

2. Click  **Deferred**.
3. Type the new value in the user input line and press ENTER.

Swapping Parametric Dimensions

The parametric dimensions generated by **Complete** may not be appropriate in all cases. In these cases it is possible to create a traditional dimension and then indicate that it should become a parametric dimension. For example, consider this drawing and the parametric dimensions that were generated with **Complete**:

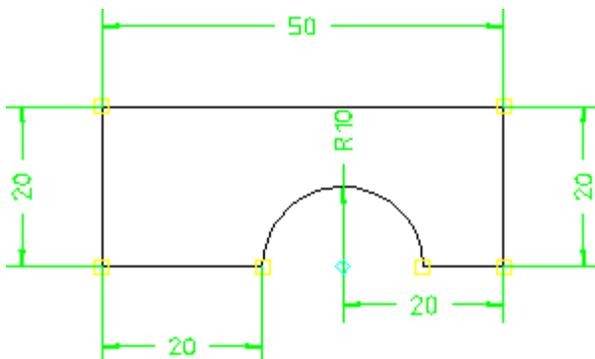
Figure 223. Parametric Dimensions Generated by "Complete"



If the center of the arc should be located from the right side, the buttons in the **Dimension** tab can be used to create an appropriate dimension. Then click 

Swap under  **Dimension** in the **Modify** group on the **Parametric** tab, and the newly-created dimension is selected to replace the parametric dimension generated by **Complete**. The resulting parametric dimensions are:

Figure 224. Parametric Dimensions After Swapping



Restrictions

In order for an exchange to occur both dimensions must control the same geometry. For instance, in the example above the height of the rectangle could not be exchanged for the distance from the center of the arc to the right edge.

Design Intent Capture

Roadmap

Read this chapter to learn how to create constraints as geometry is created. To learn how to assign constraints and parameters explicitly, read [Working with Parametric Design on page 633](#) instead.

Overview

Design Intent Capture generates and assigns parametric constraints as geometry is constructed. Constraint types depend on the method used to construct geometry. For example, when creating one line perpendicular to another line a perpendicular constraint will be created and associated with the two lines.

Design Intent Capture improves the speed, accuracy, and quality of parametric constraint definition by combining the separate steps of geometry creation and constraint assignment into one action.

Without Design Intent Capture constraints must be assigned after geometry has been created. This forces the designer to remember the relationships between geometric elements and record them by assigning constraints. Even worse, if the geometry was created by someone else the designer must try to reason backward and guess at the appropriate constraints. These problems are avoided by using Design Intent Capture.

The quality of constraints in the final parametric drawing is usually much better when Design Intent Capture is used compared with relying on **Complete** to generate all constraints. Quite possibly a drawing will be fully constrained when Design Intent Capture is used. But even if some additional constraints are required, **Complete** will have a much better starting point.

Flexibility

Although Design Intent Capture generates constraints, the designer is free to modify or remove any of them. The constraints are not history-based, so they may be modified regardless of the order in which they were created. Any constraint can be altered or removed without negating all of the work that occurred after the constraint was generated.

In other words, Design Intent Capture is a method of generating constraints — as are **Complete** and direct assignment. All of the Parametric Design module's constraint editing capabilities can be used on them. Refer to [Working with Parametric Design on page 633](#) for more information on editing constraints.

Enabling, Disabling and Inquiring

To enable Design Intent Capture:

- Type `UA_DESIGN_INTENT ON` in the user input line and press ENTER.

To disable Design Intent Capture:

- Type `UA_DESIGN_INTENT OFF` in the user input line and press ENTER.

To inquire the on/off status of Design Intent Capture use the arithmetic function `UA_GET_DESIGN_INTENT`. A value of 0 is returned if Design Intent Capture is disabled, and a value of 1 is returned if Design Intent Capture is enabled. For example, to display the current on/off status of Design Intent Capture type `DISPLAY (UA_GET_DESIGN_INTENT)` in the user input line and press ENTER.

Elements Affected

The following methods of geometry creation generate parametric constraints when Design Intent Capture is enabled:

- point
- line
- circle
- arc
- fillet
- chamfer
- construction line
- construction circle
- text
- equidistance
- overdraw

Constraint Creation

The Parametric Design module considers two guidelines when determining the appropriate constraints to generate with Design Intent Capture:

- construction technique
- CATCH information

Construction Technique

Different constraints will be generated depending on the method of geometry creation. The  **Parallel**,  **Perpendicular**,  **Horizontal**,  **Tangent by 2** buttons in the **Draw** group on the **Drafting** tab convey different intentions, and all will result in different constraints.

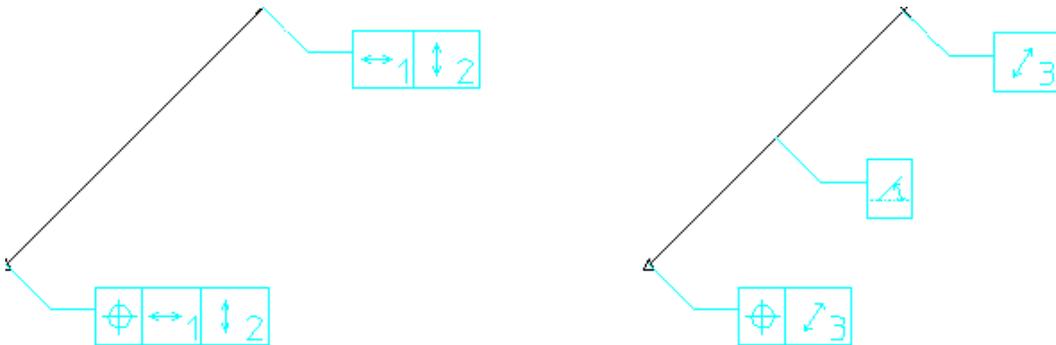
Even if two drawings look identical, they probably will have different constraints if different construction techniques were used to create them. For example, consider these two lines. Each is connected to a fixed point at one end. But one was created with  **2 Points** and the other was created with  **Angle & Length**. Both drawings look the same:

Figure 225. Lines Displayed Without Constraints



But the constraints are different:

Figure 226. Lines Displayed With Constraints



CATCH Information

Design Intent Capture also considers CATCH information when determining which constraints to generate. With Copilot enabled, the ability to catch to tangencies, etc. is much easier.

For example, assume Copilot is enabled and  **2 Points** is invoked. If the endpoints of the line are picked so that they are approximately tangent to two arcs, then two tangent constraints will be created.

 **Note**

Because different constraints are generated based on construction technique and CATCH information only advanced users should enable Design Intent Capture when macro files are being created or input.

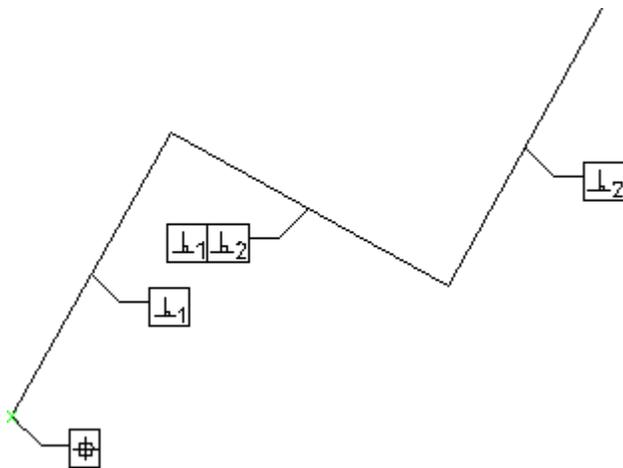
Operations on Existing Geometry

In addition to operations which create geometry, Design Intent Capture applies to operations that modify geometry. Constraints will be created, deleted, modified, or transferred appropriately for different modification operations.

Delete

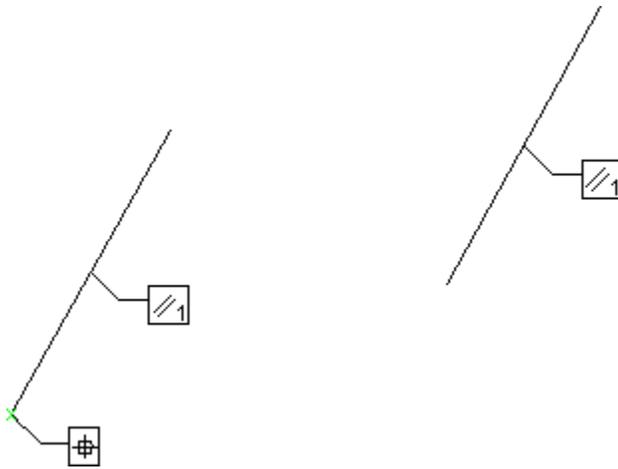
When geometry is deleted all relational parametric constraints will be transferred to other geometry whenever possible. For example, consider three perpendicular lines:

Figure 227. Perpendicular Lines with Constraints



If the middle line is deleted the perpendicular constraints associated with it will also be deleted. But a new constraint will be generated so that the parallelism between the remaining lines is kept:

Figure 228. One Perpendicular Line Deleted



Split

When elements are split constraints will be assigned to maintain consistency between the pieces. Each portion of a split line will be given a collinear constraint. Split circles and arcs will be assigned same size constraints.

In addition to the basic constraints for split lines and arcs, other constraints may need to be modified or assigned. For example, if a line with a size constraint is split, a distance constraint will be created and associated with the endpoints of the original line.

Merge

When elements are merged constraints will be combined whenever possible. For example, when line segments are merged, the collinear constraint associated with them will be removed (if it exists). Relational constraints between the segments and other elements will be associated with the new, merged line.

Modify Keep, Stretch Keep, Mirror Keep

Geometry copied with the **Modify** buttons (**Keep** check box clicked) or  **Stretch** (**Keep** check box clicked) will also have all associated constraints copied.

Note

No constraints will exist between the original and the new copies of the geometry.

Mirrored elements will be assigned constraints so that both portions of the geometry continue to have the mirror property. In other words, once geometry is mirrored with Design Intent Capture both portions will remain mirror images of each other — even as other modifications are made.

Modify NoKeep, Stretch NoKeep, Change_fillet

Design Intent Capture will attempt to modify constraint values without changing any constraint types. For example, the radius of a fillet or the distance between two elements may be changed, but a perpendicular constraint will not be changed to a collinear constraint. If Design Intent Capture is not able to calculate a new value for a constraint, the constraint will be eliminated.

Advanced Topics and Tips

Roadmap

The information in this section is mostly of interest to advanced Parametric Design users. If you are new to Parametric Design, skip this section for now and come back to it later.

Parametric Design has a number of useful features and applications that may not be immediately obvious to the casual user. This chapter describes several of these and provides some in-depth information on the workings of **Complete** and **Solve**.

Geometry Cleaning

Complete and **Solve** both have the ability to recognize and compensate for minor geometric inconsistencies in a part. Common problems such as lines which are slightly out of parallel, elements with zero length or radius, and small gaps between "tangent" elements can be recognized and corrected.

Note

The "cleanup" capabilities used by **Complete** and **Solve** group buttons are separate from the  **Clean Pnt**,  **Clean Duplicate** and  **Clean Stacked**. See [Working with Parametric Design on page 633](#) for more information on the Clean options.

Parametric Design handles most geometric inconsistencies by incorporating tolerances into several of the rules used by **Complete** to extract constraints from a part. If an element does not precisely fit the requirements for a given constraint,

but is within the tolerance, the constraint is assigned anyway. Once **Complete** is

done, you need only  **No Keep** in the **Solve** group on the **Parametric** tab to adjust these elements into the precise configuration given by the constraint.

For example, **Complete** can extract a **Horizontal** constraint for any line element with a slope that is $0 \pm$ the current angular tolerance (given by PD_AUTO_ANGLE_TOLERANCE). If the tolerance is 0.5, then lines with slopes of 0 ± 0.5 can be assigned **Horizontal** constraints. When the **Solve** group buttons are clicked, these lines are adjusted as needed to become truly horizontal.

The following tolerances are used by **Complete**:

PD_AUTO_ZERO_DISTANCE_TOLERANCE: Specifies the absolute tolerance used by **Complete** to compare a distance to zero. This tolerance is used to detect coincident points, collinear lines and **Point on** constraints. Must be greater than or equal to 0. The initial setting is 0.000001 mm.

PD_AUTO_SAME_DISTANCE_TOLERANCE: Specifies the absolute tolerance used by **Complete** to compare two non-zero distances. Used in several areas; to check whether two circular elements have the same radius, for example. Must be greater than or equal to zero. The initial value is 0.000001 mm.

PD_AUTO_ANGLE_TOLERANCE: Specifies the absolute tolerance used to compare angles. This tolerance is applied, for example, when checking whether two lines are parallel or a single line is horizontal. Must be greater than or equal to zero. The initial value is 0.000001 radians.

PD_AUTO_TANGENT_TOLERANCE: Specifies the absolute distance tolerance used by **Complete** to check whether two elements are tangent. Applied only if one of the extracted elements is circular. The tolerance is used to compare the radius of the circular element to the distance from the circular element center point to the other element. The number supplied must be greater than or equal to zero. The initial setting is 0.000001 mm.

By default, these tolerances are set very small to ensure that Parametric Design only compensates for true anomalies in the part and does not remove or "adjust away" small details that are important to you. Setting a tolerance to 0 effectively eliminates its effect on **Complete**. Depending on the type of parts you normally work with, you may want to experiment with different values for each of these tolerances until you arrive at a set of tolerances that provides maximum cleaning without removing important details. Tolerances may be redefined on the Creo Elements/Direct Drafting command line, and the default values can be changed by editing the pd_def.mac file.

An additional tolerance, PD_RESOLVE_MERGE_TOLERANCE, is used directly by the **Solve** group buttons to merge coincident points and remove very small ("zero length") elements from variations. Points that are within this tolerance are merged, thus removing duplicate points and eliminating elements that are smaller than the tolerance. The default, which is defined in pd_def.mac, is 0.000001 mm.

⚠ Caution

It is usually a good idea to disable tolerance-based merging in motion-study or kinematics applications where several entities are moved around in close proximity to one another. In these applications, it is often the case that two entities which are intended to remain separate come into close enough contact that a point on each will fall within the merge tolerance. If merging is on, the two entities will be effectively fused together, preventing further independent motion. To disable merging, set the tolerance to -1.

A Simple Sketch Input Implementation

The preceding discussion of geometry cleaning has assumed that the tolerance capabilities of the solver will be used to correct for small deficiencies in the input geometry. Suppose however, that you expect substantial inconsistencies in a part and that you don't care if you have to adjust it substantially to clean it up. A "sketch input" application is a common situation where these assumptions would be true. In these situations, you can use Parametric Design with relatively large tolerances to "square up" a quickly-drawn part.

The following macro implements a simple "sketch input" application using Parametric Design:

```
DEFINE SquareUp                               {A simple drawing
straightener}

{Accept linear and angular tolerances as arguments}

PARAMETER LinearTolerance

PARAMETER AngularTolerance

{Set adjustment tolerances to the supplied values }

PD_AUTO_ZERO_DISTANCE_TOLERANCE LinearTolerance
PD_AUTO_SAME_DISTANCE_TOLERANCE LinearTolerance
PD_AUTO_ANGLE_TOLERANCE          AngularTolerance
PD_AUTO_TANGENT_TOLERANCE        LinearTolerance
PD_RESOLVE_MERGE_TOLERANCE       LinearTolerance
```

{If AUTO must create new points, make sure we can delete them later}

```
ADD_CURRENT_INFO "DELETE" END
```

{Load the entire current part into the zone}

```
PD_ZONE_ADD ALL {ZONE Add Mult ALL}
```

{Generate constraints; tolerances are applied}

```
PD_RESOLVE GENERATE {AUTO}
```

{Resolve conditionally. If the preview is OK, accept the change}

```
PD_RESOLVE PREVIEW {SOLVE Preview}
```

```
WINDOW FIT
```

```
READ 'Please enter CONFIRM (c) or UNDO (u)' DEFAULT 'c'  
Accept_flag
```

```
IF ((Accept_flag = 'c') OR (Accept_flag = 'C'))
```

```
PD_RESOLVE REPLACE {SOLVE NoKeep}
```

```
ELSE
```

```
END
```

```
END_IF
```

{If AUTO had to generate ref. points to constrain, remove them now}

```
DELETE SELECT INFOS "DELETE" ALL END
```

```
{Cleanup: remove constraints, zone data, DELETE info. reset
tolerances}
```

```
PD_FREE NOBAN ALLTYPES CONFIRM

PD_ZONE_REMOVE ALL

CHANGE_CURRENT_INFO "DELETE" "" END

PD_AUTO_ZERO_DISTANCE_TOLERANCE 1E-06

PD_AUTO_SAME_DISTANCE_TOLERANCE 1E-06

PD_AUTO_ANGLE_TOLERANCE 1E-06

PD_AUTO_TANGENT_TOLERANCE 1E-06

PD_RESOLVE_MERGE_TOLERANCE 1E-06
```

```
{Display what I've made!}
```

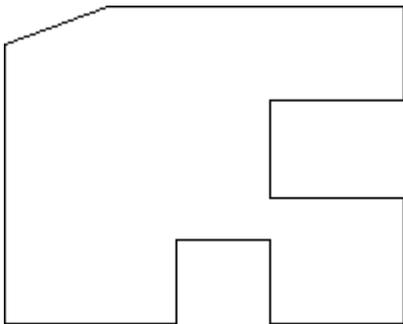
```
REDRAW
```

```
END_DEFINE {of SquareUp}
```

To get a feel for how this macro works, copy the text above into a file and then input the file to Creo Elements/Direct Drafting.

Now quickly create the part below. Use  **Polygon** under  **Line** in the **Draw** group on the **Drafting** tab with the Creo Elements/Direct Drafting catch-mode set to **OFF**. Try to reproduce the figure as shown, but don't worry about inconsistencies in your part.

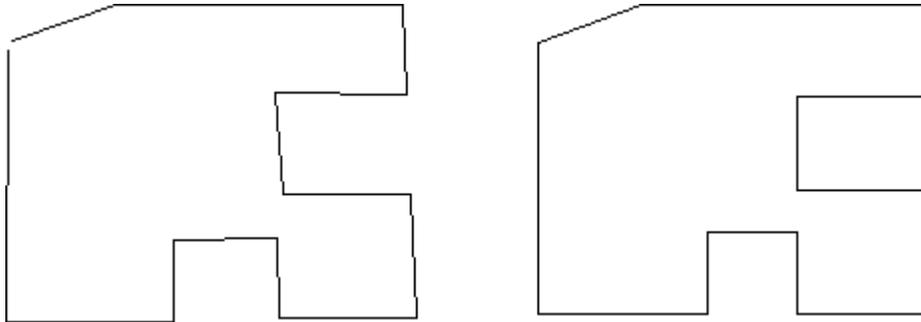
Figure 229. Sample Polygon



Now type SquareUp 2 5 in the user input line and press ENTER.

If all went well, Parametric Design has cleaned your "sketch." Specifically, gaps of 2 mm or less should now be closed. Lines that were within +/- 5 degrees of horizontal or vertical should now be "squared up." Also, lines that were not quite collinear (+/- 2 mm) should now be snapped into place, [Figure 230. A Cleaned Sketch on page 703](#).

Figure 230. A Cleaned Sketch



If the first pass at this didn't work for you (remember, `SquareUp` is a very simple macro that will handle many, but certainly not all cases!), reject the results and experiment with different values for the length and angle tolerances.

As you experiment with `SquareUp`, it may be helpful to try it with severely flawed sketches and to give "unreasonable" values for the linear and angular tolerances to learn the strengths and weaknesses of this technique.

Zone Gymnastics

Parametric Design's zone mechanism is implemented using Creo Elements/Direct Drafting infos. The info text `PD_ZONE` is used to specify which drawing elements are in the Parametric Design zone. Any drawing element in the current part that has the info text `PD_ZONE` attached to it is included in the zone. By treating the zone as simply a collection of element infos, several powerful options for working with the zone become available. The following sections contain a few ideas.

Automatic Inclusion

As shipped, the `pd_def.mac` file contains the string:

```
ADD_CURRENT_INFO "PD_ZONE" END
```

This causes all new geometry created during your Creo Elements/Direct Drafting session to be automatically included in the zone (i.e., the `PD_ZONE` info is added to all new geometry). You can turn off the auto-inclusion feature at startup by removing this line from `pd_def.mac`. To turn off auto-inclusion from within Creo Elements/Direct Drafting, you can type:

```
CHANGE_CURRENT_INFO "PD_ZONE" ""
```

 **Note**

These functions will not affect geometry input from an MI file. If the contents of an MI file were in the zone when they were saved to the file, they will be put into the zone when the file is restored. If the contents were not in the zone when the file was created, they are not placed into the zone when the file is restored.

Alternate Add/Remove Schemes

You can use **Info** commands instead of the screen menu to add or remove elements from the zone. This can be useful when using Parametric Design from within a macro. Typical commands might be:

Command	Description
ADD_ELEM_INFO "PD_ZONE"	Add selected elements to the zone
CHANGE_ELEM_INFO "PD_ZONE" ""	Remove selected elements from the zone

Pre-Define Multiple Zones

In an evolving, complex part, you may find that several different areas are frequently being modified with Parametric Design. Each time you want to work on a different area of the part, you need to redefine the contents of the zone. If your part is truly complex, this may be a time-consuming task. If you want to make sure that the same elements are always included in the zone each time you work in a particular area, you'll have to be very careful when you assign elements to the zone. This task may become even more difficult and error prone if several people are working on the same part.

One solution to this problem is to use infos to pre-define several "quasi-zones" in a part and then to activate each one as needed. Suppose, for example, you have a part with three different areas that must regularly be modified with Parametric Design. Try the following strategy:

- First, clear all zone infos from the part:

```
CHANGE_GLOBAL_INFO "PD_ZONE" ""
```
- Next, assign to all the elements in the first area an arbitrary info. Let's call it "AREA_1":

```
ADD_ELEM_INFO "AREA_1" (select elements)
```

-
- Repeat this step for the second and third area, giving each its own info: ADD_ELEM_INFO "AREA_2" (select elements) ADD_ELEM_INFO "AREA_3" (select elements).

Now that each group of elements has a unique info associated with it, you can easily "activate" each group for Parametric Design by swapping its info text for the PD_ZONE info text and "deactivate" it by restoring the old info:

- To load the elements in "AREA_1" in the zone: CHANGE_GLOBAL_INFO "AREA_1" "PD_ZONE".
- To remove "AREA_1" elements from the zone and load "AREA_3" elements into the zone: CHANGE_GLOBAL_INFO "PD_ZONE" "AREA_1" CHANGE_GLOBAL_INFO "AREA_3" "PD_ZONE".
- To remove "AREA_3" elements from the zone: CHANGE_GLOBAL_INFO "PD_ZONE" "AREA_3".

Parametric Control of Splines

Creo Elements/Direct Drafting currently supports two types of spline entities: rational cubic splines (C-splines), which are created with the SPLINE function, and B-splines, which are created with the BSPLINE function. Earlier versions of Creo Elements/Direct Drafting only supported C-splines. Parametric Design can be used to parametrically control the shape of any spline, but the procedure for constraining C-splines and the newer B-splines differs somewhat. The following two sections give some tips and techniques for controlling each type of spline.

C-Splines

To parametrically control the shape of a C-spline, you first place point elements on the data points used to create the spline and then assign the appropriate constraints to these points to actually control the shape of the spline. Any constraint that can be assigned to a point can be used.

To find the location of the C-spline's data points, turn on vertices with SHOW VERTEX ON. Before assigning point elements to the spline data points, you should also make sure that splitting is turned off.

Note

By default, Creo Elements/Direct Drafting converts all rational splines in a drawing into B-splines when you load it. The conversion will invalidate any constraints that you have assigned to the data points of these rational splines. Refelem constraints which are assigned directly to spline elements are not invalidated, and will be carried over from the rational spline to the new B-spline.

To prevent spline conversion and the subsequent invalidation of constraints on rational splines, you must enter `SPLINE_CONVERSION OFF` prior to loading any master part that contains constrained rational splines. Alternatively, you could include this line in the Creo Elements/Direct Drafting `pd_def` file to permanently disable spline conversion.

B-Splines

Unlike C-splines, which are controlled via their data points, the shape of a B-spline can only be controlled by constraining its control points. To parametrically set the shape of a B-spline, you must first display its control polygon, either with the `CNTRL POLY` command on the screen menu, or by entering `SHOW_CPOLY ON` and selecting the B-spline. With the control points displayed, use same method as for C-splines to constrain them: place point elements on each control point and then constrain those point elements. At solve time, as the control points move, the shape of the spline will also change.

If you assign constraints only to the data points of a B-spline and it appears that they are causing the spline to change shape, it's likely that splitting was on when you placed the point elements on the data points and you are now manipulating multiple B-splines!

Rigid Bodies: Applications and Tips

Rigid bodies are useful for a wide range of applications. The list below presents a few ideas.

- Motion studies of linkage assemblies becomes much simpler. Make each link a rigid body. Then use constraints to describe the motion between the links. Your model will not be cluttered with constraints for the links, so you can concentrate more on the interactions between links.

A feature of rigid bodies that is especially useful for motion studies is the fact that a single element can legally belong to more than one rigid body. A single point element included in two rigid bodies acts much like a hinge between

them, while a single construction line works much like a slider. An arc or line segment shared by two rigid bodies serves to "weld" them together.

- Parametrics can now control (to some degree) subparts and/or shared subparts. Collect each subpart into a rigid body along with some geometry from the current part. Construction geometry is often a good choice for this. You can control the position and orientation of each subpart through constraints on the construction geometry.
- Complicated parts often have large amounts of geometry which you want to move as a group. Because the automatic constraint generator has a poor notion of locality, it is usually simpler and more predictable to identify these groups of entities manually.
- Control hatch angle. Suppose you want a hatching to remain at a given angle to some line. Create a construction line collinear to this line (and constrain it to be collinear), then make a rigid body out of the construction line and the hatching. Use the construction line instead of the original line in the rigid body so that you don't prevent the original line from changing size by including it in the rigid body.
- Control the location and orientation of symbol subparts.
- Provide a form of model cleaning without modifying geometry. Stacked elements, unclosed polygons, etc. can wreak havoc on parametric applications.

The usual approach to solving these problems is to use the  **Clean Pnt**, 

 **Clean Duplicate**, and  **Clean Stacked** to merge points and remove stacked elements. In cases where it is not possible or desirable to clean the original geometry, you may be able to use rigid bodies to isolate problem geometry from the solver. Simply collect problem areas into rigid bodies so that the solver does not need to examine individual elements too closely. Of course, this approach may limit what you can accomplish with parametrics, as the contents of the rigid bodies can only be moved and rotated, not resized.

Here are a few potential trouble spots that you should keep in mind while working with rigid bodies:

- Be careful when creating a rigid body that contains only subparts. While these are perfectly valid rigid bodies, remember that you cannot assign constraints to subparts. As a result you would not be able to control the position or rotation of the rigid body via constraints. It will remain in its original position in all subsequent variations.
- Similarly, be careful when creating rigid bodies that contain just a subpart and a single point element from the current part. Such a rigid body can be translated via constraints, but its rotation cannot be changed.

-
- A subpart may be assigned to more than one rigid body. At solve time, however, it will always remain associated with one of the rigid bodies to which it was assigned. All other rigid body associations will be ignored.
 - Finally, consider the case of a rigid body that contains only a subpart and a construction line. In this case it is possible to control the rotation of the rigid body and translations that are perpendicular to the axis of the construction line, but not translations along the axis of the construction line. The subpart will retain its original position along the axis of the construction line regardless of subsequent rotation and off-axis translation.

Implementing Replication in Parametric Design

A common requirement when creating a family of parts is to have a particular geometric feature replicated a variable number of times, depending on the overall size or configuration of the surrounding geometry in each variation. A circular plate, for example, might be expected to have 8 holes if its radius is 4 inches, and 12 holes if the radius is 6 inches.

While Parametric Design provides no functionality to perform such modifications directly, the following procedure often does the trick:

1. Define the zone to be all of the geometry in the part except for the portion that is to be replicated. Constrain the geometry in the zone as if you were setting up a master part.
2. Create some additional parameterized constraints that can be used to query the result of a variation. These can be **Refpoint**, **Dimension**, **Distance**, **Angle**, **Size**, or **Slope** constraints. Assign these constraints so that they return the values needed to determine the number of replications needed.
3. Write a macro to query the necessary parameters, compute the number of replications from this data, and create those replications in the correct location.
4. Finally, write a macro which first invokes `PD_RESOLVE` to solve for the geometry in the zone, and then calls the macro defined in the previous step to take care of the replications.

A number of alternative strategies can be derived from this procedure. For example, if both the number of replications and the size of the replicated geometry needs to vary, consider using two different zones, one for the surrounding geometry and one for a single instance of the replications. In this case, the final macro might have the following logic:

1. Run `PD_RESOLVE` for the surrounding geometry.
2. Use `PD_PARAM_INQ` to read out the results.
3. Compute values for the input parameters for the instance of the replication geometry from the result you've just obtained.

4. Change the current zone to the replication geometry.
5. Use PD_PARAM_FIX calls to load the input parameters needed for the replication geometry.
6. Run PD_RESOLVE for the replication geometry.
7. Use the buttons in the **Modify** tab to move, rotate and copy the replication geometry as needed.

Allowing Rotation

It is often desirable for variations of a part to be able to rotate about a given point as a condition for their creation. The simplest way to allow a part to rotate is to define the reference constraints on the master part to be a reference point and a line slope. Usually it is best if the reference point is on the line whose slope is being constrained. After applying the reference constraints, constrain the rest of the part without using slope constraints (i.e., no **Slope**, **Horizontal**, or **Vertical** constraints. If you use **Complete** to generate constraints, make sure to check for any system-generated slope constraints, and change these to angle-type constraints.

Now by simply changing the value of the constrained slope the entire part will rotate about the reference point.

Working with Multiple Parts

Parametric Design constraints and parameters are all local to a given Creo Elements/Direct Drafting part. Thus, subparts can be stored independently from the rest of a drawing with their complete parametric definition, and parameterized subparts can be loaded into a drawing without corrupting the constraint and parameter data for other parts already in the drawing.

There are two situations, however, in which the parametric data from more than one part can come into contact. The interactions that occur are worth noting:

You can explicitly set up interrelationships between parts by writing macros like the one given in the "Macros" example in [Examples on page 715](#). This example shows how a parameter in one part can have its value depend on the resulting geometry in another part. Since a parameter can refer to an angle, slope, distance, size, or dimension, and since any number of points can be parameterized for output query, this method provides a very general solution to the problem.

A second (often less desirable) form of interaction can occur as a result of the

Creo Elements/Direct Drafting  **Smash** and  **Gather** buttons.  **Smash** and  **Gather** have the following behavior:

-
- If you use  **Gather** without copying a subset of the constrained elements in a parameterized part, all of the constraint and parameter data for that part is lost.
 - If you  **Gather** all of the constrained elements in a part or  **Smash** one or more subparts, then the constraint and parameter data will be kept. Note, however, that if the same parameter name exists in two or more of the parts being merged, the results can be unpredictable. Generally, if there is a conflict between parameter names, the parameter assigned to the part that is highest in the parts tree is maintained. If both parts are at the same level, one or the other will be selected, but it is impossible to predict which.

Using Modify With Parametric Design

The Creo Elements/Direct Drafting **Modify** tab buttons can generally be used along with Parametric Design to modify geometry. The buttons on the **Modify** tab can be used successfully between the time that a part is constrained and the time that **Solve** is issued, but modifications during this time also affect the results of **Solve**. These effects may be surprising if you are not expecting them, but can be very useful once understood.

- If a parameterizable constraint (that is, **Angle**, **Size**, **Distance**, **Slope**, or **Dimension** constraint) is assigned without giving an expression or parameter name, then the value of the constraint is determined at solve time. Thus, any modifications you make to such elements will be recognized by **Solve**. For example, if you assign a valueless **Size** constraint to a 7 inch line, and then **Stretch** that line to 12 inches prior to solving, **Solve** will keep the line at 12 inches.
- **Collinear**, **Parallel**, **Perpendicular**, and **Tangent** constraints all have orientation characteristics which can be affected by **Modify**.
 - The angle between two **Parallel** or **Collinear** lines can be 0 or 180 (see the explanation of **Slope** constraints in [Constraint Types on page 655](#)). **Solve** will use whatever angle is closest to the actual positions of the lines at solve time.
 - Similarly, the angle between two **Perpendicular** lines can be 90 or 270 degrees. The preferred angle is the angle that it is closest to at solve time.
 - If a line and circle are constrained **Tangent**, **Solve** attempts to find a location for the circle center point on the same side of the line that it started out on at solve time.
 - If two non-concentric circles are constrained **Tangent**, there are three possible solutions. Either the first circle will be inside the second, the second circle will be inside the first or both circles will be tangent

externally. **Solve** attempts to maintain the particular relationship that is in place at solve time.

In all of the above cases, **Modify** can be used to change the location of the constrained elements and thus change **Solve**'s preferred solution, even though the constraints haven't changed.

- Geometry constrained as reference elements can be modified at any time before solving. **Solve** always leaves such elements in their current position at solve time.

Modifying Constraint Icons

Parametric Design constraint icons are simply text elements whose contents and location are managed by the software. While there is nothing to prevent you from modifying the location, text attributes, or contents of the icons with Creo Elements/Direct Drafting, doing so has several implications that you should be aware of:

- Each constraint icon string is derived from the constraint and from the applicable icon viewing commands (for example, **Show** and **Clear**). Modifying the contents of the string, or deleting the string altogether, doesn't alter the underlying constraint in any way.
- Copying an icon string or placing the leader line on a different element doesn't copy the constraints to that element. In fact, a copy of an icon string is not recognized by Parametric Design and becomes just another text element that can be added to the zone and constrained!
- Any change in the attributes of an icon string, such as slant, will be forgotten when the entire icon string is cleared from the screen. The next time the icon string is constructed it will use the default settings for new text elements.
- The constraint icon characters are part of the `hp_symbols` font. These characters are inserted into locations 65-96 of `hp_symbols` when Parametric Design is enabled. You should not attempt to use these locations either before or after Parametric Design is enabled. Unpredictable results can occur.

Automatic Constraint Generation

To extract constraints from an existing part, **Complete** uses the Parametric Design solver to apply a set of extraction rules to the geometry in the zone. Understanding how these rules work will help you to anticipate what kinds of constraints are generated by **Complete**.

When you select **Complete**, the solver first scans the geometry for existing constraints. If these fully constrain the part, **Complete** is done. If the existing set of constraints does not fully constrain the part, the solver then starts to evaluate the

under-constrained members of the part and assign new constraints to them according to the rules listed below. Each time a new constraint is extracted, it is added to the list of known constraints and the part is checked again to see whether or not it is now fully constrained. As soon as it is, the process stops. Otherwise, the solver continues to go down the list of rules, adding constraints and checking until the part is fully constrained. The solver will not go on to a new rule until all possible constraints have been extracted according to the current rule. A rule will not be applied to elements that have the **Ban** flag set for the constraint type assigned by the rule. The rules and their basic functions are (in order of application):

Rule	Action
Special Symmetry lines	If PD_AUTO_SYMMETRY is on, find all lines that have the auto-symmetry linetype and color and assign them Symmline constraints.
Collinear Lines	Find collinear lines and assign them Collinear constraints.
Coincident Points	Find coincident points and constrain them together.
Points On Geometry	Find suitable elements and assign them Point on constraints.
Mirrored Elements	If symmetry lines are defined, find mirrored elements and assign them Mirror constraints.
Dimensions	Constrain any free dimensions to their current value.
Fillets	Find arcs that qualify as fillets and assign them Fillet constraints.
Parallel Lines	Find parallel line elements and assign them Parallel constraints.
Perpendicular Lines	Find perpendicular elements and assign them Perpendicular constraints.
Tangent Elements	Find tangent linear or circular elements and assign them Tangent constraints.
Horizontal Lines	Assign Horizontal constraints to horizontal elements.
Vertical Lines	Assign Vertical constraints to vertical elements.
Mirrored Points	If symmetry lines are defined, find mirrored points and assign them Mirror constraints.
Same Radius	Find circles with the same radius and assign Samesize constraints.
Free Angle	Find elements with free slopes and assign Angle constraints.
Free Slope	Find elements with free slopes and assign Slope constraints.
Free Radii	Find circles with free radii and assign Size constraints.
Free Length	Find lines with free lengths and assign Size constraints.

Rule	Action
Free Elements	Find any remaining free elements and constrain them with Distance and/or Refelem constraints.
Free Parameter	Look through the parameter table. For all parameters that do not have a set value, assign the current value.

When extracting constraints from the part, the solver must determine the location and connectivity of each element in order to decide if that element is a suitable candidate for the constraint type currently being assigned. The user can specify several tolerances for the solver to use when determining an element's location and connectivity. These tolerances, described in detail in [Geometry Cleaning on page 698](#) above, allow the solver to handle minor inconsistencies that may occur in the part.

Finally, after all the geometry has been examined and all new constraints have been extracted, **Complete** goes back through the part looking for redundant or unused constraints. These are removed. For example, if a line element is found that is constrained horizontal and is also constrained collinear with another line that is constrained horizontal, the horizontal constraint can be freed from the first line.

A Short Description of the Solver

The Parametric Design solver is a constraint solution kernel. The input to this kernel is a list of the geometric elements in the zone and the set of topological and metric constraints assigned to the geometry. Using the geometry list and constraints, the kernel can solve for the (new) location of the geometry.

The kernel is really just a smart sequencer of solution steps. Each step determines, or partly determines, some geometric entity (point, line, etc.) This is done by applying one (or more if necessary) constraints and some adjacent fixed entities to make a change in some other entity. When a change is made to an entity, constraints associated with that entity are scanned to see if one or more might be applicable.

Once the solution sequence has been determined, an expanded set of algebraic closed-form algorithms is invoked, rather than an iterative numerical technique. These algorithms employ simple geometrical constructions, such as curve/curve intersector. The kernel is designed for speed and robustness, and to minimize calculation error.

One of the limitations in the kernel is that coupled or circular constraints are not resolved, since they usually imply a simultaneous system of equations, which the kernel does not attempt to solve. In these situations, the kernel calls the automatic constraint generator, which proposes an alternative set of constraints.

How Expressions are Evaluated

User expressions for parameters are evaluated when all of the independent parameters are known. When it is time to evaluate an expression, **Solve** constructs the following macro:

```
DEFINE Pdeval
  PARAMETER <independent parameter name 1>
  :
  PARAMETER <independent parameter name N>
  <user expression>
END_DEFINE
```

The dependent parameter is then assigned to the value of

```
Pdeval <value of independent parameter 1> ...
```

For example, if the expression for parameter A is (B+10) and at some point in the solution process B becomes 6, then Pdeval will be:

```
DEFINE Pdeval
  PARAMETER B
  (B+10)
END_DEFINE
```

And A is assigned to be the result of

```
Pdeval 6
```

A syntax error can occur during the evaluation. When this happens, a message is displayed indicating which dependent parameter has the faulty expression and **Solve** is terminated. Since the offending expression is the last expression to be evaluated, you can view it by running `EDIT_MACRO Pdeval` immediately after the failure occurs. This procedure may point out problems that might otherwise escape attention, since there is very little syntax checking done when the expression is defined.

For example, if the above sample expression were (rad+10) then the above macro would be:

```
DEFINE Pdeval
  (rad+10)
END_DEFINE
```

which fails because `rad` is a keyword and keywords are not recognized as parameters, even if there was a parameter called `Rad`.

Invisible Geometry

As constraints are created the Parametric Design module may need to create points and construction lines to serve as reference geometry. For example, when Design Intent Capture is enabled and LINE PARALLEL is invoked construction lines are created between the original and new lines, so that the two lines will remain both parallel and the same length.

Invisible Geometry is managed completely by the Parametric Design module. Whenever delete, modify, or stretch operations are performed on real geometry, the necessary changes are made to invisible geometry.

Viewing

Invisible geometry is displayed whenever an associated constraint is displayed. Typically constraints are displayed in the **Display** dialog box.

Viewing invisible geometry can be helpful in understanding the relationship between constraints and geometry.

Attributes

The only two types of invisible geometry that are created are points and construction lines.

`PD_NEW_POINT_VISIBILITY` and `PD_NEW_C_LINE_VISIBILITY` determine whether or not invisible geometry created by the Parametric Design module should be displayed at all times or only when constraints are displayed. Allowable qualifiers are `VISIBLE` and `INVISIBLE`.

Other functions that set display attributes of invisible geometry are:

- `PD_NEW_POINT_COLOR`
- `PD_NEW_C_LINE_COLOR`
- `PD_NEW_POINT_LINETYPE`
- `PD_NEW_C_LINE_LINETYPE`

Examples

Roadmap

Work through the examples in this chapter to familiarize yourself with the Parametric Design software. If you have not yet done so, you should read [Introduction on page 624](#) before continuing. If you are not familiar with Creo Elements/Direct Drafting, we also recommend that you read through [Creo Elements/Direct Drafting User's Guide](#) before continuing.

Introduction

This chapter contains several examples that will help you become familiar with Parametric Design tools and techniques.

All the examples assume that you are at a workstation, have Creo Elements/Direct Drafting up and running, and that Parametric Design is installed.

The first example introduces all the Parametric Design tools, proceeding step-by-step through an extended session with a simple drawing. We strongly recommend that you spend a little time going through this example before going on to the others. Subsequent examples present particular techniques or applications that you are likely to encounter as you use the software. These examples assume that you are comfortable with the Parametric Design interface.

The Creo Elements/Direct Drafting parts used in the examples are included online with the software. They are installed as uncompressed MI-format files in the `pd_demos\` directory. Several of the examples depend on information in these files, so we encourage you to load each file as you work through the corresponding example.

The examples were chosen to be simple, and yet introduce you to the breadth and depth of this product. After going through these examples and carefully considering the questions at the end of each one, you should have developed a feel for nature of parametric design.

It would be a mistake to hope that you will be an expert Parametric Design user after just using these examples. In the real world, geometry is often more complicated. Models are underdimensioned, have unwanted symmetries, suffer from geometric inaccuracies, or have other "features" that are difficult for the automatic constraint generator to interpret correctly.

There are two rules of thumb that apply when using Parametric Design:

- If it isn't clear to you, at least in general, how a part should change in response to a modified dimension, then AUTO will likely have trouble as well.
- The better you understand the interplay between the various constraint types, and the rules governing each constraint type, the more effectively you'll be able to use Parametric Design.

One very good way to become expert in the science of constraining a part, is to practice using manual constraints on parts of increasing complexity. Use the automatic constraint generator to see how it would solve a particular problem, but make sure you understand the constraints that are generated. Once you fully understand how the various constraint types can be used and you get used to using the debugging tools, then begin to rely more on the automatic constraint generator.

Example 1: Introductory Design Example

This example introduces the tools and working methods that the Designer or Engineer must use to constrain parts and generate variations with the Parametric Design module. During this first example, you will:

- Place the part in the zone.
- Fully constrain the part with manual and automatic methods.
- Examine the constraints.
- Modify the constraints needed for a variation.
- Solve for variations.
- Parametrize constraints to generate more useful variations.
- Add new geometry to a constrained part.

You should come away from this demo with a clear understanding of the tools and basic working techniques for constraining parts — information that you can apply in subsequent examples, which focus in more depth on particular applications of Parametric Design.

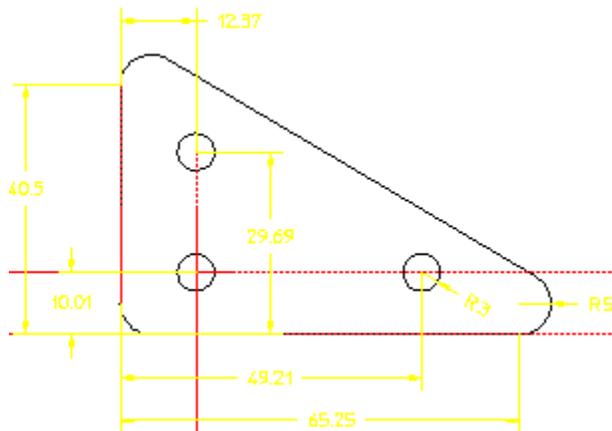
As with all of the examples in this section, the part for this example is supplied on line so that you can focus on parametric design instead of on drawing — after all, the main goal of Parametric Design is to reduce the time you spend drawing!

Load the Part

The part shown in this example is supplied with Creo Elements/Direct Drafting. To load this part:

1. Click **File** ►  **Open** to open the Creo Elements/Direct Drafting File Browser.
2. Double-click the `pd_demos` directory.
3. Select the `demopart01.mi` file and click **Open**.

Figure 231. Demopart01



The important features of this part for parametric design are as follows:

- It is an ordinary Creo Elements/Direct Drafting part.
- It is fully dimensioned. The length and height are specified, as is the radius of one corner. The location and radius of the holes is also given. Generally, a part with full dimensioning works best for Parametric Design.
- It is "clean." There are no duplicate or stacked elements. Such inconsistencies can lead to unexpected results. Parametric Design has several commands that find and fix inconsistencies in a part, but you won't have to use them right away!
- Construction lines have been used to mark the "origin" of the part as well as the centerlines of the holes. Construction geometry, if available, is often very useful to the Parametric Design solver. When possible, you should include construction geometry to mark important features of a part: major axes, lines of symmetry, centerlines of circles, and so on.

Include the Part in the Zone

Parametric Design only modifies elements included in the zone. To include the entire part in the zone:

1. Click **Parametric** and then, in the **Modify** group, click  **Zone**.
2. Click  **Add Multiple**.
3. Type **ALL** in the user input line and press ENTER.
All elements are highlighted as they are added to the zone.
4. Click  **OK**.

The entire part is now included in the zone.

Fully Constrain the Part

Before you can use the Parametric Design solver to generate variations from an existing part, each element in the zone must have one or more constraints assigned to it. Enough constraints must be assigned to tell the solver exactly what to do with each element in the part. If elements in the zone are not fully constrained or are incorrectly constrained, the solver will not be able to generate the variation.

The best way to assign a full set of constraints is to first constrain all the elements in the part in their current configuration. After the part is fully constrained in its current configuration, you can modify constraints as needed to generate a variation.

Both manual and automatic options are available for assigning constraints. Typically, the manual options are used to assign a few important constraints like reference elements, and then the **Solve** group buttons generate the remaining constraints required to fully constrain the part.

Manually Fix Reference Elements

It is always a good idea to "anchor" a part by manually constraining one or more elements from moving. This helps to give the software a frame of reference for automatically assigning constraints later on. For this example, fix the lowest horizontal and leftmost vertical construction lines in place with **Refelem** constraints to anchor the part.

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Refelem**.
4. Click **Apply**.
5. Click the lowest horizontal construction line (collinear with the bottom edge of the plate).
6. Click the left-most vertical construction line (collinear with the left edge of the plate).
7. Click  **OK**.

Notice the reference-element icons. The leader lines of these icons are dotted so that you know they are assigned to construction geometry and not to the edges of the triangular plate. Each constraint type has its own unique icon. See [Figure 206. Constraint Icons on page 656](#) for a complete list of the constraint-type icons.

Modify Constraints

To generate a variation that is different from the original, you must change one or more constraints from their current values to the values you want to see in the variation.

1. In the **Generate Constraints** dialog box, click **Assign** in the **Action** area.
2. In the **Type** box, select **Dimension**.
3. Click **Apply**.
4. Click the dimension text that gives the height of the triangle (40.5).
5. Type 55 and press ENTER.
6. Click the dimension text that gives the length of the triangle (65.25).
7. Type 55 and press ENTER.
8. Click  **OK**.

The dimension text is updated.

Automatically Generate the Remaining Constraints

Rather than assign the remaining constraints by hand, which would take considerable time, use the **Complete** option provided in the **Generate Constraints** dialog box. This option evaluates the part and assigns sufficient constraints to fully constrain it in its current configuration.

1. In the **Generate Constraints** dialog box, click **Complete** in the **Action** area.
2. Click **Apply**.
3. Click  **OK**.

Solve

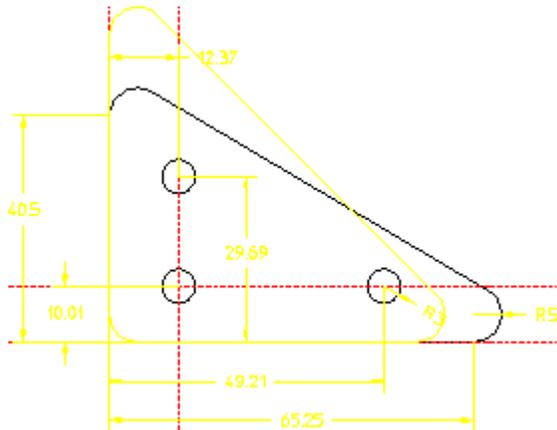
Now use the buttons in the **Solve** group to generate the variation.

1. Click **Parametric** and then, in the **Solve** group, click  **Preview**.
The magenta outline shows how the variation will be created without disrupting the current part (see [Figure 232. Preview Mode on page 721](#)).
2. Click  **OK**.
3. Click **Parametric** and then, in the **Solve** group, click  **No Keep**.

Your original part is now replaced with the variation. Notice that the dimensions you assigned to be 55 earlier are now indeed 55. The hole geometry has not moved.

4. Click  **Undo** to restore your original part.

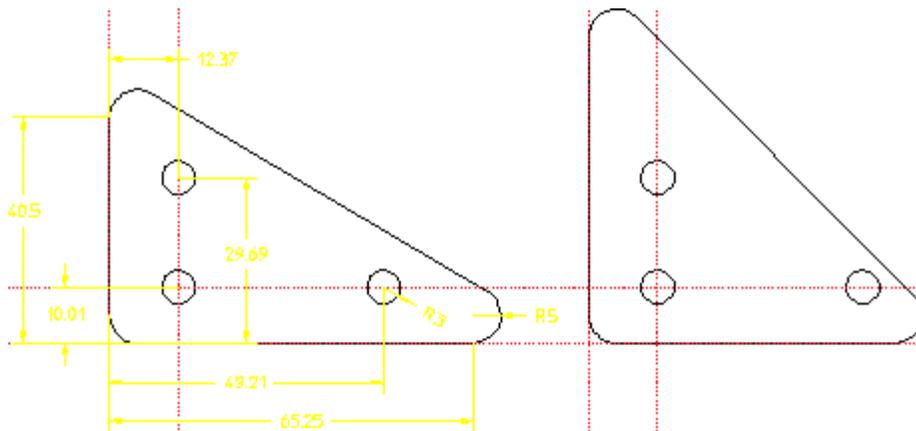
Figure 232. Preview Mode



The  **Keep** button in the **Solve** group on the **Parametric** tab, allows you to keep both the original and the variation [Figure 233. Part and Kept Variation on page](#)

721. If you try  **Keep** now, make sure you delete all the variations (but not the master part) when you have finished.

Figure 233. Part and Kept Variation



Parameters

At this point, you've created a variation of your original part, but it's probably not exactly what you want. One obvious flaw is that the holes in the plate are not constrained to adjust along with the perimeter. Although you could move them by giving new values to their dimensional constraints (just like you did for the length and height of the plate), this is not efficient, because you would have to calculate and re-enter these values each time you generated a variation with different

dimensions. Parameters provide a much better method for accomplishing this. Parameters allow you to use expressions to define relationships among constraints.

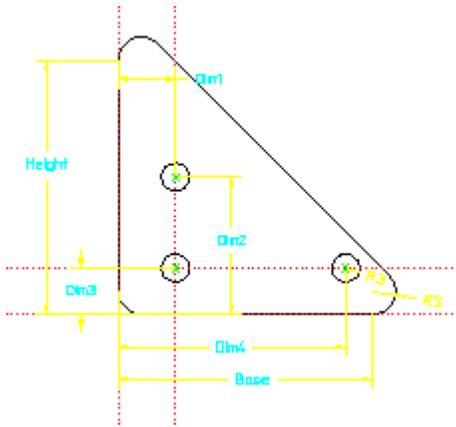
Assign Parameter Names

First, assign parameter names to the appropriate dimensional constraints.

1. Unless you have already done so, click **Parametric** and then, in the **Solve** group, click  **No Keep**.
2. Click  **OK** to replace the original part with the variation you created in the last step.
3. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
4. In the **Action** area, click **Assign**.
5. In the **Type** box, select **Dimension**.
6. Click **Apply**.
7. Click the dimension text that gives the length of the triangle (55).
8. Type 'Base' in the user input line and press ENTER.
9. Click the dimension text that gives the height of the triangle (55).
10. Type 'Height' in the user input line and press ENTER.
11. Click the "12.37" dimension on the top hole.
12. Type 'Dim1' in the user input line and press ENTER.
13. Click the "29.69" dimension on the top hole.
14. Type 'Dim2' in the user input line and press ENTER.
15. Click the "10.01" dimension on the leftmost hole.
16. Type 'Dim3' in the user input line and press ENTER.
17. Click the "49.21" dimension on the rightmost hole.
18. Type 'Dim4' in the user input line and press ENTER.
19. Click  **OK**.

The dimension text should now look like [Figure 234. Parameter Names Assigned on page 723](#).

Figure 234. Parameter Names Assigned



With the names assigned, you now define the relationships among the parameters in the parameter definition table. For this example, we will set up the parameters so that all parameterized dimensions are functions of the Base parameter.

Define Parameter Values

1. Click **Parametric** and then, in the **Show** group, click  **Current**.

The **Current Constraints** dialog box opens.

2. In the **Name** column, select the list entry for **Height**.
3. Click the parameter value entry field near **Apply**.
4. Type $(.75 * \text{Base})$ and click **Apply**.

The value for the **Height** constraint is now defined to be $.75$ of the value of **Base**.

5. Click the **Dim1** entry.
6. Type **Dim3** and click **Apply**.
7. Click the **Dim2** entry.
8. Type $(\text{Height} - \text{Dim3})$ and click **Apply**.
9. Click the **Dim3** entry.
10. Type $(.2 * \text{Height})$ and click **Apply**.
11. Click the **Dim4** entry.
12. Type $(\text{Base} - \text{Dim1})$ and click **Apply**.
13. Click **Advanced** to view the result of these changes in the parameter table.

Solve for Variations

With the parameters defined, you can now use the **Solve** group buttons on the **Parametric** tab for variations:

3. Type a new value.
 4. Click **Apply**.
- By using the parameter value table:
 1. Click **Parametric** and then, in the **Show** group, click  **Current**.
The **Current Constraints** dialog box opens.
 2. Click **Advanced**.
The **Advanced** dialog box opens. It contains the parameter value table.
 3. Click the value you want to change in the **New Value or Expression** column.
 4. Type the new value and click **Apply**.

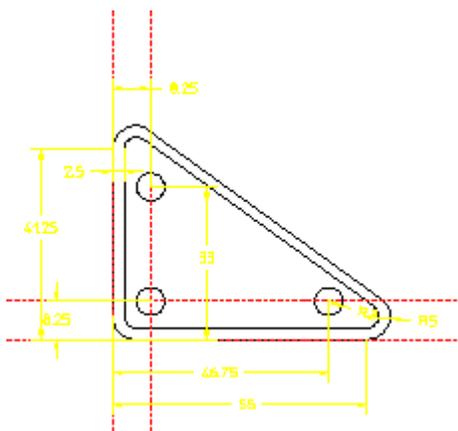
Adding New Geometry

It is often useful as you work with a constrained part to be able to add or delete geometry. Many parametric design systems do not allow this, but Parametric Design handles this situation easily. In the final section of this example, we'll add a new feature to our triangular plate to illustrate how new geometry is integrated into a constrained part.

First, let's modify the triangular plate by adding a small "lip" to the outside edge.

Use the Creo Elements/Direct Drafting  **Equidistance** button to create a contour that is 2.5 mm inside the edge of the plate. Dimension the width of this new lip as shown in [Figure 236. New Geometry on Demopart01 on page 725](#).

Figure 236. New Geometry on Demopart01



To incorporate this new feature into the Parametric Design constraint system, you only need to click **Complete** in the **Current Constraints** dialog box again. Clicking **Complete** generates the additional constraints needed for the new geometry. Subsequent use of the **Solve** group buttons recognizes the new feature:

1. After adding new geometry, click **Parametric** and then, in the **Create** group, click  **Constraints**.

The **Generate Constraints** dialog box opens.

2. Click **Complete**.
3. In the **Generate Constraints** dialog box, click **Show** in the **Action** area.
4. In the **Act On** box, select **New**.
5. In the **Type** box, select **All Types**.
6. Click **Apply** to see how the new geometry was constrained.
7. In some cases, the addition of new geometry adds information to the part that causes existing constraints to become unnecessary. **Complete** will indicate if it finds unused constraints. If applicable, display these constraints with **Show, Unused, All Types** and **Free** them if you wish.
8. Change the value of the base dimension to 48.
9. Use  **No Keep** in the **Solve** group on the **Parametric** tab to generate the variation.

Note

The new lip adjusts along with the rest of the part.

To Consider

Here are a couple of additional tasks that you can try with this drawing. Both will give you some practice with manual constraint assignment.

- Through the center of each hole, add a vertical and horizontal line that extends a short distance beyond the circle (like a cross hair). Use manual constraints to force each line to move with the hole and to continue to extend the correct distance beyond the circle when the hole changes size. Hint: Apply **Point on** and point-to-circle **Distance** constraints.
- Go back to the original geometry, but first remove the both fillets on the left vertical line of the model. Go through exercise again, including the creation of the equidistant inner contour. This time, **Complete** will not cause the inner

contour to remain equidistant. Why? Can you add manual constraints to fix the problem? Hint: Use **Samedist** constraints.

Example 2: Symmetry Lines

In this example, you're given a drawing of a plate with a large slotted hole and several smaller boltholes. Your goal is to generate variations of this plate in which the slotted hole has different sizes and configurations. Each variation should be generated by simply changing the dimensions for the radius of the hole and the width and depth of the slots.

To accomplish this task, you'll assign several **Symmline** constraints, which help to maintain the configuration of hole as it changes size. You'll also use parameters to control how the variation is generated.

As you work through this example, pay particular attention to the way in which symmetry lines are used by Parametric Design, and how mirror constraints are generated. You should come away from this example with a clear idea of how mirroring works.

Load the Drawing

1. Click **File** ►  **Open** to open the Creo Elements/Direct Drafting File Browser.
2. Double-click the `pd_demos` directory.
3. Select the `demopart02.mi` file and click **Open**.
4. Click **Parametric** and then, in the **Modify** group, click  **Zone**.
5. Click  **Add Multiple**.
6. Type **ALL** in the user input line and press ENTER.
7. Click  **OK**.

-
4. Click **Parametric** and then, in the **Create** group, click  **Constraints**.

The **Generate Constraints** dialog box opens.

5. In the **Action** area, click **Clear** and in the **Type** box, select **All Types** to clear the small icons.
6. In the **Action** area, click **Show** and in the **Type** box, select **All Types** to display the icons in the new size.

The icons are now visible, but may still be overlapping. If all four icons are not visible, move them to uncover the obscured icons:

1. Click **Parametric** and then, in the **Modify** group, click  **Move Icon**.
2. Click the constraint icon you want to move.

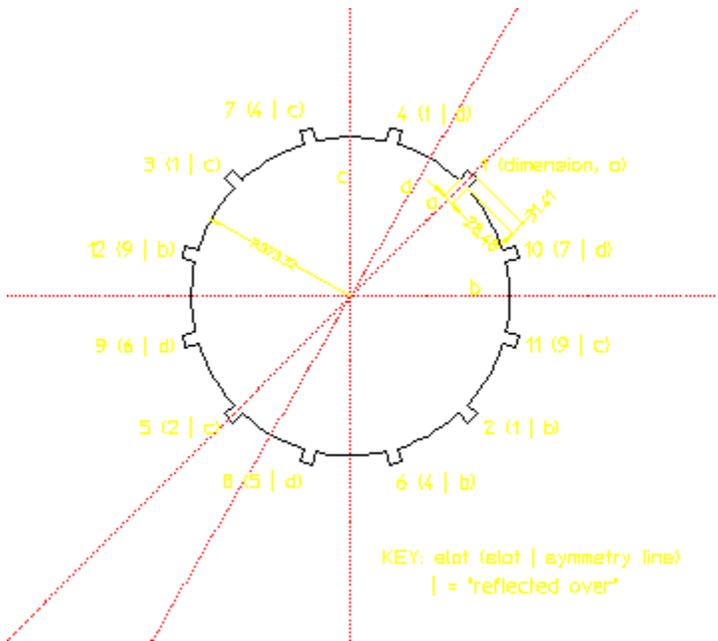
The icon is replaced with a "rubberband" line.

3. Drag the rubberband line to where you want the icon and click at that location.
4. Move the rest of the icons till you can see all the four icons.

Assign Symmetry Lines

The most notable feature of this part is that although only one of the slots around the edge of the hole has known dimensions, the positions of the remaining slots can be determined by mirroring this slot over one or more lines of symmetry. The construction lines in the part serve as these symmetry lines. [Figure 238. Symmetry Lines in Demopart02 on page 730](#) shows the general method by which the width of all the slots in this part can be determined by reflecting a known width over lines of symmetry. **Solve** may not use this exact scheme, but the process it uses is similar.

Figure 238. Symmetry Lines in Demopart02



By assigning **Symmline** constraints to the construction geometry, you ask the software to look for mirror relationships across these lines.

To assign the **Symmline** constraints:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Symmline**.
4. Click **Apply**.
5. Click the four construction lines that run through the center of the part.
6. Click  **OK**.

Automatically Generate the Remaining Constraints

Having manually assigned the initial constraints, use **Complete** to generate the remaining constraints needed to fully constrain the part.

1. In the **Generate Constraints** dialog box, click **Complete** in the **Action** area.
2. Click **Apply**.
3. Click  **OK**.

Examine Constraints

Following a **Complete** command, you should always check the part to make sure that a reasonable set of constraints has been assigned. Check the following:

1. Are all important dimensions constrained?

- a. Click **Parametric** and then, in the **Create** group, click  **Constraints**.

The **Generate Constraints** dialog box opens.

- b. In the **Action** area, click **Show**.
- c. In the **Act On** box, select **Used**.
- d. In the **Type** box, select **Dimensions**.
- e. Click **Apply**.

Notice that the dimension for the top of the plate has not been constrained. This is not needed in this case because other constraints fix this line.

2. Are there any **Distance** or **Size** constraints? **Complete** usually only assigns these constraints as a "last resort" if it can find no other way to constrain an element. A distance-constrained element may move along with other elements when it not intended to.

In the **Generate Constraints** dialog box, click **Show** in the **Action** area and select **Size** and **Distance** in the **Type** box to display any instances of these constraints.

An alternate way to check the constraints on your part is using  **Inq Elem** and  **Inq Icon** in the **Show** group on the **Parametric** tab. For this example, you can use  **Inq Elem** to see how **Complete** used the symmetry lines you defined to constrain components of the center circle:

1. Click **Parametric** and then, in the **Show** group, click  **Inq Elem**.
2. Click each line in the dimensioned slot of the part.

Notice how mirror constraints were extracted for each element in the slot.

Try clicking on other components around the center circle.

3. Click **Parametric** and then, in the **Create** group, click  **Constraints**.

The **Generate Constraints** dialog box opens.

4. In the **Action** area, click **Clear**.
5. In the **Type** box, select **All Types**.

-
- Click  **OK** when you have finished with  **Inq Elem.**

Generate a Variation

The part is now fully constrained. To generate the first variation:

- Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
- In the **Action** area, click **Assign**.
- In the **Type** box, select **Dimension**.
- Click **Apply**.
- Click the dimension for the radius of the large hole (currently 373.32).
- Type 250 in the user input line and press ENTER.
- Click the dimension for the width of the slot (currently 28.48).
- Type 20 . 75 in the user input line and press ENTER.
- Click the dimension for length of the slot (currently 31.41).
- Type 58 in the user input line and press ENTER.

- Click  **OK**.

- Click **Parametric** and then, in the **Solve** group, click  **Preview**.

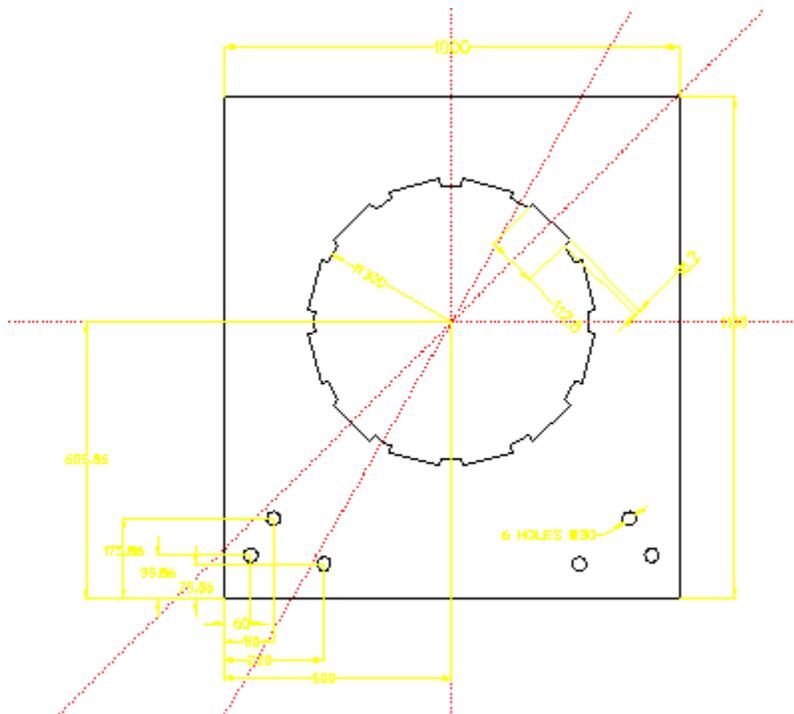
The magenta outline shows how the variation will be drawn without changing the part.

Alternatively, you can replace the master part with the new geometry by clicking  **No Keep** in the **Solve** group on the **Parametric** tab.

- Click  **OK** to remove the preview, or  **Undo** if you solved with  **No Keep**.

Repeat this procedure several times, substituting new values of your choice.

Figure 239. Demopart02 Variation



Parameterize

Suppose you are asked to generate several variations of this part in which the slots change size proportionally to the radius of the large hole. You could accomplish this using the methods shown above, but you would have to recalculate the dimensions of the slots each time you re-sized the hole. Your task is much easier if you parameterize the three dimensions in question. For example:

Assign Parameter Names

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Dimension**.
4. Click **Apply**.
5. Click the dimension for the radius of the large hole.
6. Type 'HoleRad' in the user input line and press ENTER.
7. Click the dimension for length of the slot.
8. Type 'SlotLen' in the user input line and press ENTER.

-
- Click the dimension for the width of the slot.
 - Type 'SlotWid' in the user input line and press ENTER.
 - Click  **OK**.

Define Parameter Values

- Click **Parametric** and then, in the **Show** group, click  **Current**.

The **Current Constraints** dialog box opens.

There should be three table entries; one for each parameter name you assigned.

- Click the list entry for HoleRad.
- Click the parameter value entry text box.
- Type 375 and click **Apply**.
- Click the SlotLen entry.
- Click the parameter value entry text box.
- Type $(0.084 * \text{HoleRad})$ and click **Apply**.
- Click the SlotWid entry.
- Click the parameter value entry text box.
- Type $(0.076 * \text{HoleRad})$ and click **Apply**.

Notice the "New Value Type" field for each of the three parameter definitions. While HoleRad is defined directly by a value (375), SlotWid and SlotLen are defined by evaluating expressions.

The parameters now define a variation very similar to the original part. If you wish, you can use the buttons in the **Solve** group on the **Parametric** tab now (use



No Keep) to restore this configuration.

Generate Variations

Because HoleRad is defined as a "value" parameter, and both SlotWid and SlotLen can be resolved from it, you need only change the value of HoleRad to generate variations. Use the parameter value table to do this:

- Click **Parametric** and then, in the **Show** group, click  **Current**.

The **Current Constraints** dialog box opens.

Only parameters defined as "Value" parameters appear in the parameter value table.

- Click the HoleRad entry.

3. Click the parameter value entry text box.
4. Type 425 and click **Apply**.
5. Click **Parametric** and then, in the **Solve** group, click  **No Keep** to create the variation.
6. Click  **OK** to accept the variation.

As you can see, the dimensions of the slots now change with the hole radius. Input several different values for HoleRad now.

Other Things to Try

- You may wish to try some other parameter definitions for this part. How would you define parameters so that the width of the slots controls the radius of the hole?
- Another interesting feature of this example is the way in which the small boltholes are constrained. If you assign a new diameter to one hole, the rest will resize when you solve. Why? Examine the constraints on these holes to find out.
- Start again at the beginning of this example. After using **Complete**, try modifying the position of the uppermost pair of boltholes (the pair whose y-position is specified by the dimension 175.86). Modify this to be 240. The x-position dimension (110) moves to be above the dimension 60. Use **Distance** constraints on the dimension texts to keep them in the same relative positions.
- Normally either the top horizontal dimension of 1000 or the bottom horizontal dimension of 500 will be extracted by the automatic constraint generator. What if you wanted to be able to control both dimensions independently? This would imply that the two vertical sides of the part need not remain vertical. How would you constrain the part to allow this to happen?

This is a very simple example of a very common situation. The default part contains obvious constraints that are unintended in general. Even in this simple example, the number of unintended constraints that the automatic constraint generator could extract is large (side lines are parallel, each side line is perpendicular to the top and bottom lines, each side line is parallel and perpendicular to some of the lines in the internal cutout and to some of the construction lines, etc).

There are three general strategies for this situation. Try each one.

- Use **Complete** and then **Ban** those unintended constraints. Continue this process until the goal has been reached.

-
- Manually constrain the side elements so that **Complete** doesn't need to add any constraints.
 - Use the  **Stretch** buttons in the **General** group on the **Modify** tab to change the default part to one that is more general.

Which is easiest?

Example 3: Two Views

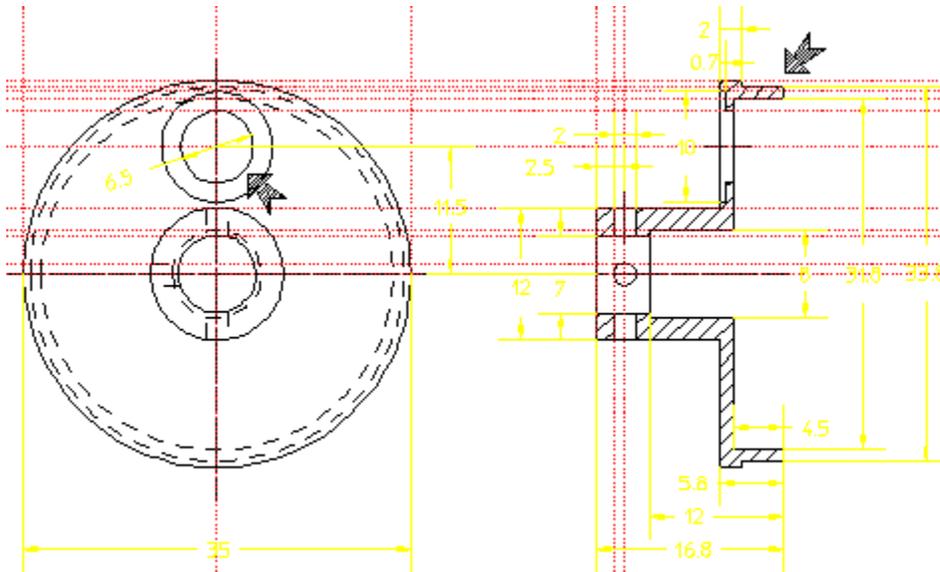
In this example, you're given a drawing that contains a front and side view of the same part. You need to change the size of several features in the part, but don't want to have to redraw both views to reflect the changes. The goal, then, is to use Parametric Design to carry the changes you make to one view over to the second view.

The key to accomplishing this task is to use construction lines to transfer constraints placed on one section of a part to another section.

Load the Drawing

1. Click **File** ►  **Open** to open the Creo Elements/Direct Drafting File Browser.
2. Double-click the `pd_demos` directory.
3. Select the `demopart03.mi` file and click **Open**.
4. Click **Parametric** and then, in the **Modify** group, click  **Zone**.
5. Click  **Add Multiple**.
6. Type **ALL** in the user input line and press ENTER.
7. Click  **OK**.

Figure 240. Demopart03



For this example, you will resize the hole and the ring indicated by the arrows in [Figure 240. Demopart03 on page 737](#). Modifications to the large hole in the front face should carry over to the side view. Changing the width of the ring in the side view should change the radii of the hidden-line circles in the front view.

The main mechanism used to carry dimension information from one section of the part to the other is the construction line. When building this part, we included many construction lines whose only purpose was to link elements for Parametric Design. Look, for example, at the small hole in the side view. In most situations, it would be considered excessive to place horizontal and vertical construction lines on the perimeter as well as the center of this hole. For Parametric Design, however, these lines are critical: The vertical lines "link" the diameter of this hole with the two vertical holes in the side view, while the horizontal construction lines give us a link to the front view.

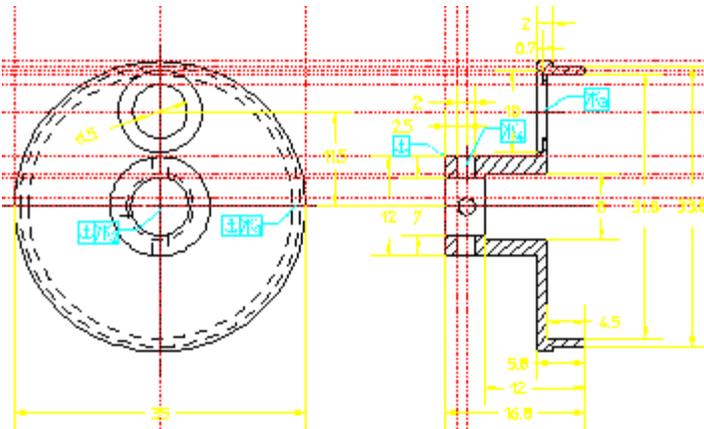
Assign Reference Elements

With a two-view part like this, it is important to anchor both views so that they don't move to overlap each other during the solve. For this part, assign **Ref Elem** constraints to the two centerlines on the front face and to the vertical construction line on the side view that's closest to the front view, [Figure 241. Manually Assigned Constraints on page 738](#):

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Refelem**.

4. Click **Apply**.
5. Click two centerlines and one construction line as shown in the figure below.
6. Click  **OK**.

Figure 241. Manually Assigned Constraints



Assign Symmetry Lines

In addition to the reference elements, the configuration of this part suggests that it would be useful to assign symmetry line constraints to the centerlines of the elements that we're going to be resizing.

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Symmline**.
4. Click **Apply**.
5. Click the four centerlines that run through the holes in the two views. [Figure 241. Manually Assigned Constraints on page 738](#) shows the placement of these constraints.
6. Click  **OK**.

Automatically Generate the Remaining Constraints

Having manually assigned the important constraints for this example, use **Complete** to generate the remaining constraints needed to fully constrain the part.

Examine Constraints

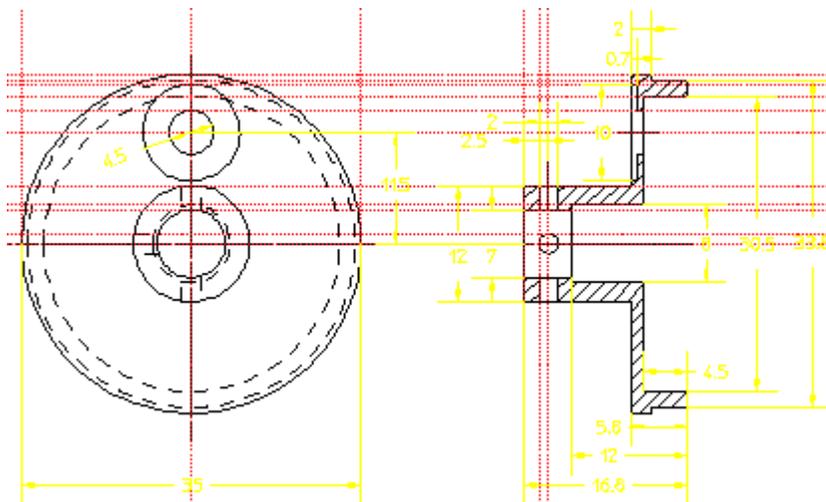
When **Complete** finishes, examine the part carefully with both **Show** and  **Inq Elem** to see how **Complete** used the construction geometry to link up the elements we want to resize.  **Inq Elem** is especially useful for showing how individual construction lines are associated with other geometric elements.

Generate the Variation

When you feel you understand the connections between the two views of the part, modify the constraints on the hole and the ring and use the **Solve** group buttons to see how Parametric Design makes use of these connections.

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Dimension**.
4. Click **Apply**.
5. Click the dimension for the diameter of the large hole in the front view (This dimension is currently 6.5).
6. Type 4 . 5 in the user input line and press ENTER.
7. Click the dimension for the inner diameter of the ring in the side view. (This dimension is currently 31.8.)
8. Type 30 . 5 in the user input line and press ENTER.
9. Click  **OK**.
10. Click **Parametric** and then, in the **Solve** group, click  **No Keep**.

Figure 242. Demopart03 Variation



To Consider

- What other dimensions can be carried over from view to view in this part with the existing construction lines? Try resizing some other dimensions and see how the solver handles them.
- Could you delete some of the construction geometry in this part and still accomplish the example? How?
- Suppose that you want to increase the distance between the two views of this part. This is trivial to do with Parametric Design, but there is one hitch: How do you get the centerline that runs through both views to "stretch" as the views move apart? Hint: Try adding a point element to each end of the center line and then, for each point, fix the distance between it and an element in the nearest view.

Example 4: Macros

Your task is to design a series of fan housings. The housings are identical except that the air intakes and exhaust ports of each housing must vary to accommodate fans of different power. It is important that the area of the air intakes equal the area of the exhaust ports to maintain proper air flow through the housing. Unfortunately, the intakes are wedge-shaped while the exhaust ports are slots. This makes the area calculations more complicated.

To accomplish this task, you'll use a Creo Elements/Direct Drafting macro that:

1. Calls Parametric Design to design the air intakes.
2. Calculates the area of the intakes.
3. Transforms the area value into the exhaust port size.

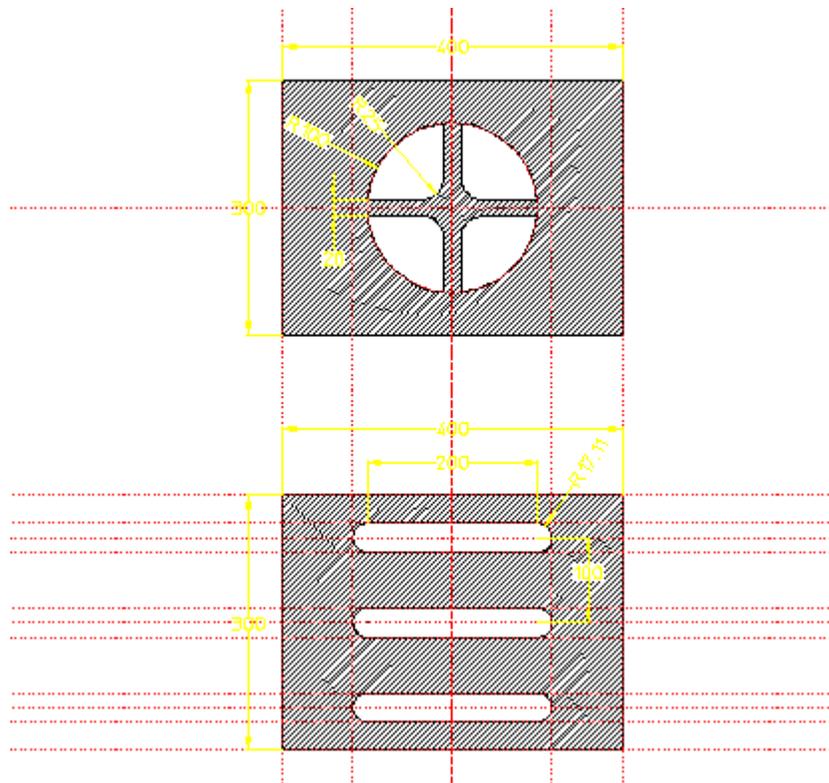
4. Calls Parametric Design to design the exhaust ports.

Most of the setup for this example has been done for you. The part, demopart04.mi, has already been constrained and parameterized, and the macros you need are already defined in the file demopart04.mac. As you work through this example, focus on the contents of the macro and how they relate to what you see on your screen.

Load the Drawing

1. Click **File** ►  **Open** to open the Creo Elements/Direct Drafting File Browser.
2. Double-click the pd_demos directory.
3. Select the demopart04.mi file and click **Open**.

Figure 243. Demopart04



Examine the Drawing

Take a few minutes now to look at the features of this drawing.

- Notice the part structure. The front face is the top part, while the back face is a sub-part named "slot-grid." Parametric Design only works on the current part, so our macro must be able to switch from part to part.
- Both parts of the drawing are already in the zone.
- Examine the initial constraints on the top part, [Figure 244. Intake Constraints on page 742](#):

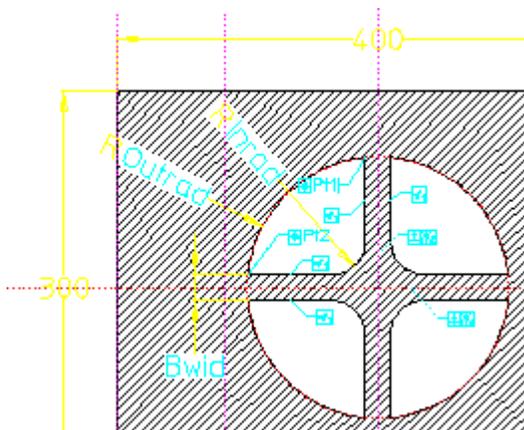
1. Make the top part current.

To make the top part current:

- a. Click **Structure** and then, in the **Parts** group, click  **Edit**.
 - b. Click  **Top**.
2. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
 3. In the **Action** area, click **Show**.
 4. In the **Act On** box, select **User**.
 5. In the **Type** box, select **All Types**.
 6. Click **Apply**.

The **Dimension**, **Distance**, and **Symmline** constraints ensure that all four intake ports retain their configuration in variations. The **Refpoint** constraints Pt1 and Pt2 were created only to provide information to our macro. This is explained later.

Figure 244. Intake Constraints



- Examine the constraints on the "slot_grid" part now. (Hint: make "slot_grid" the current part in order to examine it). Notice that the slot radius has been

parameterized as "Newrad." How did we ensure that all three exhaust ports will remain the same size?

When you have finished looking at the constraints, make sure that top part is current and that your main window shows both parts.

Examine the Parameters.

Display the parameter definition table to examine the parameters on the part:

1. Click **Parametric** and then, in the **Show** group, click  **Current**.

The **Current Constraints** dialog box opens.

2. Click **Advanced**.

Assuming that the top part is active, there should be five parameters defined:

Bwid	Gives the width of the bars between the air intakes. Set to 20.
Inrad	Gives the inner radius of each air intake. Set to 25.
Outrad	Gives the outer radius of each air intake. Set to 100.
Pt1	A reference point. Pt1 is a geometry parameter, so its value will always correspond to the top corner of an air intake, no matter how this part is resized.
Pt2	A reference point. The value of Pt2 will always correspond to the bottom corner of Pt1's air intake.

To prove that the parameters work as we say they do, try them:

1. Change the value for "Out_rad" to 120. Click the "Outrad" entry in the table and type 120.

2. Click **Parametric** and then, in the **Solve** group, click  **Preview**.

Notice how the hole configuration changes.

3. Restore the parameters to their original values when you've finished experimenting.

Take a look at the parameters on the exhaust ports now.

4. Make "slot_grid" the current part.

5. Preview a few variations of this part by inserting new values for the

"Newrad" parameter and solve using  **Preview** in the **Solve** group on the **Parametric** tab.

6. Set "Newrad" to 5 and solve by using  **No Keep** in the **Solve** group on the **Parametric** tab.

7. Make the top part current.

The configuration of the parameters should give you an idea of the strategy we'll use to accomplish our task. We can easily vary the size and configuration of the four air intakes in the top part by changing the parameter values for "Bwid," "Inrad," and "Outrad." On the "slot_grid" part, we can vary the width of all three exhaust ports by entering a new value for the "Newrad" parameter. All that needs to be done now is to translate the area defined by "Bwid", "Inrad", and "Outrad" to a radius that can be fed to "Newrad". This is exactly what our macro, Do_it, does.

Try the Do_it Macro

Before looking closely at Do_it, let's try it. Do_it is defined in the file demopart04.mac in the pd_demos\ directory. To load Do_it in your system, type input ``<Creo Elements/Direct Drafting installation directory>\pd_demos\demopart04.mac`' in the user input line and press ENTER. `<Creo Elements/Direct Drafting installation directory>` is the directory where you installed Creo Elements/Direct Drafting.

Now, type Do_it in the user input line and press ENTER.

What happened? If all went well, the exhaust ports returned to their original size. Now try this:

1. Display the parameter value table using the **Current Constraints** dialog box.
2. Change "Bwid" to 10, "Inrad" to 35, and "Outrad" to 75.
3. Type Do_it in the user input line and press ENTER.

It looks like your task is now very easy! All you need to do in order to generate your fan housings is to enter the parameters for the intake ports in the parameter table and type Do_it to compute the necessary width for the exhaust slots. Let's look at Do_it now to see how it's done.

Examining Do_it

The definition for Do_it is listed below:

```
DEFINE Do_it                                     {Generate holes in demopart04.mi}

    EDIT_PART TOP                               {Setup: make sure we are at top
part}

    SPOTLIGHT OFF

    PD_RESOLVE REPLACE                          {Solve for front holes}

    AREA_PROPERTY 1 Find_midp DEL_OLD '\temp\area'
```

```

END                                {Get the area properties of 1
front hole}

                                {and write to a temporary file.
}

EDIT_PART 'slot_grid'             {Goto the second part, slot_grid}

PD_PARAM_FIX 'Newrad' New_radius  {Find radius needed to give
equivalent}

                                {area. Fix part radius to this
value.}

PD_RESOLVE REPLACE                 {Solve for back slots}

EDIT_PART TOP                       {Clean up: go back to top part}

DISPLAY ('Variations Generated. Done.')
```

END_DEFINE {of Do_it}

Do_it calls the macros Find_midp and New_radius, both of which are listed here:

```

DEFINE Find_midp {find a point inside a front-face hole}

LOCAL A                             {Declare local variable A}

LOCAL B                             {Declare local variable B}

PD_PARAM_INQ 'Pt1'                  {Find Pt1 location from parameter
table}

LET A (PNT_XY (INQ 3) (INQ 4))      {assign Pt1 location to A}

PD_PARAM_INQ 'Pt2'                  {Find Pt2 location from parameter
table}

LET B (PNT_XY (INQ 3) (INQ 4))      {assign Pt2 location to B}

((A+B)/2)                           {Return midpoint of A and B}

END_DEFINE                          {of Find_midp}
```

```

DEFINE New_radius                    {Get area of front hole,}

                                {return radius of back slot.}

LOCAL L                             {Declare local variable L}
```

```

LOCAL A                                {Declare local variable A}

LET A 'junk'                            {Set A to some junk value!}

LET L 200                                {Set L to length of a back slot}

OPEN_INFILE 1 '\temp\area' {open file of area properties written
above}

WHILE ((SUBSTR A 1 3)<>'A =') {scan area_spec for the area (A=)
field}

READ_FILE 1 A                            {and load the area field into A}

LET A (TRIM A)

END_WHILE

LET A (SUBSTR A 4 (LEN A))               {set A to be just area value
(area of 1 hole)}

LET A ((VAL A)*4)                        {total area of front holes is 4A}

(((SQRT(L*L + (PI*A)/3))-L)/PI) {Given area of holes A and length
of slots L}

                                        {return hole radius for
equivalent area.    }

END_DEFINE                                {of New_radius}

```

The algorithm used by `Do_it` is as follows:

1. Solve the variation for the intake ports.
2. Use `AREA_PROPERTIES` to find the area properties of an intake port and write this data to a file. Call the macro `Find_midp` to generate the point which identifies this port.
3. Change the current part to the exhaust ports.
4. Call the macro `New_radius` to calculate the needed radius from the data in the file written above. Assign this radius to "Newrad."
5. Solve the variation for the exhaust slots.

This procedure is straightforward, but does use some interesting features of both `Creo Elements/Direct Drafting` and `Parametric Design` commands:

- Although `Parametric Design` only operates on the current part, a macro like `Do_it` can easily manage interchanges among multiple parts.

- The macro `Find_midp` uses the reference-point constraints `Pt1` and `Pt2` to determine a point that is guaranteed to be inside an intake port. Since `Pt1` and `Pt2` are constrained to always be on the corners of this port, the point between them will always be inside the port. Passing this point to `AREA_PROPERTIES` makes sure that we always get the properties for an intake port. Note how `PD_PARAM_INQ` is used to grab the locations of these points from the parameter table.
- `New-radius` simply reads the area data from a file and uses it along with hard-coded data about the slot length to generate the appropriate radius. The output is input to Parametric Design via `PD_FIX_PARAM`. Of course a much more complex macro could be used to supply this input!

To Consider

- Currently, `Do_it` counts on the slot length being 200. How would you rewrite `Do_it` to inquire the slot length directly from the part? (Hint: You probably need to add a new parameter to the part.)
- `Do_it` is not foolproof. You can cause problems by setting "`B_wid`" to 0. How would you fix this flaw? How would you implement general error or range checking in `Do_it`?
- Could you implement `Do_it` so that slot size drives intake size? So that either dimension could drive the other?
- `Do_it` is linked very tightly to this particular drawing. How would you generalize `Do_it`?

Example 5: Rigid Bodies

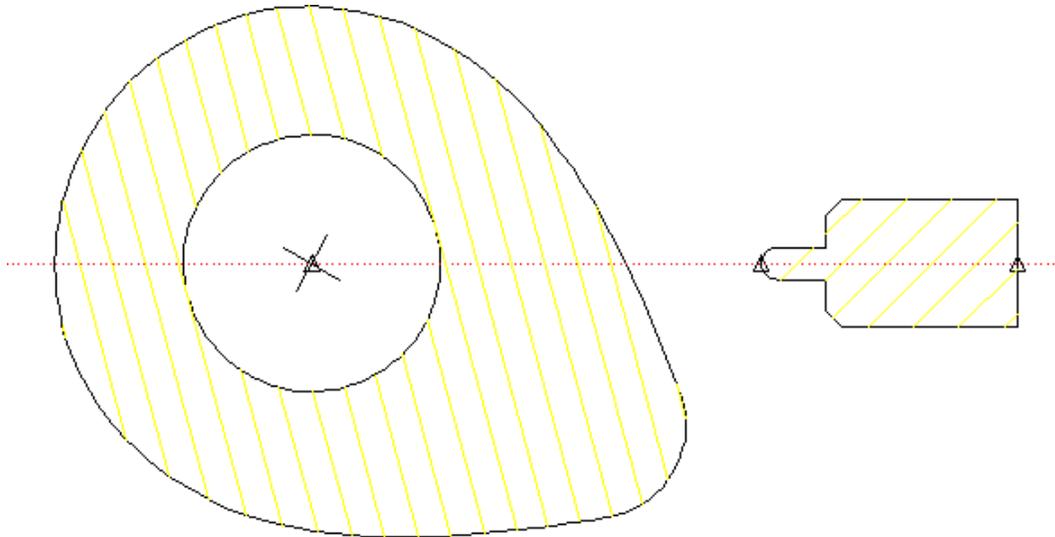
This example illustrates how rigid bodies can be used to reduce the number of constraints in a drawing while allowing a complex range of motion. You're given a drawing of a cam and follower. By collecting each into a rigid body, you can set up a simple motion study using just six constraints.

Load the Drawing

1. Click **File** ►  **Open** to open the Creo Elements/Direct Drafting File Browser.
2. Double-click the `pd_demos` directory.
3. Select the `demopart05.mi` file and click **Open**.
4. Click **Parametric** and then, in the **Modify** group, click  **Zone**.

5. Click  **Add Multiple**.
6. Type ALL in the user input line and press ENTER.
7. Click  **OK**.

Figure 245. Demopart05



The goal of this example is to force the follower to stay in contact with the cam profile as the cam rotates. By setting up both the cam and the follower as rigid bodies, we dramatically reduce the number of constraints needed to implement this action, and can concentrate more on interaction between the two pieces.

The `demopart05.mi` model is fairly simple. The only elements in the drawing that are needed specifically for parametrics are the construction line and the three point elements, which are used to facilitate placement of several **Point on** constraints.

Create Rigid Bodies

The cam and the follower each need to be collected into a rigid body. The construction line is the only part of the model that is not in a rigid body. Each rigid body must be named before elements can be collected into it:

1. Click **Parametric** and then, in the **Create** group, click  **Rigid**.
The **Generate Rigids** dialog box opens.
2. Type `cam` in the text box and click **Assign**.
3. Gather all of the cam geometry in the `cam` rigid body by enclosing it in a box.
4. Click  **OK**.

-
5. Repeat the procedure to create a rigid body called follower that contains all of the follower geometry.
 6. In the **Generate Rigids** dialog box, click **Show** and select each of the table entries you just created to confirm the contents of each rigid body. The construction line should not be included in either.
 7. Click **Close** to close the **Generate Rigids** dialog box when you finish.

Assign Constraints

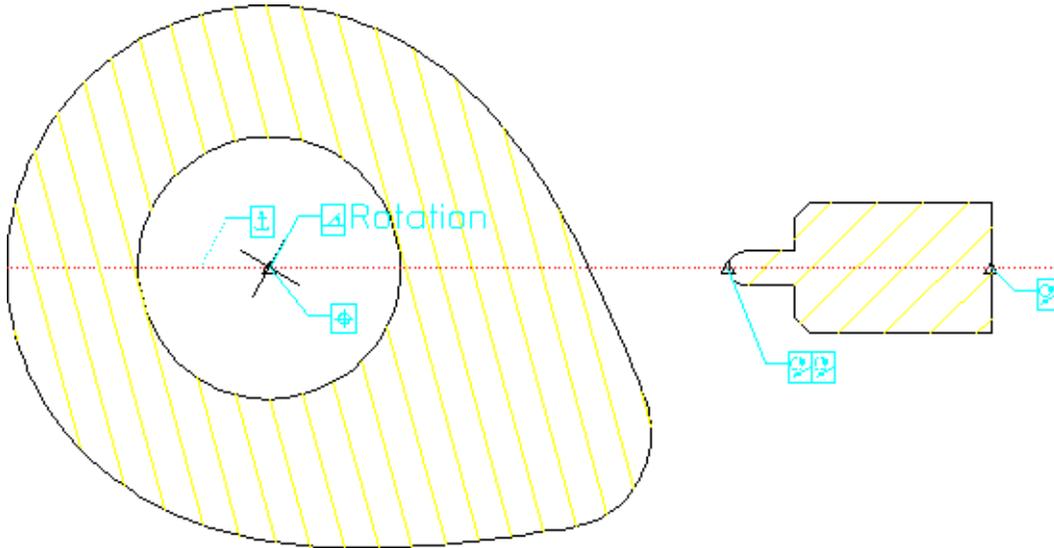
Only six constraints are needed to completely constrain this drawing. Assign each one manually:

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Refelem**.
4. Click **Apply**.
5. Click the construction line to fix it in place.
6. Use the same general method to assign a **Refpoint** constraint to the point element at the centerpoint of the cam. This fixes the cam in place on the construction line while allowing it to rotate.
7. Use the same general method to assign a **Slope** constraint to one of the small line elements that cross at the centerpoint of the cam. Name this constraint *Rotation*, but don't assign it a specific value yet.

Note that this constraint and the previous one are all that's needed to completely constrain the "cam" rigid body. The reference point fixes translation while the slope constraint fixes rotation.
8. Constrain the follower: **Assign a Point on** constraint between the left-most point element on the follower and the construction line. Fix the rightmost point element to the construction line with a second **Point on** constraint. These two constraints fix the rotational degree of freedom for the follower, but its motion along the axis of the construction line still needs to be fixed.
9. **Assign** a final **Point on** constraint between the left-most point element on the follower and B-spline profile of the cam. This fixes the translation of the cam.
10. Click **Parametric** and then, in the **Show** group, click  **Inq Icon** to confirm that you've assigned each of the constraints as described above. The constraints should resemble [Figure 246. Demopart05 Constraints on page 750](#).

As you examine the constraints, make sure you understand how each one constrains the motion of the rigid body to which it's applied.

Figure 246. Demopart05 Constraints



Solve

Before running the solver on this drawing, it is important to turn off the point-merging functionality of the **Solve** command. Remember that by default the solver will merge any two points that fall within the `PD_RESOLVE_MERGE_TOLERANCE` distance of one another. Normally this is desirable, as it helps to keep the drawing clean. In this case however, it is possible for the endpoint of the follower profile to come into contact with the endpoint of the cam profile as the cam rotates. Were this to occur, these two points would be merged, and further parametric manipulation of this drawing would be impossible. To keep this from happening, type `pd_resolve_merge_tolerance -1` in the user input line and press ENTER.

Now, click **Parametric** and then, in the **Solve** group, click  **No Keep** to generate the initial variation.

To confirm that the follower will indeed maintain contact with the cam as it rotates, open the **Current Constraints** dialog box and enter several different values for the `Rotation` parameter, solving after each one.

To Consider

- Rotating the cam by "hand" (entering a new slope and re-solving) is fairly tedious. It should be simple to write a macro to do this.

- Try this example with `pd_resolve_merge_tolerance` set to the default value (1E-6). Rotate the cam until the follower comes to rest at the very top of the cam profile. What happens with the very next rotation?
- Try to replicate this example without using rigid bodies. How many constraints are needed? How would you force the hatching on the cam to rotate along with the elements?

Tutorial

The exercises presented in this tutorial familiarize you with:

- Dimension-Driven Modification
- Design-Intent Capture

The drawings and macros that are needed for this tutorial can be found in the `$MEDIR\pd_demos` subdirectory (or in the subdirectory where you installed Creo Elements/Direct Drafting).

This tutorial is intended for users who are already familiar with the Creo Elements/Direct Drafting user interface for doing such things as creating geometry, windowing, and dimensioning. Instructions for the mouse input device are given first.

Exercise 1

In this exercise, you draw a plate and then modify its shape by changing the driving dimensions.

Ensure that **Copilot** and **Parametric Design** are enabled in the **System** dialog box.

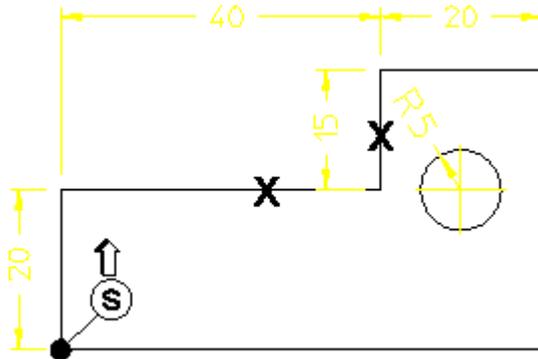
- Click **File** ► **Settings** ►  **System** to open the **System** dialog box.

Draw the basic design

1. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Line**.
2. Click  **Polygon**.
3. Draw the shape shown in [Figure 247 on page 752](#), starting at point S in the direction shown.
4. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Circle**.
5. Click  **Center & Radius**.
6. Draw the hole (the offset is from points X at values 10 0).

7. Click  **OK**.
8. Dimension the drawing as indicated in [Figure 247 on page 752](#).

Figure 247



Assign Constraints

1. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
The **Generate Constraints** dialog box opens.
2. In the **Action** area, click **Assign**.
3. In the **Type** box, select **Refpoint**.
4. Click **Apply**.
5. Click point S (see [Figure 247 on page 752](#)) to mark it as the reference point.
6. Assign the **Dimension** constraint to the dimensions (see [Figure 247 on page 752](#)).

Modify the design

To modify the design:

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Show**.
3. Enclose the design in a box.

Creo Elements/Direct Drafting displays the design's dimension in green to indicate that these are the driving dimensions. It is the driving dimensions that are used to modify the design according to the design-capture rules. These dimensions reflect the way in which you created the design.

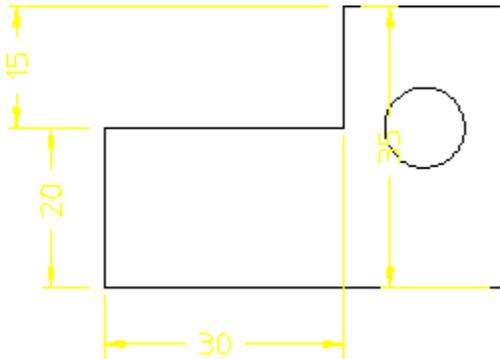
Note

Remember to use  **Fit**,  **Pan** and  **Zoom** for fitting, panning, and zooming!

To modify the design:

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Deferred**.
3. Click the dimension 40 (it changes to yellow).
4. Type 30 in the user input line and press ENTER.
5. Click  **OK**.

The design changes according to the modified dimension.



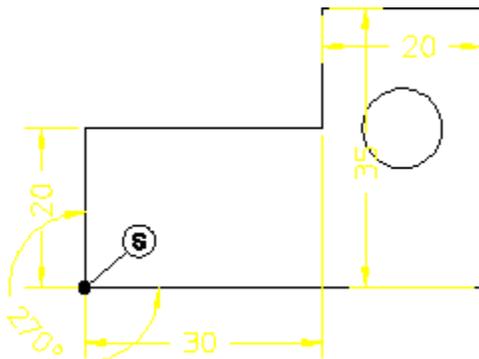
Swap the dimensions

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Immediate**.
3. Click the dimension 20 on the left of the drawing.
4. Type 25 in the user input line and press ENTER.

The shape changes. Since this is not the design you want, you need to re-dimension it.

5. Click  **Undo**.
6. Click **Viewing** and then, in the **View** group, click  **Redraw**.

7. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Angular**.
8. Click  **Angle**.
9. In the **Orient** gallery, click  **Direct**.
10. Click the intersecting lines at S.
11. Click **View** and then, in the **Viewing** group, click  **Redraw**.
12. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
13. Click  **Show**.
14. Enclose the entire design in a box.
15. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
16. Click  **Swap** and then click the yellow angle.
17. Click the dimension 15 (this dimension disappears because the angle becomes the driving dimension).
18. Click **View** and then, in the **Viewing** group, click  **Redraw**.



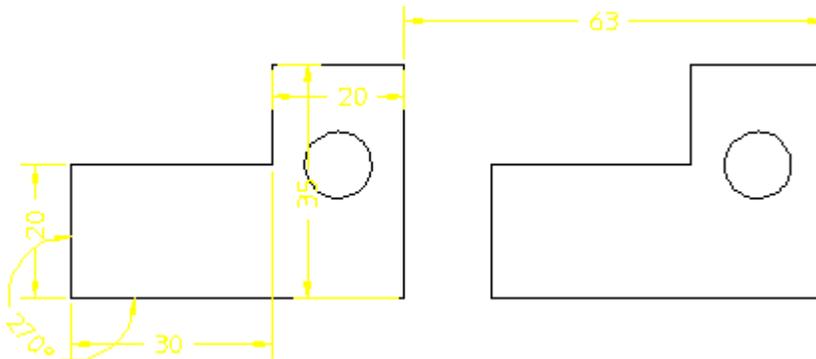
Copy the design

The next step is to repeat the above but this time copy the design to show that the copy does not contain driving dimensions. The design changes and the position of the hole remains relative to the defining offset.

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Deferred**.

3. Click the dimension 20 on the left of the drawing.
4. Type 25 in the user input line and press ENTER.
5. Click  **OK**.
6. Click **Modify** and then, in the **General** group, click the **Keep** check box.
7. Click **Modify** and then, in the **General** group, click  **Move**.
8. Enclose the design in a box and position the copied drawing.
9. Click **Parametric**, and then, in the **Modify** group, click  **Dimension**.
10. Click  **Show**.
11. Enclose both drawings in a box.
12. Click **Parametric**, and then, in the **Modify** group, click  **Dimension**.
13. Click  **Deferred**.
14. Click the dimension 25.
15. Type 20 in the user input line and press ENTER.
16. Click  **OK**.

Both the original and the copy change shape — Creo Elements/Direct Drafting remembers that you want both designs to have the same shape.



To clear the display:

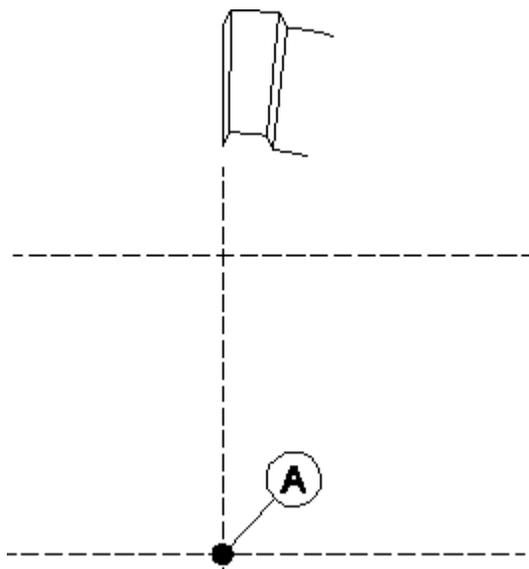
1. In the **Utilities** group, click  **Delete**.
2. In the **Utilities** group, click  **Select**.
3. Click  **All**.
4. Click  **Confirm**

Exercise 2

In this exercise, you copy and modify a cogwheel to demonstrate how to create families of parts.

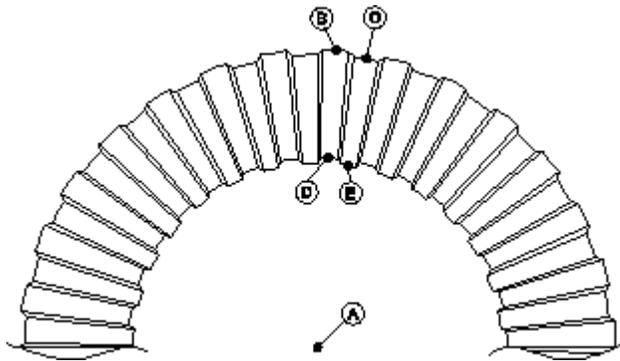
Create the basic design

1. Click **File** ►  **Open** to open the Creo Elements/Direct Drafting File Browser.
2. Double-click the `pd_demos` directory.
3. Select the `cogwheel` file and click **Open**.
Creo Elements/Direct Drafting displays a single tooth together with some construction geometry.
4. Click **Modify** and then, in the **General** group, click the **Keep** check box.
5. Click **Modify** and then, in the **General** group, click the **Repeat** check box.
6. Click **Modify** and then, in the **General** group, click  **Rotate**.
7. Click  **Center**.
8. Type 29 (the repeat factor) in the user input line and press ENTER.
9. Click point A to mark the center of rotation.
10. Type 12 to define the rotation angle in the user input line and press ENTER.
11. Enclose the single tooth in a box.
12. Click **View** and then, in the **Viewing** group, click  **Fit**.



Identify the dimensions to change

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Show**.
3. Click point B on arc of tooth.
4. Repeat for points C, D, and E.
5. Click **View** and then, in the **Viewing** group, click  **Fit**.
6. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
7. Click  **Deferred**.
8. Click the dimensions in turn and move them to unclutter the drawing.

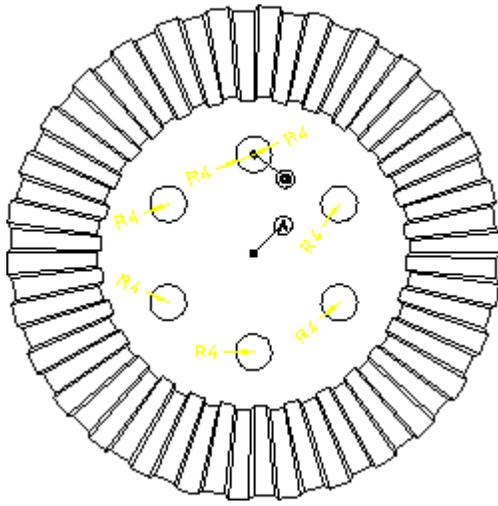


Change the tooth dimensions

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Deferred**.
 - a. Click R40.
 - b. Type 55 in the user input line and press ENTER.
 - c. Click R39.
 - d. Type 54 in the user input line and press ENTER.
 - e. Click R31.
 - f. Type 35 in the user input line and press ENTER.
 - g. Click R30.
 - h. Type 34 in the user input line and press ENTER.
3. Click  **OK**.

Creo Elements/Direct Drafting updates the cogwheel accordingly.

4. Click **View** and then, in the **Viewing** group, click  **Fit**.
5. Click **Drafting** and then, in the **Draw** group, click the arrow next to  **Circle**.
6. Click  **Center & Radius**.
7. Draw a circle with radius 4 mm at point G.
8. Click **Modify** and then, in the **General** group, click the **Keep** check box.
9. Click **Modify** and then, in the **General** group, click the **Repeat** check box.
10. Click **Modify** and then, in the **General** group, click  **Rotate**.
11. Click  **Center**.
12. Type 5 (the repeat factor) in the user input line and press ENTER.
13. Click point A to mark the center of rotation.
14. Type 60 to define the rotation angle in the user input line and press ENTER.
15. Enclose the 4 mm circle in a box.
16. Click **Drafting** and then, in the **Annotate** group, click the arrow next to  **Circular**.
17. Click  **Radius**.
18. Dimension the radii of the six circles.
19. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
20. Click  **Show**.
21. Click the original circle (not one of the copies).



Modify the design

The final step is to modify the design by changing parametric values.

1. Click **Parametric** and then, in the **Modify** group, click  **Dimension**.
2. Click  **Immediate**.
3. Click R4 (green dimension).
4. Type 3 in the user input line and press ENTER.
 Creo Elements/Direct Drafting changes the radii of all six circles — the yellow (annotation) dimensions are also updated.
5. In the **Utilities** group, click  **Delete** and delete the six yellow dimensions.
6. Click **Parametric** and then, in the **Create** group, click  **Constraints**.
 The **Generate Constraints** dialog box opens.
7. In the **Action** area, click **Show**.
8. In the **Type** box, select **Size**.
9. Click **Apply**.
 The **Parametric** module has used the actions performed during the creation of the design to generate constraints for dynamic modification of the design.
10. In the **Generate Constraints** dialog box, click **Assign** in the **Action** area.
11. In the **Type** box, select **Size**.
12. Click **Apply**.
13. Click the original circle at G.

14. Type 'Borehole_radius' in the user input line and press ENTER.

15. Click  **OK**.

16. Click **Parametric** and then, in the **Show** group, click  **Current**.

The **Current Constraints** dialog box opens.

17. Click **Advanced**.

The **Advanced** dialog box opens.

18. Click the value 3 in the **New Value or Expression** column.

19. Edit the value to 4 and click **Apply**.

20. Click **Parametric** and then, in the **Solve** group, click  **Preview**.

21. Click **Parametric** and then, in the **Solve** group, click  **Keep**.

22. Click the design.

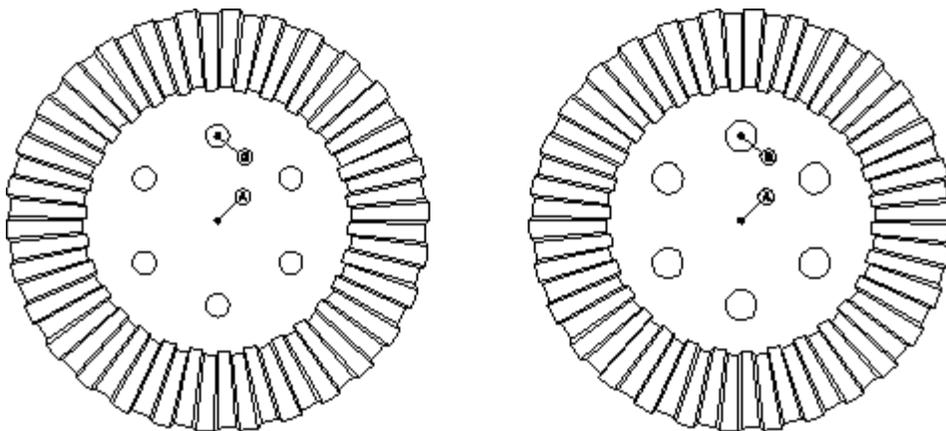
23. Move the copy of the design to another location (you may need to use  **Pan** in the **Viewing** group on the **View** tab).

24. Click the destination point for the copy.

25. Click  **OK**.

26. Click **View** and then, in the **Viewing** group, click  **Fit**.

Note the smooth transition from simple design creation using the green driving dimensions to parametric design.



Hints and Tips

You have had a look now at the latest technology available in the Mechanical Engineering CAD environment.

Here are some hints and tips when creating parametric drawings:

- To make sure your design intent drawing is always stored right away, enable **Parametric Design** and **Copilot** every time you create geometry. This avoids having to give any constraints afterwards.
- When you modify geometry using the **Solve** group buttons in the **Parametric** tab, you can decide whether you want to keep the original geometry and let the modified be a copy or if you want to modify the original geometry without a copy. This is done by clicking either  **Keep** or  **No Keep**, respectively.
- When you specify a parameter, you can either specify a figure or enter an expression that is evaluated when the design is solved. This is useful, for example, when maintaining relations between width and height of designs.
- In case you want to do specific modification to more complex designs, consider forming rigid bodies of geometry you want to move and translate as a whole. This can facilitate the modification of large designs.
- How you create the design is important. For example, try creating [Figure 46 on page 752](#) using horizontal and vertical geometry instead of using  **Polygon**. You would get very different results.

Parametric Design Defaults

User-changeable defaults exist for several entities used by Parametric Design.

All of these defaults may be set at Creo Elements/Direct Drafting startup time by including the appropriate Parametric Design functions in the file `pd_def.mac`. The default constraint icon size and color may also be set through Parametric Design screen buttons.

Setting Icon Defaults From the Screen

Use  **Move Icon** and the **Icon Setting** dialog box to set the location, size and colors of Parametric Design information labels and highlighting.

Move Icon

 **Move Icon** allows you to change the location of constraint icons on the part.

To move icons:

-
1. Click **Parametric** and then, in the **Modify** group, click  **Move Icon**.
 2. Click the icon to be moved.
 3. Drag the cursor to the new location and click again.

 **Note**

You can only change the position of the constraint icon itself; you cannot change the position of the callout line on the referenced geometry.

Icon Size

To change the icon size:

1. Click **File** ► **Settings** ►  **Parametric** ►  **Parametric Icon**.

The **Icon Setting** dialog box opens.

2. In the **Size** box, type the new size.
3. Click **OK**.

Subsequent **Show** or  **Inq Icon** functions draw constraint icons in the new size. Existing icons can be cleared and re-displayed to get the new size. The Creo Elements/Direct Drafting NEW_SCREEN function also re-displays icons in the new size and location. The default icon size is the same as the Creo Elements/Direct Drafting default text size, that is, 3.5 mm.

Set Color

To set the icon color:

1. Click **File** ► **Settings** ►  **Parametric** ►  **Parametric Icon**.

The **Icon Setting** dialog box opens.

2. In the **Color** box, select the new color.

Subsequent **Show** or  **Inq Icon** functions draw constraint icons in the new color. Existing icons must be cleared and re-displayed to get the new color. The initial show color is CYAN.

The PD_Defaults File

Parametric Design reads the file `pd_def.mac` at startup to determine defaults. You can change these defaults by editing this file. As supplied, the `pd_def.mac` file contains the following lines:

```
PD_AUTO_SYMMETRY_LINE YES           Auto symmetry lines are
enabled and

PD_AUTO_SYMMETRY_COLOR YELLOW       defined to be
yellow,

PD_AUTO_SYMMETRY_LINETYPE DOT_CENTER dot-centerlines.

used by AUTO                         Zero-distance tolerance

PD_AUTO_ZERO_DISTANCE_TOLERANCE 1E-6 Distance tolerance used by
AUTO

PD_AUTO_SAME_DISTANCE_TOLERANCE 1E-6 Same-angle tolerance used
by AUTO

PD_AUTO_ANGLE_TOLERANCE 1E-6        Tolerance for determining
tangency in AUTO

PD_AUTO_TANGENT_TOLERANCE 1E-6      Tolerance for merging

icons is CYAN                        Default SHOW color for

PD_RESOLVE_MERGE_TOLERANCE 1E-6     Default Icon size is 3.5

replace dimension text               Dimensional constraints

PD_SHOW_COLOR CYAN                  ZONE Auto On feature is
enabled

PD_SHOW_LABEL_SIZE 3.5              Default color for
generated points

PD_SHOW_USE_POSTFIX NO              Linetype for generated

points                               Visibility of generated

ADD_CURRENT_INFO "PD_ZONE" END      Default color for
generated construction lines
```

construction lines	Linetype for generated
PD_NEW_POINT_COLOR GREEN construction lines	Visibility of generated
PD_NEW_POINT_LINETYPE DOT_CENTER color is MAGENTA	Default SOLVE Preview
PD_NEW_POINT_VISIBILITY INVISIBLE parametric dimensions is GREEN	Default color of
enabled by default	Design Intent Capture
PD_NEW_C_LINE_COLOR GREEN	
PD_NEW_C_LINE_LINETYPE DOT_CENTER	
PD_NEW_C_LINE_VISIBILITY INVISIBLE	
PD_PREVIEW_COLOR MAGENTA	
PD_DEFAULT_DIM_COLOR GREEN	
Design_intent_on	

PD_AUTO_SYMMETRY_*

The PD_AUTO_SYMMETRY_* options enable and disable automatic symmetry line constraining. When PD_AUTO_SYMMETRY_LINE is set to YES, the constraint generator and solver will automatically assign symmetry-line constraints to all lines that have the PD_AUTO_SYMMETRY_COLOR and PD_AUTO_SYMMETRY_LINETYPE.

PD_AUTO_*_TOLERANCE

The PD_AUTO_*_TOLERANCE options set various tolerances used by **Complete** to determine the relationships among elements in a part. These tolerances are used to compensate for minor inconsistencies in a part (e.g., fillets that are not quite tangent, slightly disconnected elements, lines slightly out of parallel, etc.).

PD_AUTO_ZERO_DISTANCE_TOLERANCE

This function specifies the absolute tolerance used by **Complete** to compare a distance to zero. This tolerance is used to detect coincident points, collinear lines and point-on-element constraints. The number supplied must be greater than or equal to zero. The initial setting is 0.000001 mm.

PD_AUTO_SAME_DISTANCE_TOLERANCE

This function specifies the absolute tolerance used by **Complete** to compare two non-zero distances. This tolerance may be used, for example, to check whether two circular elements have the same radius. The number supplied must be greater than or equal to zero. The initial value is 0.000001 mm.

PD_AUTO_ANGLE_TOLERANCE

This function specifies the absolute tolerance used by **Complete** to compare angles. This tolerance is applied, for example, when checking whether two lines are parallel or a single line is horizontal. The number supplied must be greater than or equal to zero. The initial value is 0.000001 radians.

PD_AUTO_TANGENT_TOLERANCE

This function specifies the absolute distance tolerance used by **Complete** to check whether two elements are tangent. This tolerance is applied only if one of the extracted elements is circular. The tolerance is used to compare the radius of the circular element to the distance from the circular element center point to the other element. The number supplied must be greater than or equal to zero. The initial setting is 0.000001 mm.

PD_RESOLVE_MERGE_TOLERANCE

PD_RESOLVE_MERGE_TOLERANCE specifies the absolute tolerance used by the Parametric Design solver to decide whether the modification should result in the merging of two or more points, or in the deletion of small elements. The merge process can be eliminated by setting this tolerance to a negative number. The initial setting is 0.000001 mm.

PD_SHOW_*

The PD_SHOW_COLOR and PD_SHOW_LABEL_SIZE options correspond to the color and size settings in the **Icon Setting** dialog box.

PD_SHOW_USE_POSTFIX controls the display of dimensional constraints. When set to the default value of NO, the constraint value replaces the normal dimension text. If set to YES, the constraint value is displayed as a postfix to the main dimension text. Earlier versions of the software always displayed dimensional constraints as postfixes.

ADD_CURRENT_INFO

ADD_CURRENT_INFO "PD_ZONE" END causes the "PD_ZONE" info to be included in all new geometry. As a result, all new geometry is added to the zone.

PD_NEW_C_LINE_*

The PD_NEW_C_LINE_* options determine the default color, linetype, and visibility for any new construction lines that are created when constraints are created. The default color and linetype causes these construction lines to be displayed as green lines. Visibility of these construction lines can be set so they are either always visible or visible only when constraints are displayed.

PD_NEW_POINT_*

The PD_NEW_POINT_* options determine the default color, linetype, and visibility for any new points that are created. Typically, these points are created only when needed to fix point-on or reference-point constraints. The default color and linetype causes these points to be displayed as green ×'s (crosses). Visibility of these points can be set so they are either always visible or visible only when constraints are displayed.

PD_PREVIEW_COLOR

PD_PREVIEW_COLOR sets the default color used by the  **Preview** mode of **Solve**. This color cannot be set from the screen menu.

PD_DEFAULT_DIM_COLOR

PD_DEFAULT_DIM_COLOR sets the default color used when parametric dimensions are displayed for Dimension Driven Modification. This color cannot be set from the screen menu.

Toolbar and Classical Menu Support

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From Creo Elements/Direct Drafting Rev. 18.1 onwards, MFC-based toolbar and menu customizations (ASCII Format Layouts) created and saved in earlier versions can be loaded in the Creo Elements/Direct Drafting Fluent user interface.

To load the customizations:

1. Click **File** ► **Import/Export** ► **Import Macro** ► **Input Macro**.

The **Input Macro** dialog box opens.

2. Select the macro and click **Open**.

Alternatively, you can use the `INPUT` command to load the customizations.

Note

After you load the customized toolbars and menus in Creo Elements/Direct Drafting, you cannot modify or customize them.

You can also use the emulated “classical” menus and tables. You can load them by specifying their macro name.

Emulated "classical" menus and tables (referencing the `MENU` and `TABLE` commands) cannot be scaled to wrap to a changing number of rows and columns.

The number of rows and columns always remain fixed as defined in the underlying macro. Note that these menus are not saved for subsequent relaunch when you exit Creo Elements/Direct Drafting.

Displaying and Hiding Toolbars

To display or hide a toolbar:

1. Click **View** and then, in the **Viewing** group, click  **Toolbars**.
2. Click **Drafting**.
3. Click or clear the check box of a toolbar to show or hide the toolbar, respectively.

Excluding Toolbars

You can exclude a toolbar from loading in Creo Elements/Direct Drafting by specifying the name of the toolbar in a text file.

To exclude a toolbar, add the name of the toolbar in the `FilterToolBars.txt` file, which is available in the following path:

```
<Creo Elements/Direct Drafting Installation Folder>/  
fluent_ui
```

Note

Creo Elements/Direct Drafting creates a copy of the `FilterToolBars.txt` file in the AppData (Application Data) folder after every session. If you make any changes to the `FilterToolBars.txt` file, you have to remove its copy from the AppData (Application Data) folder to implement the changes.

To avoid redundancy, toolbars like **Command Line**, **Status Bar**, and **Main Toolbar** are excluded by default.

Managing Layouts

You can load, save, or reset toolbars in Creo Elements/Direct Drafting to manage layouts.

To load a toolbar:

1. Click **Miscellaneous** and then, in the **Toolbars** group, click **Load Toolbars**.

The **Load Toolbars** dialog box opens.

2. Select the toolbar and click **Open**.

To save a toolbar:

-
1. Click **Miscellaneous** and then, in the **Toolbars** group, click **Save Toolbars**.
The **Save Toolbars** dialog box opens.
 2. Under **Caption**, select the toolbar.

 **Tip**

To save multiple toolbars, press CTRL and select multiple toolbars.

3. Click **Save**.
The **Save Toolbar** dialog box opens.
4. Specify a name for the toolbar and click **Save**.

To reset a toolbar:

- Click **Miscellaneous** and then, in the **Toolbars** group, click **Reset Toolbars**.

 **Note**

You can activate the **Miscellaneous** tab when you customize the ribbon. For more information about customizing the ribbon, see [Customizing the Creo Elements/Direct Drafting Ribbon on page 32](#).

Customizing Toolbars and Toolbar Items

You can add, edit, or delete toolbars and toolbar items using the **Customize** dialog box.

To open the **Customize** dialog box:

- Click **File** ► **Customize**.

The **Customize** dialog box displays a list of toolbars and toolbar items in Creo Elements/Direct Drafting. When you start Creo Elements/Direct Drafting, all pre-defined toolbars are loaded by default, but are hidden. You can display a toolbar by clicking its check box in the **Toolbars** list of the **Customize** dialog box. You can

also display or hide toolbars using  **Toolbars** in the **Viewing** group on the **View** tab. For more information, see [Displaying and Hiding Toolbars on page 769](#).

 **Note**

You retain a toolbar layout across Creo Elements/Direct Drafting sessions. When you exit Creo Elements/Direct Drafting, the toolbar layout information is stored in the `layout_fm` file in the AppData (Application Data) folder. Creo Elements/Direct Drafting loads this file (if available) when you start a new session.

Creating a Custom Toolbar

To create a custom toolbar:

1. Click **File** ► **Customize**.
The **Customize** dialog box opens. The **Toolbars** pane opens by default.
2. Click **Add**.
The **Add Toolbar** dialog box opens.
3. In the **Caption Key** box, type a label for the toolbar.
Creo Elements/Direct Drafting displays this label in the user interface.
4. Click or clear the **Locked** check box to lock or unlock the toolbar, respectively.

 **Note**

You cannot delete a locked toolbar.

5. In the **Unique Name** box, type a unique name for the toolbar.
This name is internal to Creo Elements/Direct Drafting.
6. In the **On Right Click** box, type a command, function, or macro name.
This command, function, or macro is executed when you right-click the toolbar.
7. In the **Domain** box, type the name of the message file (if any).
Creo Elements/Direct Drafting refers to the message file for the localized version (if applicable) of this toolbar.

Note

Every toolbar and toolbar item in the Creo Elements/Direct Drafting user interface is associated with a specific domain file. The domain file contains the strings associated with the toolbar or toolbar items, which you can edit. The strings include:

- Caption
- Category
- Tooltip
- List Items

8. Click **OK**.

Tip

- You can place the toolbar anywhere in the Creo Elements/Direct Drafting user interface using a drag-and-drop operation.
- To display tooltips of the toolbar items, click the **Show Tooltips** check box in the **Toolbars** pane of the **Customize** dialog box.

Selecting a Toolbar Menu Style

You can select the look and feel of the toolbar menus. You can view the toolbar menus as icons, text, or both.

To select a toolbar menu style:

1. Click **File** ► **Customize**.

The **Customize** dialog box opens. The **Toolbars** pane opens by default.

2. In the **Text Options** box, select one of the following:

- **Text below**
- **Text on right**
- **Text only**
- **Icon only**

Editing a Toolbar

To edit a toolbar:

1. Click **File ► Customize**.

The **Customize** dialog box opens. The **Toolbars** pane opens by default.

2. In the **Toolbars** list, select a toolbar.
3. Click **Edit**.

The **Edit Toolbar** dialog box opens.

4. Edit the settings.
5. Click **OK**.

Deleting a Toolbar

To delete a toolbar:

1. Click **File ► Customize**.

The **Customize** dialog box opens. The **Toolbars** pane opens by default.

2. In the **Toolbars** list, select a toolbar.
3. Click **Destroy**.

Creating a Custom Toolbar Item

To create a custom toolbar item:

1. Click **File ► Customize**.

The **Customize** dialog box opens. The **Toolbars** pane opens by default.

2. Click **Toolbar Items**.
3. Click **Add**.

The **Add Toolbar Item** dialog box opens.

4. Define the following parameters in the **Add Toolbar Item** dialog box:

Parameter	Description
Caption Key	Type a label for the toolbar item. Creo Elements/Direct Drafting displays this label in the user interface.
Enabled	Click or clear the check box to enable or disable the toolbar item, respectively.
Unique Name	Type a unique name for the toolbar.

Parameter	Description
	This name is internal to Creo Elements/Direct Drafting.
Item Type	Select a standard MS Windows item type.  Note The item types in the Item Type list are context-sensitive. For example, you can select items from a context menu when you right-click a button that is created by using Popup Button .
Submenu	Select a submenu.  Note The Submenu list is available only if you select Popup Button in the Item Type list.
ToolTip Key	Type a brief description of the toolbar item.
On Action	Type a command, function, or macro name (for example, <code>change_color</code>). Creo Elements/Direct Drafting executes this command, function, or macro when you click the toolbar item.
On Right Click	Type a command, function, or macro name. Creo Elements/Direct Drafting executes this command, function, or macro when you right-click the toolbar item.
Category Key	Define a category for the toolbar item.
Domain	Type the name of the message file (if any). Creo Elements/Direct Drafting refers to the message file for the localized version of this toolbar item (if applicable).  Note Multiple toolbars may share the same domain file. Optionally, you can create a separate domain file for each toolbar.
Image	Define an icon for the toolbar item: a. In the Image area, click Edit .

Parameter	Description
	<p>The Image Editor dialog box opens.</p> <p>b. Use the drawing tools to design the icon.</p> <p>c. Click OK.</p> <p> Tip</p> <ul style="list-style-type: none"> To save the icon, click Store in the Image Editor dialog box. To use an existing image as an icon, click Load in the Image Editor dialog box.
List Keys	<p>Define an action string:</p> <ul style="list-style-type: none"> Click  to add an item to the list. Click  to remove an item from the list. Click  or  to scroll within the list of keys. <p> Note</p> <p>An action string requires an associated action definition in a separate macro.</p>

5. Click **OK**.

Note

Combo boxes, radio buttons, and check boxes require a separate macro. For details about the Creo Elements/Direct Drafting user interface customization capabilities, see `TOOLBAR_ITEM` in the *Creo Elements/Direct Drafting / Creo Elements/Direct 2D Access Programming Reference Guide*.

Adding an Icon to the Toolbar

You can add icons (as buttons) to the Creo Elements/Direct Drafting toolbars.

To add an icon to a toolbar:

1. Click **File** ► **Customize**.
The **Customize** dialog box opens. The **Toolbars** pane opens by default.
2. Click **Toolbar Items**.
3. In the **Categories** list, select a category.
4. Select a toolbar item.

-
5. Click **Edit**.

The **Edit Toolbar** Item dialog box opens.

6. In the **Image** area, click **Edit**.

The **Image Editor** dialog box opens.

7. Create an icon.

 **Tip**

To add an existing icon, click **Load** in the **Image Editor** dialog box.

8. Click **OK**.

The icon is associated with the toolbar item.

9. Drag the toolbar item to the toolbar.

 **Tip**

- To remove a button from a toolbar, drag the toolbar button to the **Customize** dialog box.
 - You can copy or move toolbar buttons to other toolbars.
 - Press CTRL and place a toolbar button on the destination toolbar (with a drag-and-drop operation) to copy the toolbar button.
 - If the **Customize** dialog box is not open, you can use ALT to move toolbar buttons or use ALT+CTRL to copy toolbar buttons to other toolbars.
-

Editing a Toolbar Item

To edit a toolbar item:

1. Click **File** ► **Customize**.

The **Customize** dialog box opens. The **Toolbars** pane opens by default.

2. Click **Toolbar Items**.
3. In the **Categories** list, select a category.
4. Select a toolbar item.
5. Click **Edit**.

The **Edit Toolbar Item** dialog box opens.

-
6. Edit the parameter values.
 7. Click **OK**.

Deleting a Toolbar item

To delete a toolbar item:

1. Click **File** ► **Customize**.
The **Customize** dialog box opens. The **Toolbars** pane opens by default.
2. Click **Toolbar Items**.
3. In the **Categories** list, select a category.
4. Select a toolbar item.
5. Click **Destroy**.

Note

- **Destroy** removes items from the **Toolbar Items** pane of the **Customize** dialog box and deletes the associated files.
 - You cannot undo **Destroy**.
-

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Managing drawings with Windchill

The Creo Elements/Direct Drafting and Windchill integration allows you to load, save, and manage data created by Creo Elements/Direct Drafting in Windchill.

Activating the Windchill PDM Actions from Creo Elements/Direct Drafting

1. Register Creo Elements/Direct Drafting with the Windchill Workgroup Manager:
Install and start the Windchill Unified Workgroup Manager (UWGM) and set the UWGM server as the primary server.
2. Activate the Windchill Workgroup Manager in Creo Elements/Direct Drafting:

Click **File** ►  **Modules** ►  **Windchill Workgroup Manager**.

The **Windchill** menu is now available in Creo Elements/Direct Drafting.

Windchill PDM commands

The following table describes the PDM commands that are inserted into Creo Elements/Direct Drafting when connected to the Windchill Workgroup Manager.

Action	Menu Location	Description
Open from Windchill	Windchill ► Open from Windchill [Also] File ► Open from Windchill	Opens the Open from Windchill window. By default, the contents of your active workspace are displayed. You can also navigate to the server common space to select an object to open.
Save to	Windchill ► Save to Workspace	Opens the Save to Workspace

Action	Menu Location	Description
Workspace	[Also] File ► Save to Workspace	window. Upon execution of the save, the document is uploaded to the server-side workspace.
Check out	Windchill ► Check out	Checks that the current drawing is not already checked out. If not, checks out the current document to the workspace.
Revise and Check out	Windchill ► Revise and Check out	Checks that the current drawing is not already checked out. If not, creates a new revision of the current file and checks the new revision out to the workspace, replacing the earlier revision in the workspace and in Creo Elements/Direct session.
Undo Check out	Windchill ► Undo Check out	<p>Cancels the check out of the current document.</p> <p> Note</p> <p>If the document was checked out via the Revise and Check out action, the higher revision label is maintained.</p>
Custom Check In	Windchill ► Custom Check In [Also] File ► Custom Check In	Accesses the Check In page and allows you to change check in options.
Auto Check In	Windchill ► Auto Check In	Automatically checks in the current file to the commonspace.
Load File from Windchill	Windchill ► Load File from Windchill	Opens the Open from Windchill window, allowing you to navigate to another document with drawing geometry to load into the current drawing.
Load Frame from Windchill	Windchill ► Load Frame from Windchill	Opens the Open from Windchill window, allowing you to navigate to a library containing

Action	Menu Location	Description
		frames and select a frame to load into the current drawing.
Save Frame to Windchill	Windchill ► Save Frame to Windchill	Opens the Save to Workspace window, allowing an administrator to save a new drawing recognized as a frame to an appropriate library workspace.
Text Reference	Windchill ► Text Reference	Calls or displays a submenu for text references enabling a user to assign, unassign, or update text references to document attributes. In addition, actions are available to display references, values, or a table of name-value pairs.
Workgroup Manager	Windchill ► Managers ► Workgroup Manager	Opens the Windchill Workgroup Manager (if closed); accesses the Workgroup Manager (if open).
Server Management	Windchill ► Managers ► Server Management	Opens the Server Manager window, where you can view or change Windchill servers and server active workspaces.
Event Management	Windchill ► Managers ► Event Management	Opens the Event Manager on the current server.
Update	Windchill ► Update	Checks the server for a later iteration. If one is found, replaces the drawing in session and in the workspace with the latest server iteration. An Update action also updates all drawing text references.

Action	Menu Location	Description
View Drawing Information	Windchill ► View Drawing Information	Opens the server information page (workspace view) for the drawing in session.
Windchill Workgroup Manager Help	Windchill ► Windchill Workgroup Manager Help [Also] Help ► Windchill Workgroup Manager Help	Opens the Windchill Workgroup Manager Help Center.

For more information on the Windchill PDM commands, see the Windchill WGM help from the Creo Elements/Direct Drafting user interface.

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Unicode

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Introduction

Creo Elements/Direct Drafting's different internal and external encoding schemes were Unicode incorporated for version 2007. Unicode allows text and symbols from all languages to be consistently represented and manipulated by computers.

All textual and string data processed by Creo Elements/Direct Drafting is held internally in Unicode (UTF-8) encoding. That means most of Drafting's functionality processes user input from almost any language, and string data from different languages can be mixed. For example, it is now possible to have German, Japanese, and Russian characters within a single string, such as a part name.

Drafting's font list contains all the Creo Elements/Direct true type fonts that are dynamically converted to internal font format when referenced. An appropriate font that contains all desired characters must be selected to display texts and dimension texts correctly. Fonts containing characters from the whole Unicode range are relatively rare, and they usually contain the word Unicode in their name (for example, Arial Unicode MS).

Symbols are a special Creo Elements/Direct Drafting feature and display with `hp_symbols` and `hp_symbols2` fonts. They can still be composed in old-style escape sequences (`#15#XY#16`) but for compatibility with Unicode are mapped to a private-use Unicode area [E000-E1FF]:

- `15#XY#16` — `0xE000 + XY`
- `30#XY#31` — `0xE100 + XY`

MI file format

Version 3.20 MI files are encoded in Unicode (UTF-8). Files stored to older MI versions are encoded as before, depending on the current locale. When loading older MI formats, the Creo Elements/Direct Drafting loader uses the information recorded in the info section (~1) of the MI file. MI formats without encoding information in their info sections are treated as if encoded in ROMAN8 or SJIS (Japanese locale).

Macro file encoding

Old macro files did not contain any encoding information. They were assumed to be encoded for current locales (e.g. ROMAN8 for English and SJIS for Japanese locales). However, this causes problems when using the same macros in different locales.

Version 2007 supports macro Unicode but maintains backward compatibility.

We recommend you change old macro files such that their encoding is explicitly specified with the `DEFINE_ENCODING` directive.

Text macro files containing Unicode UTF-8 or UTF-16 BOMs (Byte Order Markers), supported by most popular Windows text editors, are read as Unicode.

Files containing the `DEFINE_ENCODING` directive are always read in the specified encoding. Otherwise they are treated as old macro files and encodings are locale dependent:

- SJIS for Japanese
- BIG5 for traditional Chinese
- GB2312 for simplified Chinese
- ROMAN8 for all else

Limitations

The displaying of Unicode texts is limited. Features such as combining characters, Arabic cursive connection, and bidirectional algorithm are not fully implemented.

`CHR`, `NUM` and `#` still work in old locale-dependent internal encodings (ROMAN8 & SJIS). They return different results in different locales. They are obsolete; their use is only for maintaining backward compatibility. For example, `((CHR 154)+(CHR 223))` or its equivalent `(#154#223)` displays a Japanese character with Unicode code 22756 (hex 58E4) on a Japanese locale and two Latin characters on other locales. To avoid problems and to display the same character regardless of locale, use Unicode versions of these functions: `(UCHR 22756)`, `(UCHR '58E4')`, `#u22756`, or `#ux58E4`.

Combinations of `#` and `CHR` to compose 2-byte characters and symbols are not allowed. For example, `((CHR 154)+#223)` does not work.

Strings that begin with `(CHR 255)` are icons and are not encoded in UTF-8. Pre-2007 CoCreate Drafting versions let you compose icons using a large variety of combinations (for example `#255'abc'#212+CHR 12`). In version 2007, only `CHRs` can be used to compose icons.

MI files of versions prior to 2.90 that do not contain encoding information in their info section (~1) load correctly only if loaded in the same locale as they were drawn. For example, you have 2 options to correctly load a Japanese MI file of version 2.80 in English Creo Elements/Direct Drafting:

1. Load the file in Japanese locale and store it back in new format:
 - a. `CHANGE_LOCALE 'ja'`
 - b. `LOAD 'japanese.mi'`
 - c. `STORE MI ALL DEL_OLD 'japanese.mi'`
 - d. `CHANGE_LOCALE 'en'`
2. Load the MI file in a text editor and manually add section ~1 at the beginning of MI file:

-
- a. #~1
 - b. ENCODING:SJIS
 - c. Such files load correctly in any locale.

All modules call internal Creo Elements/Direct Drafting functions and use strings in internal encoding. Before CoCreate Drafting 15.00 (2007) these strings were encoded in ROMAN8 or SJIS, depending on locale. Compatibility is maintained only for characters in range [0, 127]. If Application Interface (AI) modules use characters in a different range, they must be changed and recompiled to support UTF-8.

File names encoding

In CoCreate Drafting 2007, file names display in the same fashion as other Windows applications. Prior to 2007 they were shown in the internal encoding, which was different and dependant on locale. Files names were consequently shown differently if extended characters were used.

You can set the MEEXTFN option in the .ini file to zero to keep the old behavior.

Changed font concept

CoCreate Drafting 2007 does not distinguish between 1-byte and 2-byte fonts (in terms of default settings or text properties). Because Creo Elements/Direct Drafting now has Unicode support, there is just one current font at a time and each text and dimension text has just one assigned font.

CoCreate Drafting 2007 includes new default fonts which have been combined from the old 1- and 2-byte definitions:

- osd_default combines hp_i3098_v and hp_kanj2_c
- osd_default2 combines hp_i3098_c and hp_kanj2_c
- osd_default3 combines hp_d17_v and hp_kanj2_c

When loading old 2D files with texts or dimension texts using characters from 1- and 2-byte fonts (fonts that are not already included in any of the osd_default fonts), Creo Elements/Direct Drafting automatically combines the 2 fonts into a single font containing the character definitions from both fonts. It names them "1-byte=font name"#"1-byte=font name".

You can combine those fonts (1- and 2-byte fonts) into a single font using a new command MERGE_TWO_FONTS.

TrueType fonts converted to Creo Elements/Direct Drafting fonts contain all defined characters. Separate symbol fonts are used.

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