Creo Elements/Direct Drafting User's Guide: Windows User Interface

Creo Elements/Direct Drafting 20.4.4.0

Contents

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, annotating, 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 Manual

Read this manual 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 manual is written for users who have limited experience with CAD software.

Purpose of this Manual

This manual gives information on:

- 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 Manual

Use this manual 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 Manual is Organized

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

Chapter 1	Getting Started with Creo Elements/Direct Drafting — a brief overview of the system and supplies you with hints to help in getting started.
Chapter 2	Preparing the Drawing Environment — how to set up the environment and store it as a file.
Chapter 3	Commonly Used Commands — how to escape from a current activity.
Chapter 4	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 5	Using Creo Elements/Direct Drafting's Copilot Functionality — how to create a design drawing with the minimum of construction geometry.
Chapter 6	Drawing Methods — how to create a drawing with or without the aid of construction geometry.
Chapter 7	Modifying a Drawing — how to modify your drawing.
Chapter 8	Dimensioning a Drawing — how to automatically dimension a drawing.
Chapter 9	Hatching a Drawing — how to hatch drawing sections.
Chapter 10	Adding Text To a Drawing — how to add text to a drawing.
Chapter 11	Adding Symbols to a Drawing — how to add geometric tolerance symbols and other special annotation to your drawing.
Chapter 12	Handling Parts and Assemblies — how to use parts and assemblies when creating drawings.
Chapter 13	Generating Hidden Lines — how to generate hidden lines in assembly drawings.
Chapter 14	Using Layers and Infos — how to use layers and how to add textual information (infos) to elements in your drawing.
Chapter 15	Storing, Loading, and Filing — how to load and save your Creo Elements/Direct Drafting drawing files.
Chapter 16	Revision Checking — how to compare revised drawings.
Chapter 17	Editing Imported Bitmap Files — how to handle imported bitmap files in Creo Elements/Direct Drafting.
Chapter 18	Linking and Embedding OLE Objects — how to insert OLE objects into a Creo Elements/Direct Drafting drawing.

Chapter 19	Creo Elements/Direct Annotation — how to visualize, annotate, and edit MI files created by means of Creo Elements/Direct Annotation.
Chapter 20	Plotting Drawings and Dumping Screens — how to plot your finished drawing and how to dump screens.
Chapter 22	Customizing the Creo Elements/Direct Drafting User Interface — how to add custom icons to the toolbar, how to create custom menus, how to define custom key commands, and how to display menus and dialog boxes via mouse buttons.
Appendix A	Using Filetran — how to convert file formats between UNIX and Microsoft Windows systems.

1

Getting Started with Drafting

The Creo Elements/Direct Drafting User Interface

The Creo Elements/Direct Drafting user interface is designed to ensure a short learning curve and to provide maximum efficiency in 2D CAD projects.

All of the menus, dialog boxes, commands, functions and tools specified in this manual are accessible via the Creo Elements/Direct Drafting user interface. The following figure illustrates the general (default) layout of the Creo Elements/Direct Drafting user interface.

Figure 1. The Creo Elements/Direct Drafting User Interface
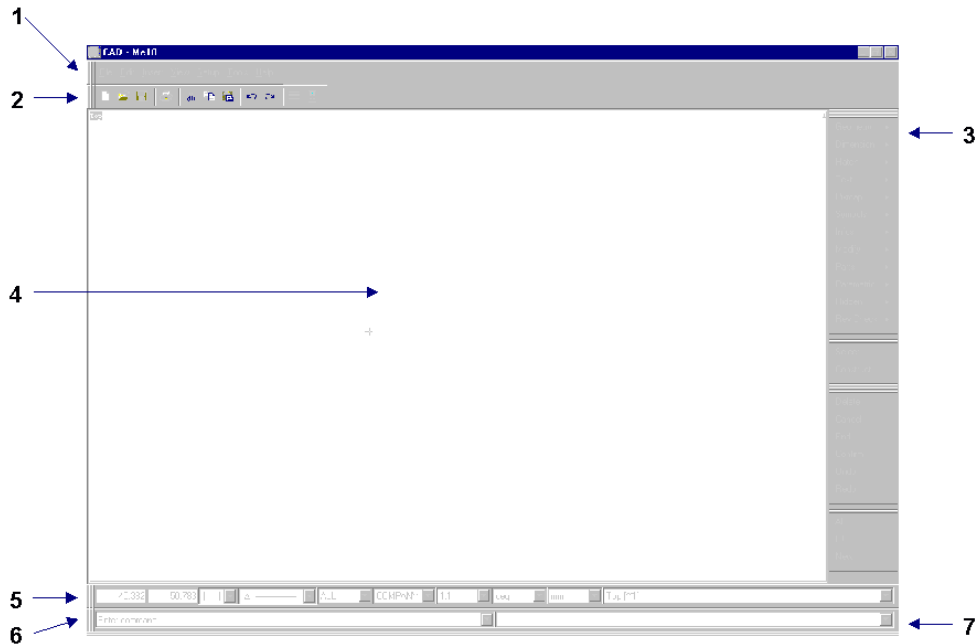
1. The top menu bar. This menu bar provides access to menus such as **File** and **Setup**. Optionally, you can use hotkeys to select menu items.
2. The toolbar. Move the cursor to each of these icons to display the associated tooltips.
3. The design and drafting related menus.
4. The drawing area. This is the area where you will create and modify drawings. Additional viewports can be created in this area.
5. The bottom menu bar (which includes the Status Line). The first two items in this menu bar reflect the current cursor position. The other items enable you to select or specify settings such as the current linetype, color, catch range, dimensioning standard, layer, scale, angular unit, length unit and current part.
6. The prompt line. This line tells you how to proceed to complete the current operation.
7. The user input line. When prompted, enter the required input (such as a numerical value or a name) into this line.

Optionally, you can run multiple Creo Elements/Direct Drafting instances (separate Creo Elements/Direct Drafting windows) at the same time. Refer to Creating a New Creo Elements/Direct Drafting Instance on page 434 for further information.

Basic Working Methods

It will not take you very long to become familiar with Creo Elements/Direct Drafting, but a few hints may help you to get started.

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

An index is included at the back of the manual; this 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** functions which operate via the Clipboard. The **Select** functions (including **Poly**; see Selecting Drawing Elements on page 39) can be used to facilitate the selection of elements to be cut or copied.

	Enables you to cut elements.
	Enables you to copy elements.
	Enables you to paste elements.

The handling of elements via the Clipboard can be manipulated via the **Clipboard** tab. Click **Setup**, **Edit Environment**, **System**, **Clipboard** to display this tab. Via this tab, you can change the following settings:

Inverse Black and White	White changes to black, while black changes to white.
Global Selection	Like the **Global** selection function (see Selecting Drawing Elements on page 39), 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 will be smashed.

Paste As Subpart

When you have cut or copied an item, you can use **Paste As Subpart** to paste this item such that it is owned by the current part. Refer to Hierarchical Parts Structure on page 338 for details on the Creo Elements/Direct Drafting parts structure concept.

Undo and Redo

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

Click ↰ to undo an operation (before you click **Confirm**).
Click ↱ to redo an operation (after you have clicked **Undo**).

Each time you click Undo/Redo, you will undo or redo all the changes that were created by the last command you entered.

You can set a limit on the number of steps to keep in the Undo/Redo history in the System Setting dialog.

1. In the main menu, click **Setup ▸ Edit Environment ▸ System**.

2. You can select:

   • **-1**: no limit.

   • **0**: disable the Undo/Redo option.

   • **+[number]**: the maximum number of steps you can undo or redo when no command is active. If you already have more steps in your history than the number you selected as the limit, Creo Elements/Direct Drafting will automatically delete the extra steps.

Limitations:

• You can undo or redo complete macros, but you cannot use Undo/Redo inside macros.

• Undo/Redo cannot undo or redo:

   ○ Commands related to parametric design, hidden lines, PIXMAPs, or OLEs. 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/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/redo history. Creo Elements/Direct Drafting marks the new state with a unique ID. With this information, you can:

• Use UR_GET_CURRENT_ID to see the ID of the current undo/redo state.

• Use UR_ID_EXISTS to see if a specified ID exists in the undo/redo history.

• Use UR_GET_FIRST_ID to see the ID of the first state in the undo/redo history.

- Use UR_GET_LAST_ID to see the ID of the last state in the undo/redo history.
- Use UR_MOVE to undo/redo the drawing to a specified state.

Select, Click, Specify, and Enter

Many operations require similar user interaction, For example, you will often be 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 on 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.

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

Positioning Menus

Menus can be repositioned anywhere around the screen so that you can display several menus at a time. The menus can be overlapped and closed as needed for your current work.

Additionally, menus and submenus can be "pinned" at a certain position. Pinned menus always open at the position where they were pinned.

By opening and pinning menus or submenus, you can quickly and easily create a custom menu layout. Note that this menu layout is automatically retained by Creo Elements/Direct Drafting (for future sessions) until you change it or until you click **Tools**, **Customize** and **Reset Layout** to restore the initial (default) Creo Elements/Direct Drafting user interface.

How to Delete Elements

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

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

Getting Help

The Programmer's Reference file contains technical reference information about all the commands and functions that you can use within Creo Elements/Direct Drafting.

To use the help system:

1. Click **Help ▸ Program Ref. Guide**.
2. Input the command or function on which you require help by either:

    • Clicking on a command or function from the screen menu.

    • Entering the command or function name.

   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 display.

Switching Languages

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

Language	Command
English	`change_locale 'en'`
French	`change_locale 'fr'`
German	`change_locale 'de'`
Italian	`change_locale 'it'`
Spanish	`change_locale 'es'`
Japanese	`change_locale 'ja'`

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

📋 Note

The Japanese version of Creo Elements/Direct Drafting requires a Japanese operating environment. If you installed Creo Elements/Direct Drafting in Custom mode, any languages you deselected during installation will not be available at Creo Elements/Direct Drafting runtime.

Switching to the Classic User Interface

Optionally, you can run Creo Elements/Direct Drafting using the Classic User Interface. Note that a separate User's Guide is available for this user interface.

To run the Creo Elements/Direct Drafting Classic User Interface, you need to set the following variable in the `me10.ini` file (located in the Creo Elements/Direct Drafting installation directory):

`MEPELOOK=2`	Creo Elements/Direct Drafting uses the standard Windows style user interface.
`MEPELOOK=3`	Creo Elements/Direct Drafting displays any top-level menus in the usual Windows style (without the arrow indicating cascading), whereas menus cascaded from the right-hand menu bar are displayed in the "classic" style. Optionally, you can enter the `WUI_CLASSIC` command into the user input line to remove the separator lines from the "classic" style menus in this display mode.

Switching the Beeper On or Off

Optionally, you can switch the "beeper" on or off in Creo Elements/Direct Drafting. Simply enter `BEEPER ON` or `BEEPER OFF` into the user input line.

Further Reading

You can now begin to 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 advice on macro writing and command syntax.

2

Preparing the Drawing Environment

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.

The **Edit Environment** command in the **Setup** menu provides access to a number of dialog boxes which can be used to define settings for your design and drafting environment. For example, you can use **Construction Settings** to select the color and linetype in which construction geometry will be displayed in your drawing.

Some of the settings can be set in a particular menu, such as Text Size in the **Text** menu, while many of the dimension settings can be set using the commands contained in the Creo Elements/Direct Drafting environment.

Setting the Environment

The **Edit Environment** command options are:

Construction	Edit the construction geometry settings, for example, line color.
Geometry	Edit the geometry settings, for example, line color.
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.
Parts	Edit the part representation settings, for example, spotlight color.
Line Patterns	Add/edit user-defined line patterns.
Projected Reference Points	Edit the settings (such as color or linetype) for projected reference points and their associated lines and circle.
Refline	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.
System	Edit general settings for the Parametric module, for the Copilot functionality, for the Clipboard, Handles, FlyBy, for the autosave mechanism, for the menu cascade timeout and for the redraw mode.

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]\`.

Saving the Creo Elements/Direct Drafting Environment to a File

When you type `SAVE_ENVIRONMENT` in the User Input Line, Creo Elements/ Direct Drafting prompts you to enter one of the following options:

`Screen`	Outputs the environment to the screen.
`Printer`	Outputs the environment to a printer.
`Append`	Appends the environment to an existing environment.
`Delete Old`	Overwrites the existing environment file.
`'file name'`	Enables you to specify the name of a file in which the current environment settings will be stored.

*   If you wish to save a particular Creo Elements/Direct Drafting environment, enter the applicable filename (such as `'Company_environment'`) into the User Input Line.

*   To apply the settings of a saved environment file, you can input the file at the start of a drawing session (Enter **INPUT 'filename'**).

Further details of using the environment file are contained in the manual "Writing Macros with Creo Elements/Direct Drafting".

Setting Linear and Angular Units

You can select linear and angular units from the bottom menu bar. The current units are displayed in the System Status Line. Normally, you will set the current units before starting to draw. However, if you reset the current units after you have started to draw, Creo Elements/Direct Drafting will automatically convert all existing elements without changing the lengths.

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

The following options are available:

Linear units:

*   mm
*   um (micrometer)
*   inches
*   feet
*   yards
*   miles
*   mils
*   uinches
*   usr

Angular units:

*   deg
*   grd
*   rad
*   usr

Defining Your Own Units

The **usr** option in the selection lists for angular and linear units allows you to define your own units. 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. Pull down the **File** menu.
2. Click **Open** to display the Creo Elements/Direct Drafting File Browser.
3. Click **Navigator** to display the Creo Elements/Direct Drafting Navigator.
4. In the left pane of the Navigator, change to the Creo Elements/Direct Drafting installation directory.
5. Double-click the preview image of a drawing frame.

The frame is a symbol part which is always loaded below the Top part (refer to Handling Parts and Assemblies on page 335). 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 will not normally be plotted out.

Drawing frame part names have the form:

`frame_A4[symbol], frame_A3[symbol],` and so on.

The ten frame options are:

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

Setting the Drawing Scale

The drawing scale will affect 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 System Status Line. 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 will be plotted 200 mm long.

To set the drawing scale:

1. Pull down the **Setup** menu.
2. Click the **Environment** tab.
3. Enter the drawing scale in the **Scale** field.

The result can be checked by looking at the System Status Line.

Further Information

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

Enter `help environment` at the Creo Elements/Direct Drafting command line.

The screen will clear and Creo Elements/Direct Drafting will display further information about the environment.

3

Commonly Used Commands

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

Use the Delete key on your keyboard or the Delete command from the menu to delete elements.

To use Delete on the keyboard,

1. Click an element in the drawing area. The handles appear.
2. Press the **Delete** key on your keyboard.

Delete multiple elements:

1. Hold the Shift key and click all the elements in the drawing area. The handles appear.
2. Press the Delete key on your keyboard.

To use the Delete button,

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

For example, to delete everything in the active part:

1. Click **Delete**.
2. Click **All**.
3. Click **Confirm**.

To delete an element or text:

1. Press **Delete**.
2. Select the element or text to be deleted (you can also enclose the elements to be deleted in a box).

Deleting Hatching

To delete hatching, use the **Delete** command in the **Hatch** menu, then select the hatching to be deleted individually. See Hatching a Drawing on page 275 for. more information.

Highlighting Elements Before Deletion

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

1. Click **Delete**.

2. Click **Select** followed by any option or combination of options described below.

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

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:

1. Click **Select**.

2. Click **Global**.

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

Example

Click **Delete**, **Select**, **Global**, **All**, **Confirm**, **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. Example of a Parts Structure on page 37 show the power of **Select**.

Figure 2. Example of a Parts Structure
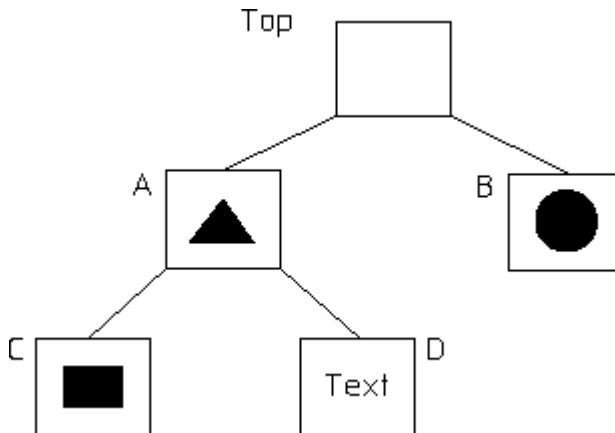
If you delete globally with 'B' as the active part, the Top part and 'B' will remain. See Figure 3. Top Part and 'B'. on page 38.

Figure 3. Top Part and 'B'.
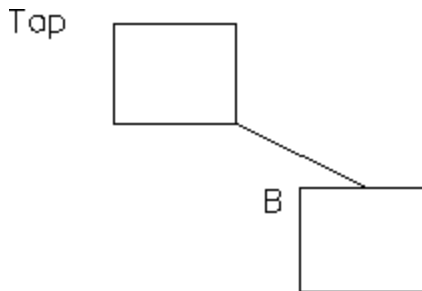
If you delete globally with 'A' as the active part, the Top part and 'A' will remain. See Figure 4. Top Part and 'A'. on page 38.

Figure 4. Top Part and 'A'.
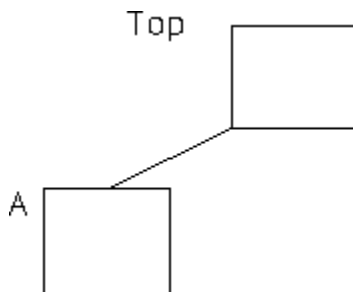
If you delete globally with 'D' as the active part, the Top part, 'D', and its parent, 'A', will remain. See Figure 5. Top Part + D + A. on page 38.

Figure 5. Top Part + D + A.
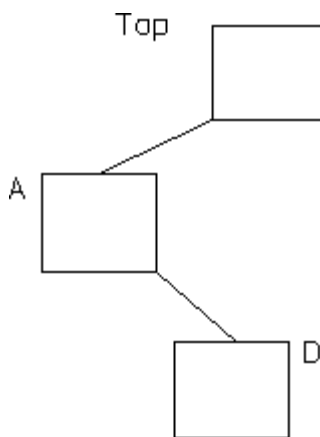
Selecting Drawing Elements

Click **Select** to let Creo Elements/Direct Drafting know you are about to make a selection (for example, to move or to delete elements). Then click any option or combination of options from the selection block.

Option	Description
All	Selects all elements in the active part.
Box	Selects all elements you enclose in a box.
Poly	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 48.
Advanced	Displays a separate **Select** dialog box which provides functions and options for complex selections. These are discussed in the following section.

Advanced Selection Functions

Click **Select** and then **Advanced** to display the **Select** dialog box. The functions and options included in this dialog box enable you to define complex selections. These include the **All**, **Poly**, **Box**, **Subtree**, **PreSelected**, **Global** and **Global_Perm** selection functions described above.

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.
Contours	Selects contours (see Selecting Contours on page 45).
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.
Hatch	Selects hatches.
Text	Selects text.
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.
CLine	Selects construction lines.
CCircle	Selects construction circles.
Part	Selects parts.
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** buttons.
Close	Closes the **Select** dialog box.

All 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 press **Select** and then specify the respective color or linetype.

1. Click **Delete**, **Select**, **Advanced** to display the **Select** dialog box.
2. In the **Select** dialog box, click **Select** and **Color**.
3. Select **Green** and click **OK** in **Select Color**.
4. In the **Select** dialog box, click **Confirm** and **OK**.

This will delete 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. Click **Select**.
3. Select several elements to be deleted.
4. Click **Add**.
5. Click **Box**.
6. Enclose further elements to be deleted in a box.
7. Click **Confirm**.
8. Click **OK**.

Subtracting Elements from Your Selection

To subtract elements from your current selection, click **Subtract**.

Removing Elements from Your Selection

Click **Not** 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**.

2. Enclose the required elements in a box.

They will then be 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.

1. Click **Confirm**.

2. Click **OK**.

Enclosing Elements in a Polygon

If you need greater flexibility when selecting elements, use **Polygon** 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
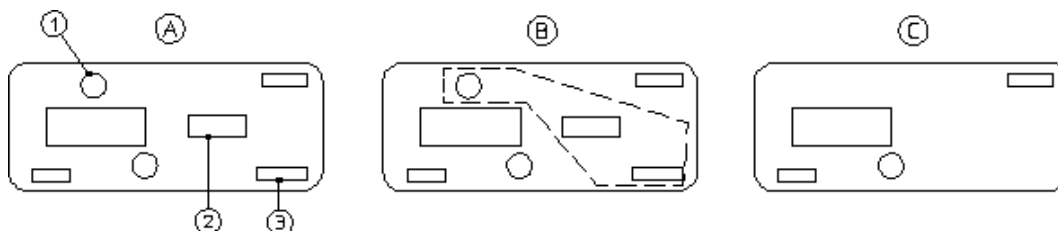
Suppose you want to delete the elements 1, 2, and 3:

1. Click **Delete**.

2. Click **Select**, **Advanced**, **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 enter CLOSE.

Selecting All Elements

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

1. Click **Delete**.

2. Click **All**.

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.

1. Click **Confirm**.

2. Click **OK**.

Selecting a Vertex

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

Selecting Elements by Type

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

Selecting Elements Using Information Strings

Click **Info** in **Select** 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 **Select** 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 **Select** 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 **Select** to select reference lines.

Selecting Construction Geometry

Click **CGeo** in **Select** 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 **Select** 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 El** to select polyline elements (see Creating a Polyline on page 120).

To make a selection on pensize or linesize:

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

Selecting Parts

Click **Part** in **Select** 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**, **Select**, **Global**, **Part**.
2. Enter the part name.
3. Click **Confirm**.
4. Click **OK**.

Note

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

Selecting Dimensions

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

Selecting Contours

The **Contours** option in **Select** enables you 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 46 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. Display the **Modify** menu and click **Move**.
   2. Click two points to define the distance and direction of the move.
   3. Click **Select**, **Advanced**.
   4. In the **Select** dialog box, click **Select**, **All**, **Contours**, **OK**.
   5. Click **Confirm** to move the selected contours.

- To delete all elements other than contours:

   1. Click **Delete**.
   2. Click **Select**, **Advanced**.
   3. In the **Select** dialog box, 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 **Geometry**, **Change**, **Color**.
   2. Set the color to **Green** and click **OK**.

3. Click **Select**, **Advanced**.

4. In the **Select** dialog box, 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.

`contour_level 0`	This is the default. Only the "outer" contours in the selection polygon are selected. See the part labeled **0** in Figure 7. Contour Levels on page 47.
`contour_level 1`	In addition to the outer contours, the first contours encountered within the outer contours are selected. See the part labeled **1** in Figure 7. Contour Levels on page 47.
`contour_level 2`	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. Contour Levels on page 47.
`contour_level 3`	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. Contour Levels on page 47.

Figure 7. Contour Levels
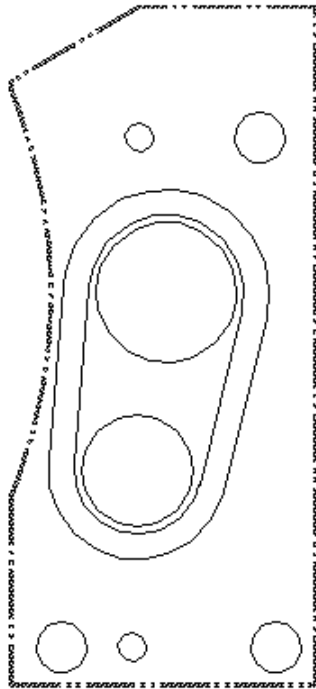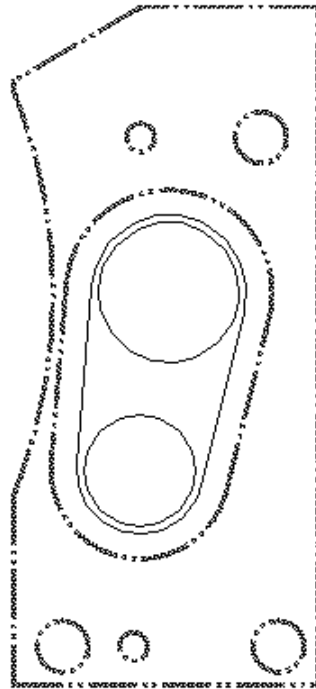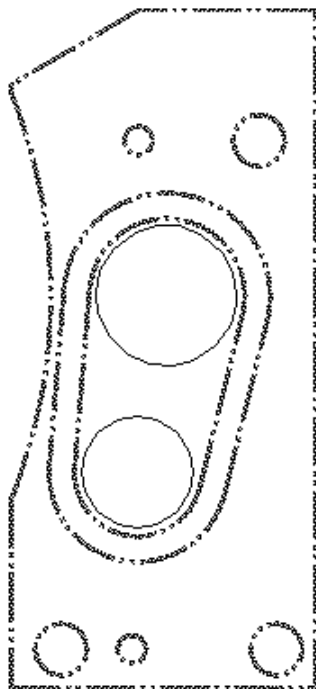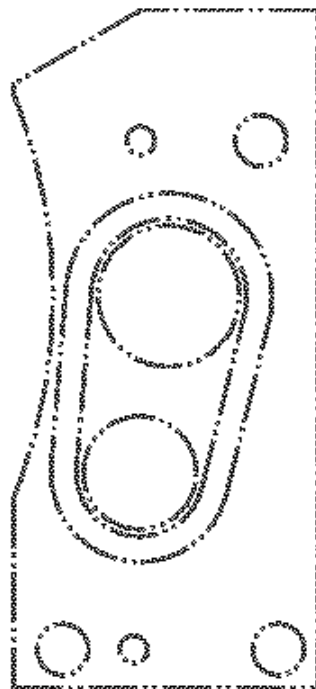
Copying Contours to the Clipboard

The **Copy Contour** command in the **Edit** menu 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.

Proceed as follows to copy a set of contours:

1. Specify the Specifying the Contour Level on page 46 for the selection you are about to make.

2. Click **Edit**, **Copy Contour**.

3. Click on points in your drawing to define a selection polygon around the area from which you want to select contours. Note that contours fully enclosed in this polygon are highlighted.

4. In the user input line, enter `contours`.

5. Click **End**.

Selecting URLs

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

Preselecting Elements

By preselecting elements, you can keep these available for subsequent operations. Creo Elements/Direct Drafting will "remember" your preselection until you replace it by another preselection.

Defining a Preselection

To preselect elements:

1. Click the Preselection icon in the Creo Elements/Direct Drafting toolbar:
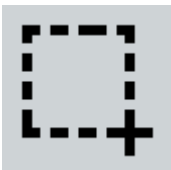
1. Select the elements you want to include in your preselection.

2. Click **Confirm**.

Retrieving a Preselection

To retrieve your preselection:

1. Click **PreSelected** or **Advanced** in the **Select** menu
2. Click **Confirm**.

Automatic Preselection

With automatic preselection enabled, you can add elements to the preselect list and use them in Modify commands without using the Select menu or Preselect command. To enable this option, click **Setup ▸ Edit Environment ▸ System ▸ Preselect ▸ Automatically use preselection**.

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 **End** 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 **End** 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.

Getting Help

The Programmer's Reference file contains general technical information about all the commands and functions that you can use in Creo Elements/Direct Drafting.

To use the help system:

1. Click **Help ▸ Program Ref. Guide**.
2. Enter the name of the command or function about which you require help.

   Creo Elements/Direct Drafting responds by displaying the required help information.
3. Exit the browser to return to the normal display.

4

Changing the Drawing Environment and Making Measurements

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** command to draw a fillet radius, you may want 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. Pull down the **Setup** menu and click **Edit Environment**.

2. Click the item for which you want to set the linetype, line color and line width or line size:

   • **Construction**

   • **Geometry**

   • **Hatch**

   • **Refline**

   • **Symmetry Line**

   • **Center Line**

3. Select the linetype from the **Type** list.

   Figure 8. Linetypes and Corresponding Symbolic Representation on page 53 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 54.

Figure 8. Linetypes and Corresponding Symbolic Representation

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

4. Select the line color from the **Color** list. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

5. Specify the line width (pen size) or line size:

   • To specify the line width (pen size), press the **Width** radio button and enter the required value.

   • To specify the line size, press the **Size** radio button and enter the required value.

   Note that 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 56.

The last line setting you defined is displayed at the bottom of the Creo Elements/ Direct Drafting display. 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(s) from the list box(es) at the bottom of the Creo Elements/Direct Drafting display.

Creating User-Defined Linetypes

In addition to the standard linetypes supplied with Creo Elements/Direct Drafting (see Figure 8. Linetypes and Corresponding Symbolic Representation on page 53), 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:

1. Pull down the **Setup** menu and click **Edit Environment**, **Line Patterns**.

2. Click **Add** in **Line Pattern**. Alternatively, display the linetype menu from the bottom menu bar and click on the questionmark item.

3. In the **Name** field in **Add Line Pattern**, specify a name for the line pattern you are about to create.

4. In the **Pattern** field in **Add Line Pattern**, enter 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, ...) number denotes the number of dots and the second (forth, sixth,

...) 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 that both of these examples produce the same linetype.

5. Click **OK** to save the new linetype.

The following figure gives you a visual impression of the new linetype defined in the two examples above. Note that the newly defined linetype pattern is appended to the list of built-in linetypes.

Figure 9. Eight Built-In Linetypes Plus a User-Defined Linetype
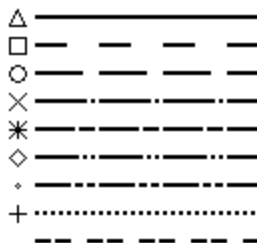
To apply your new linetype, follow the instructions in Setting Linetype, Line Color and Line Width or Line Size on page 53.

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 linetype menu.

Editing User-Defined Linetypes

To edit a user-defined (custom) linetype:

1. Pull down the **Setup** menu and click **Edit Environment ▸ Line Pattern**. Alternatively, display the linetype menu from the bottom menu bar and click on the questionmark item.

2. In **Line Pattern**, select the user-defined linetype to be edited.

3. Click **Edit** and edit the linetype as appropriate. Refer to Creating User-Defined Linetypes on page 54 for guidance.

Deleting User-Defined Linetypes

To delete a user-defined (custom) linetype:

1. Pull down the **Setup** menu and click **Edit Environment ▸ Line Pattern**. Alternatively, display the linetype menu from the bottom menu bar and click on the questionmark item.

2. In **Line Pattern**, select the user-defined linetype to be deleted.

3. Click **Delete**.
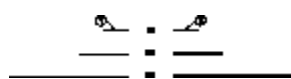
Line Width vs. Line Size

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. Difference between Pensize and Linesize on page 57 shows one benefit of specifying pensize and linesize for specific elements.

Figure 11. Difference between Pensize and Linesize
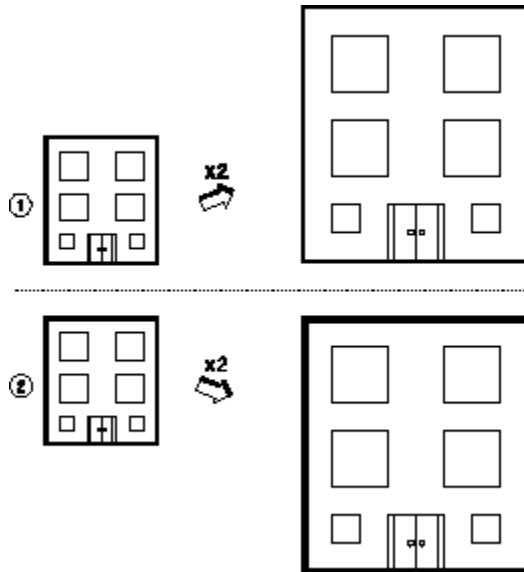
1 All geometry elements are drawn with the property pensize. When scaled, all lines remain the same size (thickness).
2 The outline of the building is drawn with the property linesize. All other elements are drawn with the property pensize. When scaled, the thickness of the outline increases but the other elements remain the same thickness.

Figure 12. Lines Drawn with Different Sizes on page 57 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
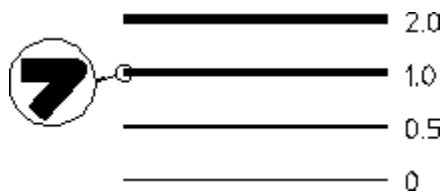
Mapping Pen Color, Linetype and Pen Width

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**

Proceed as follows to set up the **Pens** map:

1. Display the **Settings** dialog box of one of the entity types listed above. For example, pull down the **Setup** menu, click **Edit Environment** and select **Dimensions** to display the **Geometry Settings** dialog box.

2. Click **Pens** to display the **Pens** map.

3. In the **Pens** map, click the row you want to change. This row will be highlighted.

4. Click **Change** to display the **Pen Settings** dialog box.

5. In the **Pen Settings** dialog box, select a linetype and a line color, then enter the line width (pen size) 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 56.

6. Click **OK** to return to the **Pens** map.

7. Repeat this procedure until the **Pens** map contains the required combinations of pen color, linetype and pen width.

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

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

To change the linetype, color, line width and/or pen width of existing geometry:

1. Click **Change** in **Geometry**.

2. Select the appliable item:

   - **Linetype**

   - **Color**

   - **Pen Width**

   - **Line Width**

3. Select or specify the new setting. Note that you can change all four settings before you select the element(s) to which these changed settings will be applied.

4. Select the elements to be changed.

- 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.

- Note that 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. Linetypes and Corresponding Symbolic Representation on page 53.

- The spacing of all broken linetypes is set using the plot default option Ltype Len in PLOT.

Changing Color

Colors can be selected or changed 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

Proceed as follows to define custom colors for use in Creo Elements/Direct Drafting:

1. In any of the **Color** drop-down list boxes, select **Custom** to display the **Color** menu.

2. A number of basic colors are shown in the upper section of this menu. Click on the basic color(s) you want to add to your color selection list.

3. If you want to create colors other than these basic colors, click **Define Custom Colors** to display the Creo Elements/Direct Drafting **Color** dialog box.

4. In the **Color** dialog box, move the color selection recticule to change the associated hue, saturation, luminosity, red, green and blue values. Note that these values are indicated in separate fields. Optionally, you can replace these values to create a custom color.

5. When the required color is shown in the **Color|Solid** field, click **Add to Custom Colors**. The newly added custom color is shown in the **Custom colors** section of the **Color** dialog box.

6. Optionally, repeat the last two steps to create additional custom colors.

7. Click **OK**.

Note that the last custom color you defined becomes the current (active) color in the **Color** selection list. To access your other custom colors (if any), select **Custom** from the **Color** selection list.

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 System Status Line can be displayed only in the range of colors supported by the hardware.

Setting the Color for Parts Spotlight and Vertices

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

1. Pull down the **Setup** menu.
2. Click **Edit Environment** and **System**.
3. In the **System** dialog box, click on the **Geometry** tab.
4. Select the required colors for the parts spotlight and for vertices from the associated **Color** selection lists.

    Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.
5. 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:

- Type WUI_CREATE_COLOR_MAP in the user input line and press ENTER.

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 **Source One 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.

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**.

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.

📝 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.

• 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 64 for more information.

Changing the Catch Mode

The **Catch** function 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 System Status Line. 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	Catches to
VERTEX	Vertex
ELEM	Element
INTERSECTION	Intersection
CENTER	Center point
GRID	Ruler/grid point
OFF	Switches **Catch** off

Simply click on the **Catch** arrow in the bottom menu bar to list these options for selection.

When **Catch** is set to **All**, Creo Elements/Direct Drafting will catch using the following order of priority:

1. **VERTEX**
2. **INTERS**
3. **RULER/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.

Setting the Catch Range

To alter the catch range:

1. Pull down the **Setup** menu.
2. Click **Edit Environment** and select **System**.
3. In the **Cursor** section of **Environment**, specify the new catch range.
4. Click **OK**.

Measuring Distances

The **Measure** functions allow you to obtain values for arc lengths, distances, point coordinates, angles, and areas. These functions are useful if you need to enter 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. Pull down the **Tools** menu.
2. Click **Measure** and **Length**.
3. Click the existing line. The measured length of this line is displayed in the User Input Line.
4. Click **Line** in **Geometry**.
5. Click **Vertical** in **Line**.
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, simply display 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.

Measure in a loop

In older versions of Creo Elements/Direct Drafting, the MEASURE function ended after one execution. You can measure continuously instead, or keep the traditional behavior.

Use the MULTIPLE_MEASURE switch to measure in a loop.

MULTIPLE_MEASURE does not terminate other commands or functions. In the user input line, each new measurement overwrites the last. MULTIPLE_ MEASURE works with MEASURE_ANGLE, MEASURE_COORDINATE, MEASURE_DISTANCE, MEASURE_LENGTH, and MEASURE_RADIUS. MULTIPLE_MEASURE does not work with MEASURE_AREA or MEASURE_ TANG_ANG_AT_PNT.

Enter MULTIPLE_MEASURE ON in the user input line. When you're finished making measurements, type END to exit the loop.

Restore traditional measurement behavior with MULTIPLE_MEASURE OFF.

📋 Note

- The ruler Follow function, described later in this section, is not active during the Measure function.
- If any of the Measure functions are used within macros, the resultant measurement will be stored and can be obtained by using the `INQ` function.

Measuring Lengths

Click **Tools**, **Measure** and **Length** to measure the length of any element (line, circle, arc, fillet, chamfer or spline).

Measuring Distances (Horizontal and Vertical)

📋 Note

This option is dependent on the current catch mode.

---

1. Click **Tools**, **Measure** and **Horizontal** or **Vertical**.
2. Click two points in the drawing to identify the horizontal or vertical distance to be measured.

Displaying Point Coordinates

📋 Note

This option is dependent on the current catch mode.

To display the exact coordinates of any existing point:

1. Click **Tools**, **Measure** and **Point**.
2. Click the point. The exact coordinates of this point are shown in the User Input Line.

Measuring Angles

To measure the angle between two lines:

1. Click **Tools**, **Measure** and **Angle**.

2. Click a point on the first line.

3. 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 a Radius

To measure the radius of a curved element:

1. Click **Tools**, **Measure** and **Radius**.

2. Click the curved element whose radius you want to measure.

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.

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- 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. Click **Tools** and **Area Properties**. Creo Elements/Direct Drafting prompts you to enter a density (default 1) or to select a face.
2. Enter the density value if known, otherwise select the face(s).
3. Click **Confirm**. The area properties are listed in a text box.
4. Optionally, you can save the list of area properties in a file.
5. Close the text box to return to Creo Elements/Direct Drafting.

Example

Figure 13. I-Section Face on page 72 shows an I-section face.

Figure 13. I-Section Face
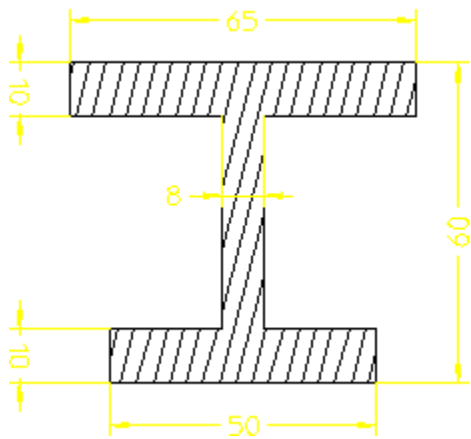
Figure 14. Resultant Diagram for I-Section Face on page 72 is a resultant diagram for the I-section face.

Figure 14. Resultant Diagram for I-Section Face
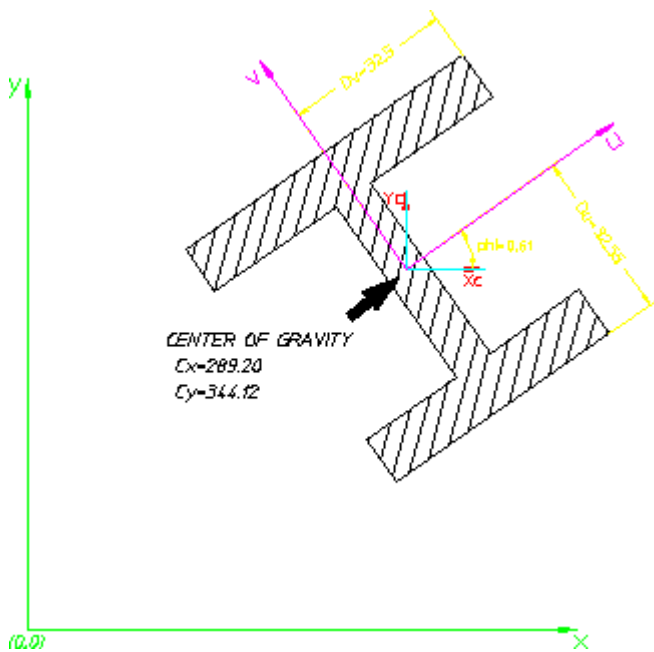
A listing of the Creo Elements/Direct Drafting output for this single face is given below:

```
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)
 Ixy = 146095161.0177031
MOMENTS OF INERTIA (ABOUT PRINCIPAL AXES)
 Jcu = 761433.673469478
 Jcv = 334727.5
MOMENTS OF INERTIA (ABOUT COORDINATE SYSTEM AXES)
 Jx = 174699726.755422
 Jy = 123421911.697053
PRODUCT OF MOMENT OF INERTIA (ABOUT COORDINATE SYSTEM AXES)
 Jxy = 146095161.017703
SECTION MODULI ABOUT PRINCIPAL AXES
 Zcu = 23392.00626959447
 Zcv = 10299.30769231674
DISTANCE FROM NEUTRAL AXIS u TO EXTREME FIBER
 Du = 32.55102040816436
DISTANCE FROM NEUTRAL AXIS v TO EXTREME FIBER
 Dv = 32.5
RADII OF GYRATION WITH RESPECT TO THE CENTER OF AREA
 Rcu = 22.7592199074477
 Rcv = 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 switch to scientific calculator mode, pull down the **View** menu in the top menu bar of the calculator and select **Scientific**.

Using the Ruler/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. Pull down the **Setup** menu.

2. Click **Grid** and **Settings** to display the **Grid Settings** dialog box.

3. Press the **Dot** or **Line** radio button below **Grid** to display a dot or line grid.

4. Press the **Ruler** radio button below **Ruler** to display a ruler.

Displaying the Origin

Press the **Origin** radio button in the **Grid Settings** dialog box. The coordinate origin is indicated in the chosen viewport by an axes symbol. If the ruler is active in the chosen viewport, it is switched off.

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. Pull down the **Setup** menu.

2. Click **Edit Environment** and **System**.

3. In the **Cursor** section of the **System** dialog box, tick the **Large** check box to switch to the large (full screen) cursor.

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

Using Follow On/Off

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** function to move the coordinate system origin with the origin of the ruler.

To make the origin follow the ruler:

1. Pull down the **Setup** menu.
2. Click **Grid** and **Follow**.

The ruler will now automatically move to each new point you create.

Moving the Ruler

To move the ruler:

1. Pull down the **Setup** menu.
2. Click **Grid** and **Move**.
3. Specify the new origin. The orientation of the ruler axes will remain the same.

Rotating the Ruler/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. Pull down the **Setup** menu.
2. Click **Grid ▸ Turn ▸ Absolute**.
3. Enter the angle required.

Through a Given Angle

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

1. Pull down the **Setup** menu.
2. Click **Grid ▸ Turn ▸ Relative**.
3. Enter the rotation angle required.

A positive value results in counterclockwise rotation.

Setting the Ruler/Grid Spacing

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

1. Display the **Grid Settings** dialog box.
2. Enter the required distance (for example: 0.5) into the **Spacing** field. 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 **View** functions if you want an enlarged view.

Drawing an Isometric Ruler/Grid

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

1. Pull down the **Setup** menu.

2. Click **Grid ▸ Isometric**.

3. Select the applicable option:

   - **Top**

   - **Front**

   - **Side**

Hiding the Ruler/Grid

To hide the ruler/grid:

1. Pull down the **Setup** menu.

2. Click **Grid** and **Hide**.

The **Hide** function switches the ruler and/or grid off. Alternatively, you can press the **None** radio button in the **Grid Settings** dialog box.

Placing the Ruler/Grid

To change the orientation of the axes:

1. Pull down the **Setup** menu.

2. Click **Grid ▸ Place**.

3. Specify the new origin and two more points that define the new axis orientation.

Figure 15. Ruler Axes on page 76 shows the ruler axes.

Figure 15. Ruler Axes
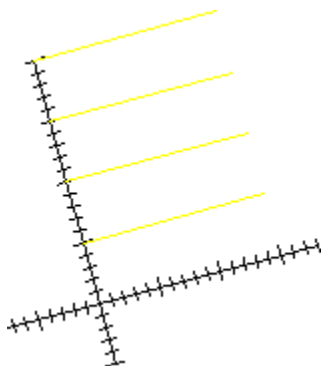
Selecting What is Displayed on Your Screen

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

You can selectively display:

- Construction geometry
- Geometry
- Dimensions
- Hatch
- Text
- Vertex
- Info
- Layers
- URLs

📝 **Note**

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

Showing All Features

Click **Show All** in the **Show** dialog box to display all features of your drawing. It is useful to start with everything displayed, then switch off what is not required.

Showing Points

Tick the **Vertex** check box in **SHOW** to display all existing points on the drawing. 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

The **Dimensions** check box in **Show** enables you to turn dimensions on and off as a group. Any dimensions created after the dimensions have been turned off will not be displayed on the screen until the dimensions are turned on again.

Changing the Layer Color

To change the color of a specified layer.

1. Click the **Layer** tab in **SHOW**.
2. Select the layer number.
3. Select a color from the **Color** list.

Displaying Infos

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

Viewing A Drawing

This section shows 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.

Click **View** to display the **View** menu. The **View** functions operate on either the current viewport (default) or on a viewport you selected.

Selecting a Viewport

You can select the viewport in which the **View** functions operate.

Current Viewport

If you want the **View** functions to operate in the current viewport:

1. Pull down the **Setup** menu.
2. Click **Viewport** and **Current**.
3. Click in the viewport or enter the viewport number.

Creo Elements/Direct Drafting will use the current viewport for any **View** functions.

Select Viewport

1. Pull down the **Setup** menu.
2. Click **Viewport** and **Select**.
3. Click in the viewport to select it.

Using View Functions

This section shows how to control the area of the drawing shown in the current viewport using the **View** functions.

Redrawing

Click **Redraw** in **View** to redraw the drawing in the chosen viewport. This is sometimes necessary if you have deleted several elements and traces can still be seen on the screen. The contents of other viewports will not be redrawn.

Fitting a Drawing

Click **Fit** in **View** to fit the current drawing into the chosen viewport.

Panning Across a Drawing

Panning is a very useful function enabling you to move across the drawing to see those areas not currently displayed.

Figure 16. Panning Across a Drawing on page 79 illustrates the use of panning.

Figure 16. Panning Across a Drawing
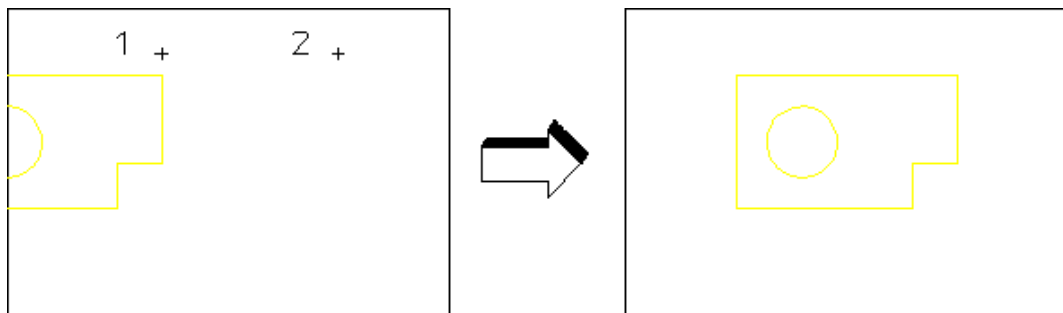
To pan in any direction:

1. Click **Pan** in **View**.

2. Click a point in your drawing.

3. Specify the new screen position for this point.

Alternatively, press the [Ctrl] key, press the left mouse button, and drag the cursor to pan to the new position.

Creating a New View in a Window

To create a new window:

1. Click **New** in **View**.

2. Specify the bottom left and top right corners of the area you want to enlarge.

Restoring the Last View

Click **Last** in **View** to return to the previous view.

Zooming In or Out

A window is always zoomed about its center point.

To zoom your window:

1. Click **Zoom** in **View**.

2. Enter the zoom factor required.

   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).

3. Press [Enter].

Alternatively, press the [Ctrl] key and press the right mouse button. Move the mouse towards the screen to zoom in, move the mouse away from the screen to zoom out.

Storing and Recalling Windows

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

Click **Store** in **View** to store the current window coordinates.

Click **Recall** in **View** to recall the window that was stored last.

Creating a New Center Point for Your Drawing

Click **Center** in **View** and specify the new center point for the display. Your drawing is then repositioned so that this point is in the center of the chosen viewport.

📝 **Note**

This option uses the current catch mode to identify the specified point.

Redrawing

1. Click **Redraw** in **View** to rapidly redraw your drawing in the current or selected viewport.

Fitting a Drawing

1. Click **Fit** in **View** to rapidly fit your drawing in the current or selected viewport.

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. Using Viewport 2 as a Reference on page 82 and Figure 18. Viewing Piston Assembly Detail in Viewport 1 on page 83.

Figure 17. Using Viewport 2 as a Reference

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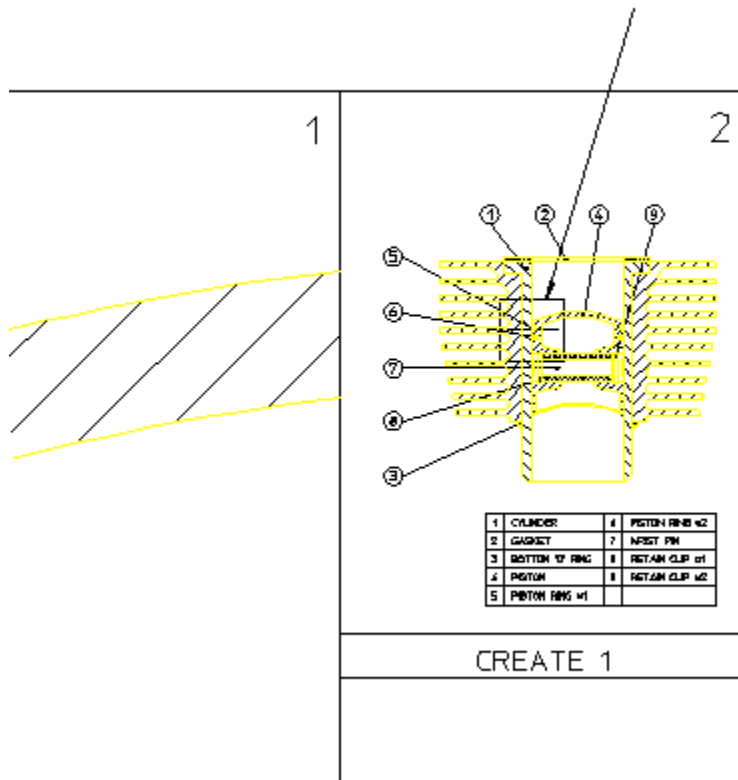
Figure 18. Viewing Piston Assembly Detail in Viewport 1
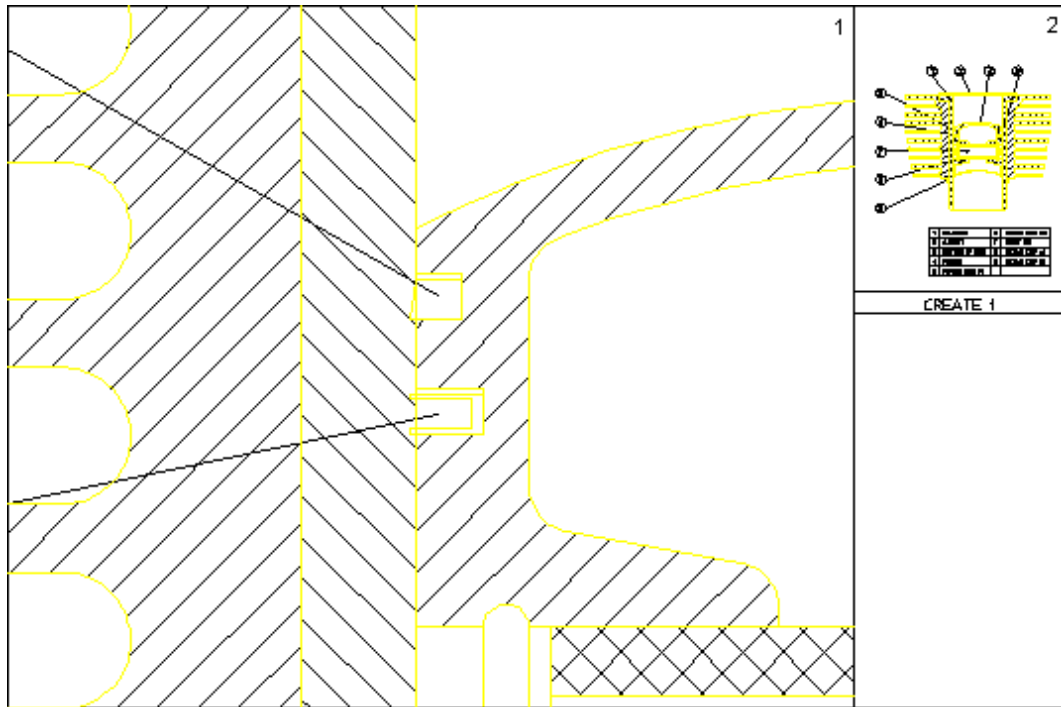
Creating a Viewport

To create a new viewport:

1. Pull down the **Setup** menu.

2. Click **Viewport** and **Create**.

3. Specify a point for the top left corner and another point for the bottom right corner of the new viewport.

The newly created 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 will not be 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. Pull down the **Setup** menu.
2. Click **Viewport** and **Current**.
3. Click the viewport required. Alternatively, enter the viewport number and press [Enter].

Changing the Size of a Viewport

To change the size of a viewport:

1. Pull down the **Setup** menu.
2. Click **Viewport** and **Change**.
3. Click the viewport.
4. Indicate the new corner points of the viewport.

Moving a Viewport

To move a viewport:

1. Pull down the **Setup** menu.
2. Click **Viewport** and **Move**.
3. Click the viewport.
4. Drag the cursor to the new position of the bottom left corner of the viewport.

The contents of the viewport follows the viewport.

Redrawing a Viewport

To redraw a viewport:

1. Pull down the **Setup** menu.
2. Click **Viewport** and **Redraw**.
3. Click the viewport.

The contents of the viewport will be redrawn.

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.

1. Pull down the **Setup** menu.
2. Click **Viewport** and **Large**.
3. 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**.

Deleting a Viewport

To delete a viewport:

1. Pull down the **Setup** menu.
2. Click **Viewport** and **Delete**.
3. Click the viewport. Alternatively, enter the viewport number and press [Enter].

Deleting All Viewports

To delete all viewports:

1. Pull down the **Setup** menu.
2. Click **Viewport** and **Delete**.
3. Click **Select**, **All**, **Confirm**, **OK**.

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

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.

Defining Viewport Settings

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

To define the viewport settings:

1. Click **Setup ▶ Viewport ▶ Settings**.

   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 59 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. 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.

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. Joining Two Widely Spaced Points Using Viewports on page 87.

Figure 19. Joining Two Widely Spaced Points Using Viewports

To draw a line between points 1 and 2 in :

1. Create the three viewports (V1, V2 and V3).

2. Make V1 the active viewport.

3. Window into point 1 in V1.

4. Make V2 active.

5. Window into point 2 in V2.

6. Click **Line** in **Geometry**.

7. Click **2 Pts** in **Line**.

8. Specify point 1 in V1 and point 2 in V2.

Creo Elements/Direct Drafting will now draw 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 via the **Geometry Settings** dialog box.

Text Properties

Text properties are:

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

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

Hatch Properties

Hatch properties are:

- Pattern
- Angle
- Distance
- Reference Point

These can be predefined via 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. Click **Tools ▸ Properties ▸ Get**.
2. Select a block of text.

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

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

1. Click **Tools ▸ Properties ▸ Get**.
2. Select the text whose properties are to be retrieved.
3. Click **Tools ▸ Properties ▸ Put**.
4. Select the text whose properties are to be changed.

Setting the Auto Save Time for Drawings

In addition to the **File** functionality described in , 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 **Setup ▸ Edit Environment ▸ System**.
2. In **System**, click on the **Save** tab.
3. Tick the **Auto Save Drawing** checkbox.
4. Specify a time interval.
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 **Setup ▶ Edit Environment ▶ System**.

2. In **System**, click on the **UI** tab.

3. In the **Cascade Timeout** field, specify the time delay (in milliseconds).

4. Click **OK** to save these settings.

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.

1. Click **Setup ▶ Edit Environment ▶ System**.

2. In **System**, click on the **Redraw** tab.

3. Select one or any of the following settings:

    • In the **Parts as Box** section, you can specify a size threshold (in pixels) below which parts are only shown as boxes. This box representation helps to unclutter your drawing display. Optionally, select a color from the **Box Color** menu.

    • In the **Draw Dynamic** section, check any items that you want to include in the redraw. Uncheck any items you want to exclude from the redraw to enhance performance.

4. Click **OK** to save these settings.

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 **Setup ▶ Edit Environment ▶ System**.

2. In **System**, click on the **Save** tab.

3. Tick the settings you want to save.

4. Click **OK** to save these settings.

5

Using the Creo Elements/Direct Drafting Copilot Functionality

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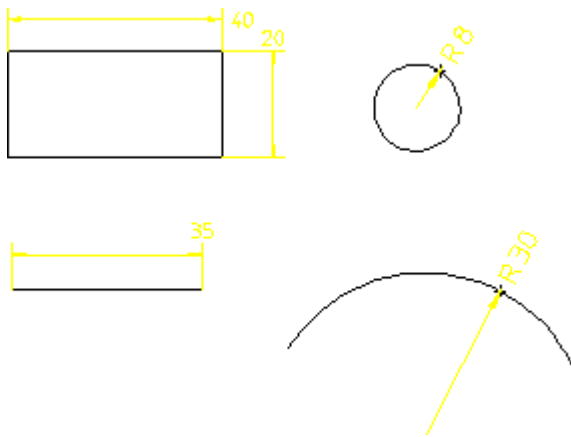
The snap marker (magenta box) over the cursor indicates that you are in copilot mode and about to click a start-point.

Enabling/Disabling the Copilot Functionality

To enable or disable the copilot functionality:

1. Pull down the **Setup** menu.

2. Click **Edit Environment** and **System**.

3. In the **System** dialog box, click on the **Copilot** tab.

4. Tick the **Enable Copilot** check box to switch on the copilot functionality. To switch off the copilot functionality, click on the ticked check box.

Copilot Settings

To define the settings for the copilot functionality:

1. Pull down the **Setup** menu.

2. Click **Edit Environment** and **System**.

3. In the **System** dialog box, click on the **Copilot** tab.

4. Define the following settings for the copilot functionality:

   • Snap settings

   • Catch range settings

   • Text settings

Snap Settings

Distance	Sets the distance that the cursor snaps to.
Angle	Sets the angle that the cursor snaps to.

For example, if the snap distance is set to 1 mm and the snap angle to 15°, 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

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

Text Settings

Color	Sets the color of the text in the text box.
Long Prefixes	Displays a full description of the circular element in the text box. Use Long Prefixes on = Radius. Use Long Prefixes off = R

To define other settings,

1. In the **System** dialog box, click the **Snapping** tab.

2. Define the following settings for snapping functionality:

   • Snap points

   • Snap marker

- Snap line
- Constraint signs

Snap Points Settings

Count	Sets the maximum number of points simultaneously present in the drawing.
Delay	Sets the delay (in seconds) for how long a cursor is caught on an element before a snap point for a snap line appears.
Color	Sets the color of the last visited snap point.

Snap marker Settings

Color	Sets the color of the snap marker.

Snap Line Settings

Color	Sets the color of the snap line.
Linetype	Sets the line type of the snap line.

Constraint Signs Settings

Enable	Displays a sign when the element you are creating is horizontal, vertical, tangent, or perpendicular to another element.
Color	Sets the color of 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 Distance and Snap Angle (**Setup ▸ Edit Environment ▸ System ▸ CoPilot** tab).
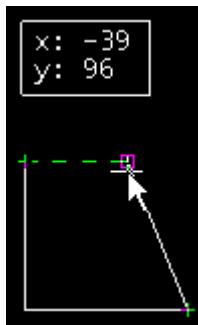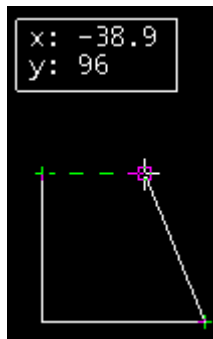
Increment = 1 mm   Increment = 0.1 mm

Size indication

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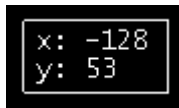
Snap lines

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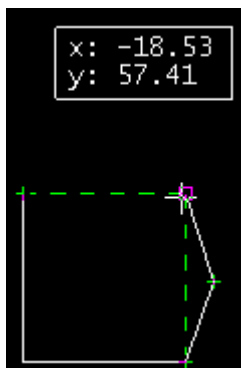
Snap line base points

When you hold the cursor at a vertex for .5 seconds (the default setting), a small plus sign appears. This is the snap point for snap lines.
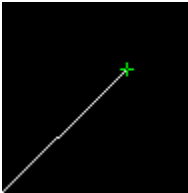
On geometry elements, you can have snap points at the start point, the end point, the midpoint (of lines), and the intersection between two elements.
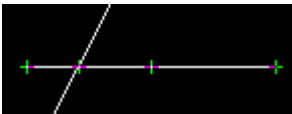
There is a limit to the number of snap points that can exist at one time; when it is reached, the snap point that hasn't been visited for the longest time disappears. Change this number in the **System** menu: **Setup ▶ Edit Environment ▶ System ▶ Snapping** tab.

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, or the intersection of a snap line and a geometry element.
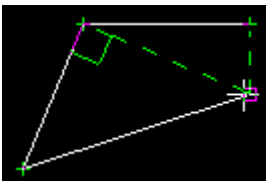
Snap lines on text

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.
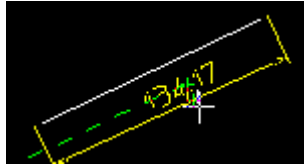
To align a new dimension with an existing dimension, the text and lines of the two dimensions must be parallel.

Snap lines on reference lines

On reference lines, the snap points appear on the vertices. You can snap the cursor to the refline 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 into an engineering design without the need for unnecessary construction geometry.

Figure 21. Design Sketch
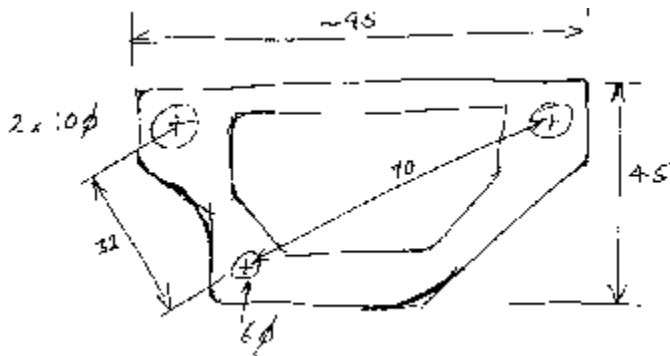
Figure 22. Design Reality
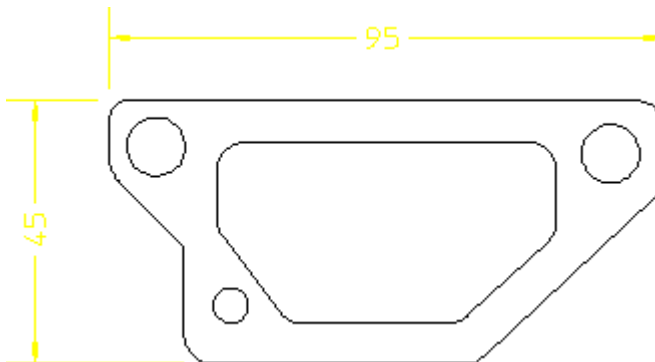
The following example illustrates how to construct the plate with the minimum of construction geometry. As you draw, you will notice that line colors may change as you intersect with existing geometry and the status line gives feedback on your current activity. During certain actions, Creo Elements/Direct Drafting displays negative values. Ignore these for this example.

1. Click **Line** in **Geometry**.

2. Click **Horizontal** and draw a horizontal line of length 50 mm (line 1 in Figure 23. Slotted Plate - 1 on page 99)

3. Click **Vertical** and draw a vertical line of length 20 mm (line 2)

4. Click **Angle & Length** and click point **A**

5. Move the cursor in an arc and specify an angle of 135°

6. Drag the cursor and specify a length of 18 mm

7. Click the parallel construction line icon and click line **1**

8. Drag the cursor vertically and click a point at **45**

9. Click **Vertical** and draw line 4 to meet the construction line

10. Click **Horizontal** and draw a horizontal line of length 95 mm

11. Click **Vertical** and draw a vertical line of length 15 (line 6)

12. Click **2 Pts** 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 **Circle** in **Geometry**.

2. Click **Center & Point or Radius** (Figure 24. Slotted Plate - 2 on page 99)

3. Click **Offset**

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. Draw construction circles at 32 and 70 with H1 as the center (center and radius in **Construct**)

8. Click **Center & Point or Radius** and construct holes H2 and H3 having radius 5 (precise position is not important)

9. Delete the circular construction geometry.

Figure 23. Slotted Plate - 1
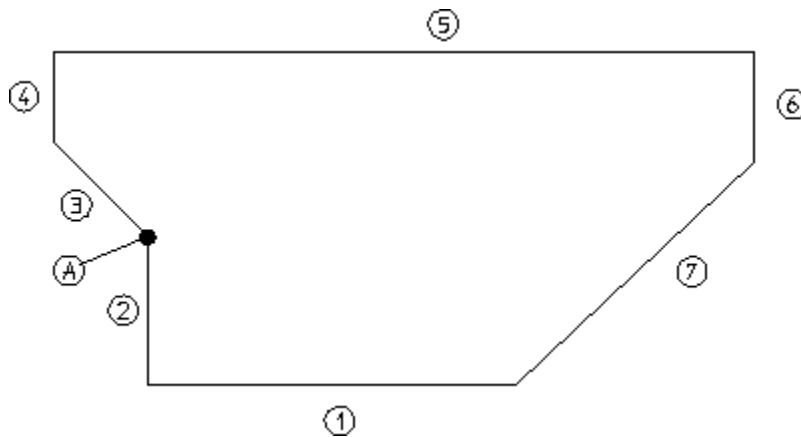
Figure 24. Slotted Plate - 2
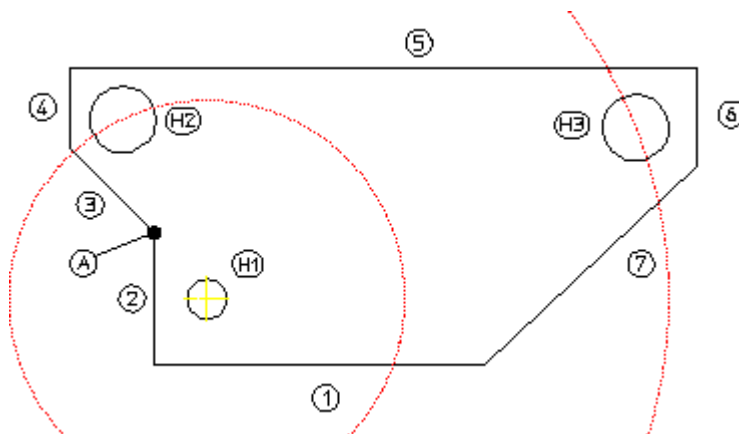
The slot can be drawn in freehand mode using the **Polygon** command. The only criteria is that the inside edge is parallel to line 7.

1. Click the parallel construction line icon in **Construct** and draw the parallel lines at a distance of 7 from lines 1, 5, and 7.

2. Click **Polygon** in **Line** and draw the slot starting at point B moving anticlockwise.

Finally, to cleanup the drawing, delete the construction geometry and add some fillets to the corners (**Fillet** in **Geometry**, size 5 mm).

Figure 25. Slotted Plate - 3

6

Drawing Methods

This chapter shows how to produce a drawing using construction geometry and "real" geometry.

Creating Construction Geometry

Construction geometry is used for positioning and as an aid to drawing.

This section shows how to draw using construction lines and circles, just as you would normally do when working on a drawing board.

Construction lines differ from real lines in that they are considered to be of infinite length. Only full construction circles can be drawn.

When creating 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. When you have typed in the coordinates, press **[Enter]**.

Creating Construction Lines

The following commands allow you to draw construction lines.

Horizontal Construction Line

Enter the y coordinate of the line required, or click a point through which the line must pass. To create several horizontal construction lines, simply enter the appropriate y coordinates separated by spaces.

Vertical Construction Line

Enter the x coordinate of the line required, or click an existing point through which the line must pass. To create several vertical construction lines, simply enter the appropriate x coordinates separated by spaces.

Construction Line by Two Points

Click this option, then click two points for the construction line required or enter the coordinates of two points.

Parallel Construction Line

You can create construction lines parallel to any existing line in the following ways:

- Click any existing line. Specify a point for the parallel construction line to pass through. You can draw multiple parallel construction lines by specifying further points for the lines to pass through.

- Enter the required separation of the line. Then click any existing line. Finally, click the side on which the parallel construction line should be drawn. Multiple lines can be drawn at the previously set separation by clicking the original lines and the desired sides.

- A set of parallel lines can be drawn from an original line. For example, if you enter line separations in the form: 10 20 30 40, four parallel lines will be drawn at the given distances from the indicated line.

Perpendicular Construction Line

You can create a construction line perpendicular to any existing line. Click the option, click the existing line, and then specify a point through which the perpendicular construction line must pass.

Bisecting Construction Line

If you do not want the line concerned divided into two equal parts, enter a value for the ratio of the division. Then specify the two end points of the line. A construction line will be drawn perpendicular to the line at the appropriate position.

Construction Line Bisecting an Angle

You have the option of entering a value for the ratio of the division, otherwise the angle will be bisected. Specify the apex, and then specify two points (usually on existing lines) to define the angle.

Construction Line at an Angle

When you select this option, Creo Elements/Direct Drafting will prompt you to specify a point through which the construction line must pass. Next enter the angle of the line with respect to the x axis. Note that a positive input will result in a line drawn counterclockwise from the x axis. The angle units will be those currently defined for your Creo Elements/ Direct Drafting environment.

Construction Line Tangent to Two Circles

You can create a construction line as a tangent to two circles or splines. Click this option and click the approximate point of tangency on each of the two circles or splines. The construction line created will be the one with points of tangency closest to those you indicated.

Construction Line Tangent from Point to Circle

This option enables you to create a construction line tangent to a circle or a spline and passing through a given point or at a set angle.

Click the approximate point of tangency on the circle or spline and then specify a point through which the construction line must pass. Or, click the approximate point of tangency, then enter the desired angle of the construction line. Note that the angle is measured in a counterclockwise arc from the horizontal.

Drawing Construction Circles

Eight commands allow you to draw construction circles.

Construction Circle by Three Points

You can create a construction circle by specifying three points on the circumference. Click this option and specify two circumference points. The construction circle created will vary with the movement of your cursor until you specify the third point.

Construction Circle by Center and Circumference Point

Specify the center of the construction circle and either specify a point on the circumference, or enter the radius required. If multiple concentric circles are required, specify multiple peripheral points or enter the desired radii.

Construction Circle by Diameter

Specify two points as the two ends of the diameter for the required circle.

Concentric Construction Circle

To create concentric construction circles, click this option and click the existing circle. Enter the radial offset of the new construction circle from the existing one, that is the difference between the two radii. To create several concentric construction circles, simply type in the appropriate radial offsets separated by spaces. Alternatively you can click existing points through which the construction circles must pass.

Construction Circle by Center and Element

Click the approximate point of tangency on the existing element. Next specify the center of the construction circle to be drawn.

Construction Circle by Radius, Point (or Two Points) and Element

Click the approximate point of tangency of the construction circle on the existing element (which must not be a spline). Next specify a point on the circumference, then either specify another point or enter the required radius.

Construction Circle by Radius or Point and Two Elements

This option draws a construction circle of given radius tangential to two existing elements. Click the approximate points of tangency (either lines or circles but no splines). Next specify a peripheral point on the construction circle or enter the desired radius. Creo Elements/Direct Drafting calculates the best solution, or informs you that no solution is possible.

Construction Circle by Three Elements

Click the approximate point of tangency of the construction circle on each of the three existing elements (none of which must be a spline).

📝 Note

If the third tangency point you click for a construction circle is located such that no circle is tangent to all three points, the first two points will move along the projection (notional extension) of the associated elements.

Drawing a Lever Using Construction Geometry

Use the following tutorial example to familiarize yourself with the various techniques used when producing a drawing with the aid of construction geometry.

Figure 26. Lever on page 107 shows the finished drawing of the lever.

Figure 26. Lever
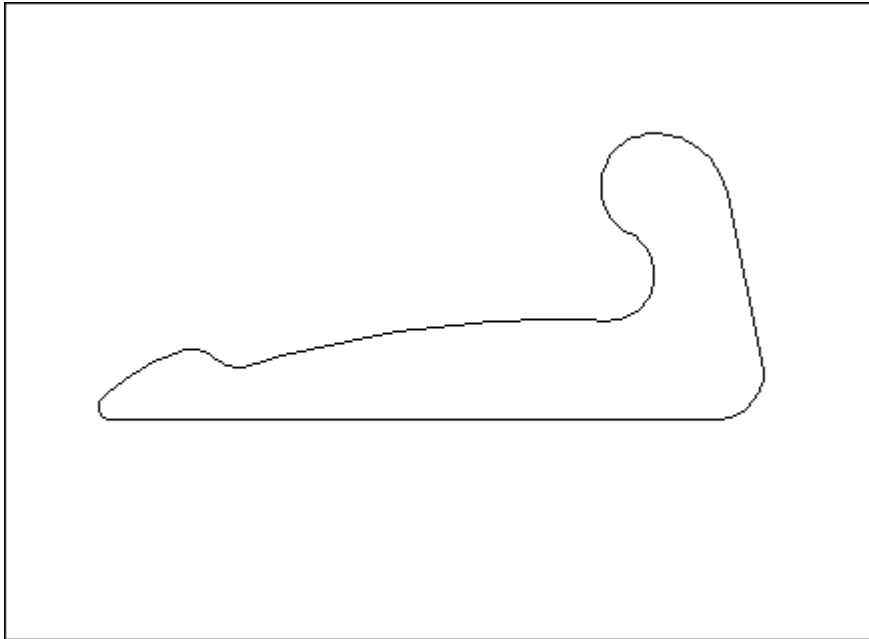
First draw horizontal and vertical construction lines. See Figure 27. Horizontal and Vertical Construction Lines on page 107.

Figure 27. Horizontal and Vertical Construction Lines
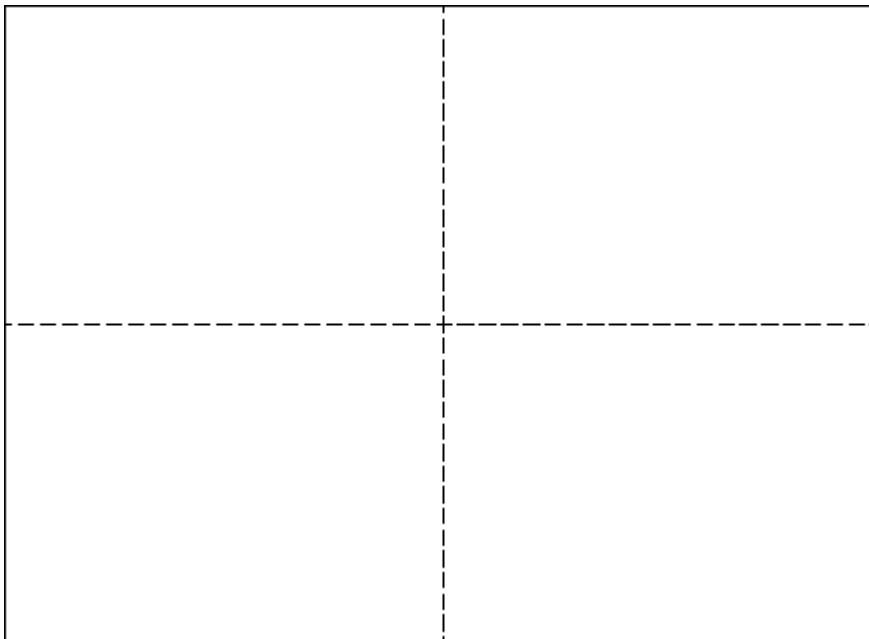
1. Pull down the **Setup** menu and click **Show**.

2. Make sure that the **Construction Geometry** and **Geometry** options are switched on (ticked) in the **Show** option box.

3. Make sure that the **Vertex** option is switched off (not ticked) in the **Show** option box.

4. Select **All** in the **Catch** menu at the bottom of the Creo Elements/Direct Drafting window.

5. Click the following icon in **Construct** to draw a horizontal construction line:

6. Enter **0,0**.

7. Click the following icon in **Construct** to draw a vertical construction line:

8. Enter **0,0**.

9. Click **Fit** in **View**.

10. Click **End**.

Now draw some parallel horizontal construction lines. See .

Figure 28. Parallel Construction Lines
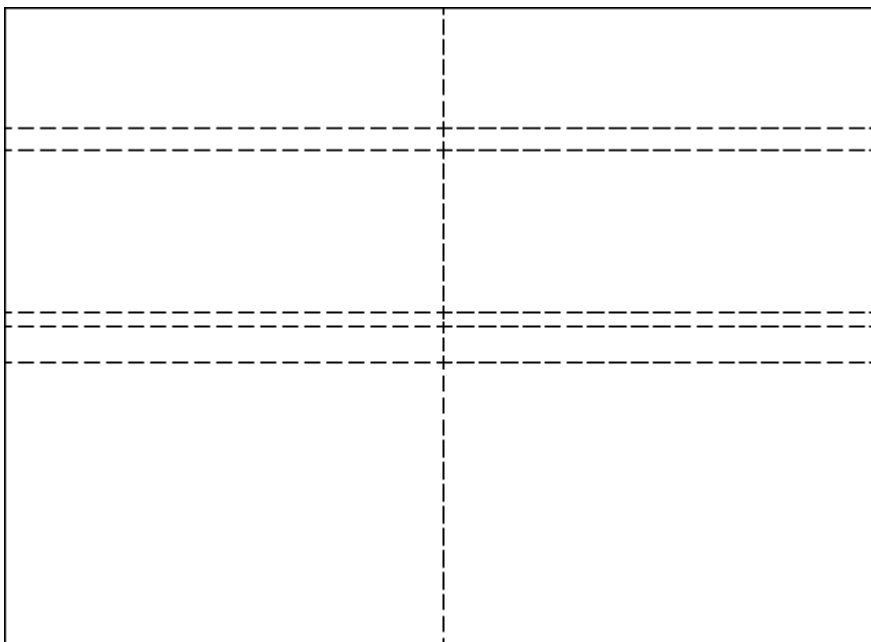
1. Click the following icon in **Construct** to draw parallel construction lines:

       Creo Elements/Direct Drafting User's Guide: Windows User Interface

2. Enter **-8 3 40 45**.
3. Click the horizontal construction line and then click a point above it.
4. Click **Fit** in **View**.

Draw some parallel vertical construction lines. See Figure 29. Parallel Vertical Lines on page 109.

Figure 29. Parallel Vertical Lines
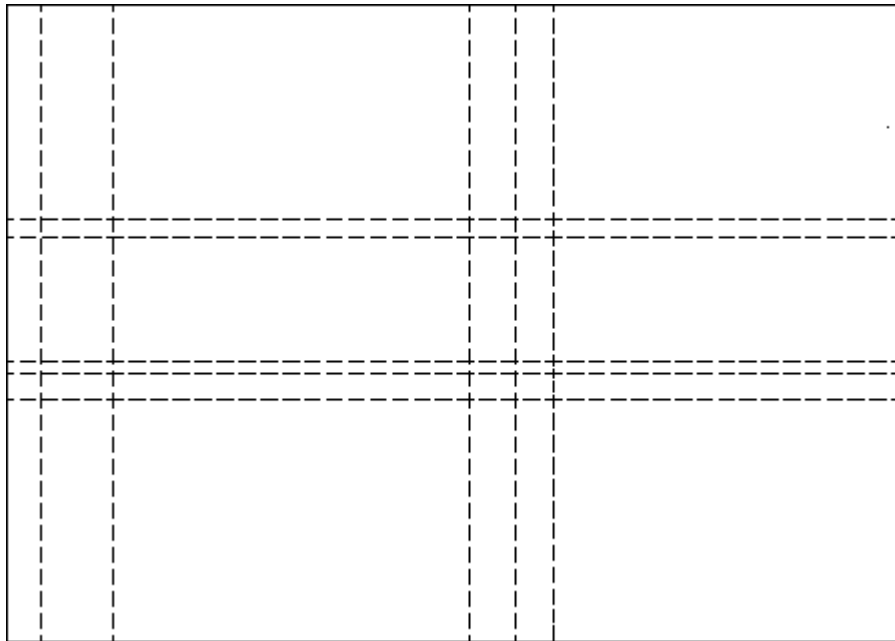
1. Enter **11 25 130 151** (the option for parallel construction lines is still active).
2. Click the vertical construction line and click a point to the left of it.
3. Click **Fit** in **View**.

Create three viewports in the drawing area. See Figure 30. Three Viewports on page 110.

Figure 30. Three Viewports
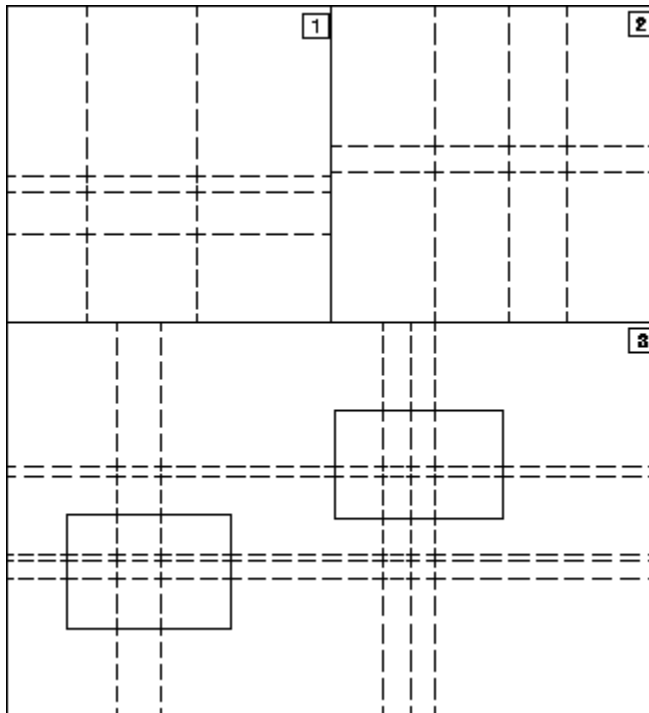
1. Pull down the **Setup** menu.
2. Click **Viewport** and then **Delete**.
3. Click **All**.
4. Pull down the **Setup** menu.
5. Click **Viewport** and then **Create**.
6. Create viewport 1 as shown above by clicking two corner points of the viewport.
7. Repeat the previous three steps to create viewports 2 and 3.

   Viewports 1 and 2 give you a closer look at certain areas of your drawing where you will do more complex work. Viewport 3 displays the entire drawing.

8. Pull down the **Setup** menu.
9. Click **Viewport** and then **Current**.
10. Click viewport 1.
11. Click **New** in **View** and click two points in viewport 3 as defined by the left hand rectangle.
12. Pull down the **Setup** menu.
13. Click **Viewport** and then **Current**.

14. Click viewport 2.

15. Click **New** in **View** and click two points in viewport 3 as defined by the right hand rectangle.

Now create some construction circles. See Figure 31. Construction Circles on page 111.

Figure 31. Construction Circles
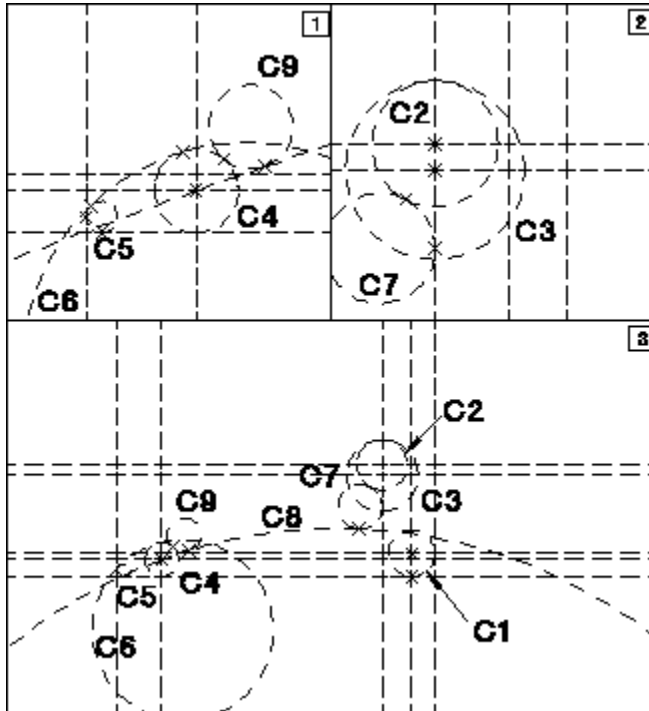
1. Click **ALL** in the **Catch** menu.

2. C1 — Click the following icon in **Construct**:

    Then click the center point and lower circumference point (in viewport 3).

3. C2 — Click the following icon in **Construct**:

    Then click the center point and enter **12** (in viewport 2).

4. C3 — Click the following icon in **Construct**:

    Then click the center point and enter **17** (in viewport 2).

5. C4 — Click the following icon in **Construct**:

   Then click the center point and enter **8** (in viewport 1).
6. C5 — Click the following icon in **Construct**:

   Then click the two tangent points and enter the radius **3** (in viewport 1).
7. C6 — Click the two tangent points and enter the radius **43** (in viewport 1).
8. C7 — Click the two tangent points and enter the radius **10.5** (in viewport 2).
9. C8 – Click the following icon in **Construct**:

   Then click the tangent point to C7 and the center point of C4. Enter the radius **246** (in viewport 3).
10. C9 — Click the following icon in **Construct**:

   Then click the two tangent points and enter the radius **8** (in viewport 3).

Draw a construction line tangent to C1 and C3. See Figure 32. Construction Line Tangent on page 113.

Figure 32. Construction Line Tangent
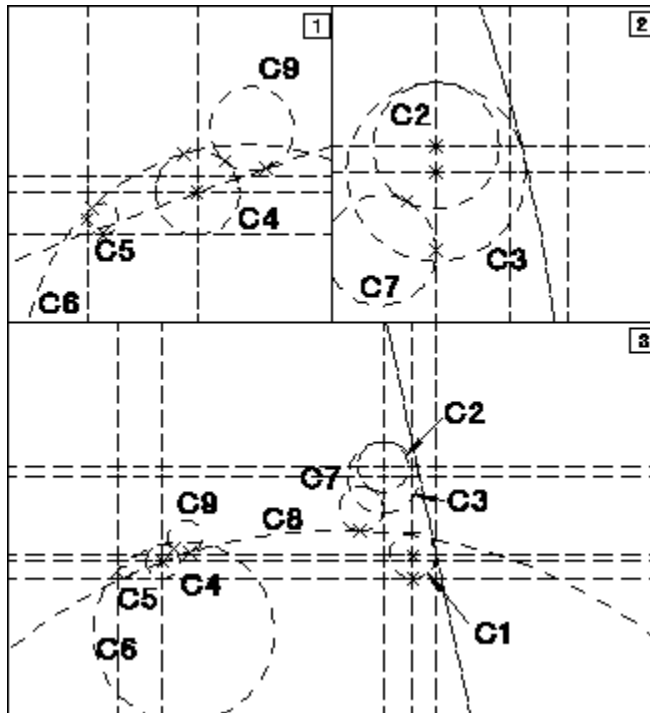
1. Click the following icon in **Construct**:

2. Click the tangent points on C1 and C3 (in viewport 3).

Click **Overdraw** in **Geometry** to draw the lever itself. See Figure 33. Drawing the Lever on page 114.

Figure 33. Drawing the Lever
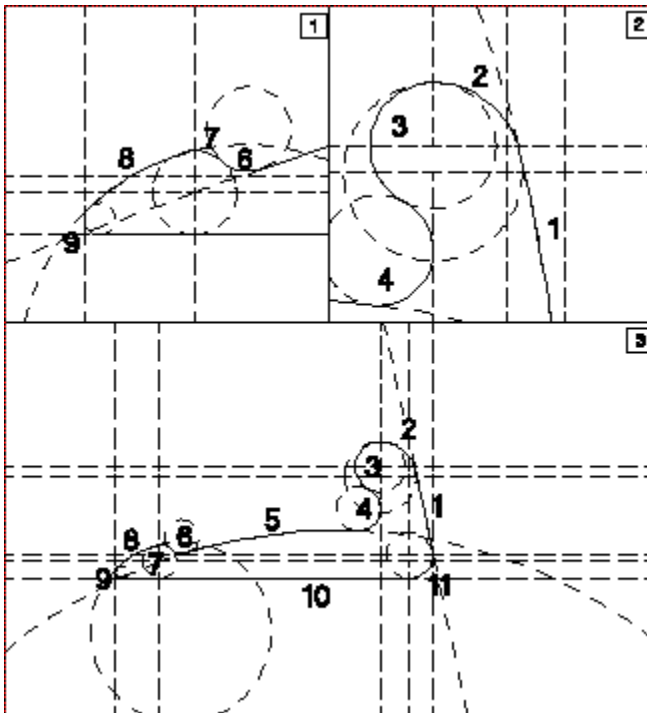
1. Click **Overdraw** in **Geometry**.
2. Click point **11/1** as the start point.
3. Click a point on element 1 and click point **1/2**.
4. Click a point on element 2 and click point **2/3**.
5. Click a point on element 3 and click point **3/4**.
6. Click a point on element 4 and click point **4/5**.
7. 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 Overdraw in Geometry and window in on the area by clicking New in View.

---

8. Continue with this procedure for elements 6, 7, 8, 9, 10, and 11.

Creating Geometry

To create geometry, click **Geometry**. The **Geometry** menu contains the following commands:

- **Point**
- **Line**
- **Circle**
- **Arc**
- **Fillet**
- **Chamfer**
- **Equidist**
- **Overdraw**
- **CenterLine**
- **SymLine**
- **Ellipse**
- **Hyperbola**
- **Parabola**
- **Interpolation Spline**
- **Control Spline**
- **Title Block**
- **Change**

Creating Points

Points can be created anywhere on your drawing. Simply click **Point** in the **Geometry** menu and then click the position in a viewport, or entering the required coordinates.

If you have difficulty deleting points, try enclosing them in a box for deletion.

Creating Lines

The **Line** menu provides the following commands:

- **Offset**
- **2 Points**
- **Horizontal**
- **Vertical**

- **Parallel**

- **Perpendicular**

- **Angle & Length**

- **Tangential Arc & Point**

- **Tangential Arc & Arc**

- **Rectangle**

- **Polygon**

- **Poly Element**

Specifying an Offset for Linear Elements

The **Offset** command enables you to specify an offset for the points defining a new line, arc, rectangle, polygon or polyline. This offset is defined by the distance from an existing element and the distance from another existing element.

To define an offset for a linear element:

1. Click **Line** in **Geometry**.

2. Enter the following into the user input line:

   OFFSET

3. Click on a linear element type (such as **Horizontal**) in **Line**.

4. Select the first existing element. Specify the required distance from this element. Click a point on one side of this element.

5. Select the second existing element. Specify the required distance from this element. Click a point on one side of this element.

6. The offset point thus defines serves as the current definition point (such as the begin point or end point) of the linear element you are creating. Complete this linear element. Note that you can use the **Offset** command multiple times when creating the new linear element.

Alternatively:

1. Select the first element.

2. Select the second element.

3. Specify the offset point between the two elements.

Line by Two Points

To draw a line by two points:

1. Click **Line** in **Geometry**.
2. Click **2 Points** in **Line**.
3. Specify the start and end points of the line.

Horizontal Line

This option is similar to line by two points but only draws a horizontal line.

To draw a horizontal line:

1. Click **Line** in **Geometry**.
2. Click **Horizontal** in **Line**.
3. Specify the start point of the line.
4. Specify the end point of the line, or enter the length of the line.

   If you specify an end point, the line is drawn through that point.

Vertical Line

This option is similar to line by two points but only draws a vertical line.

To draw a vertical line:

1. Click **Line** in **Geometry**.
2. Click **Vertical** in **Line**.
3. Specify the start point of the line.
4. Specify the end point of the line, or enter the length of the line.

   If you specify an end point, the line is drawn through that point.

Parallel Line

To draw a line parallel to an existing line:

1. Click **Line** in **Geometry**.
2. Click **Parallel** in **Line**.
3. Click the existing line.
4. Specify a point on the new line.

   You can use the ruler to ensure that the line is drawn at a given distance (refer to Using the Ruler/Grid on page 74).

Alternatively:

1. Click **Parallel**.
2. Enter the distance required.
3. Click the existing line.
4. Specify the side on which the parallel line should be drawn.

Perpendicular Line

To draw a perpendicular line:

1. Click **Line** in **Geometry**.
2. Click **Perpend** in **Line**.
3. Click the existing line.
4. Specify the point from which the perpendicular line should be dropped.

Alternatively:

1. Click **Perpend**.
2. Click the existing line.
3. Specify a point on the existing line.
4. Enter the length of the perpendicular line. If the perpendicular line appears on the wrong side of the line, repeat the operation using the opposite sign for the entered value.

Tangent from Point to Arc or Circle

This option enables you to draw a tangent to a circular element from any given point.

1. Click **Line** in **Geometry**.
2. Click **Tangential Arc & Point** in **Line**.
3. Specify the approximate point of tangency on the arc or circle.
4. Specify the point from which the tangent should be drawn.

   The system will inform you if such a tangent cannot be created.

Tangent to Two Arcs or Circles

To draw a tangent from one circular element to another:

1. Click **Line** in **Geometry**.
2. Click **Tangential Arc & Arc** in **Line**.
3. Specify the approximate point of tangency on each of the two arcs or circles.

The tangent drawn will be the one with points of tangency closest to those indicated.

Line with Given Length and Angle

To draw a line with a given length and angle:

1. Click **Line** in **Geometry**.
2. Click **Angle & Length** in **Line**.
3. Specify the start point of the line.
4. Enter the angle of the line with respect to the x-axis.
5. Enter the length of the line.

   The line will be drawn accordingly.

Rectangle

Rectangles can be created as whole entities, but the system treats them as four individual line elements. This means the sides of a rectangle can be deleted or manipulated individually.

To draw a rectangle:

1. Click **Line** in **Geometry**.
2. Click **Rectangle** in **Line**.
3. Specify two diagonal corner points.

   The rectangle size will vary with the movement of the cursor until the second point is defined.

Polygon

To draw a polygon:

1. Click **Line** in **Geometry**.
2. Click **Polygon** in **Line**.
3. Specify the start point, and then a sequence of end points.

Note that the result does not have to be a closed polygon. The lines are treated as individual elements and must therefore be deleted individually or enclosed in a box for deletion.

Once you have closed a polygon, you can continue by indicating the first point of a new polygon.

Creating a Polyline

Polylines can be created with **Poly Element**.

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 with **Polygon** in **Line**. 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 34. Modifying Polyline and Non-Polyline Elements
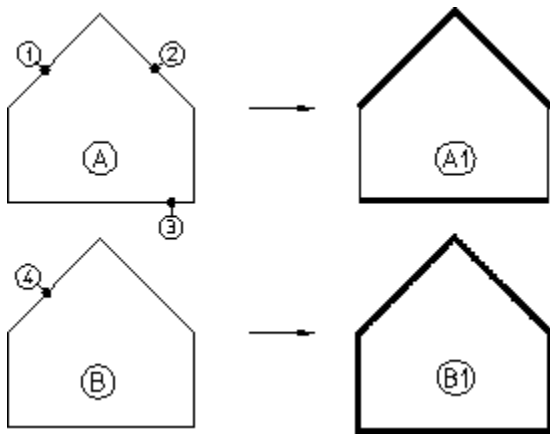
The following procedure describes how to create a polyline from a polygon and then modify the polyline by changing its pensize and shape.

To create a polyline:

1. Create a polygon.
2. Click **Poly Element** in **Line**.
3. Click a vertex of the existing polygon.
4. Click a polygon element that ends at the vertex.
5. Click adjacent elements or click the last element.
6. Click **End** when the desired shape is defined.

The polygon is now a polyline. To test it, change the pensize:

1. Click **Change** in **Geometry**.
2. Click **Pen Width** in **Change**.

3. Enter a value (for example, **0.5**).

4. Select one element of the polyline.

   Note that the complete shape changes–not just the selected element.

Similarly, you can modify the shape of the polyline:

1. Switch off **Keep** in **Modify**.

2. Click **Stretch** in **Modify**.

3. Click a **Stretch** option (such as **Horizontal**).

4. Click a vertex (or enclose it in a box).

5. Specify the new position of the vertex.

Smashing a Polyline

Use the **Smash** option to smash the polyline into individual elements.

Drawing Circles

The commands used for drawing circles are:

- **Offset**
- **Center & Point or Radius**
- **3 Points**
- **Tangential**
- **Concentric**
- **Diameter**

Specifying an Offset for Circles

The **Offset** command enables you to specify an offset for a new circle. This offset is defined by the distance from two existing linear elements.

To define an offset for a circle:

1. Click **Circle** in **Geometry**.

2. Click on a circle option (such as **3 Points**) in **Circle**.

3. Click **Offset** in **Circle**.

4. Select the first linear element. Specify the required distance from this element. Click a point on one side of this element.

5. Select the second linear element. Specify the required distance from this element. Click a point on one side of this element.

6. The offset point thus defines a point of the circle you are creating. Complete the circle. Note that you can use the **Offset** command multiple times when creating the circle.

Alternatively:

1. Select the first element.

2. Select the second element.

3. Specify the offset point between the two elements.

Circle by Center and Circumference Point

To draw a circle by indicating the center and point:

1. Click **Circle** in **Geometry**.

2. Click **Center & Point or Radius** in **Circle**.

3. Specify the center of the circle.

4. Specify a point on the circumference, or enter the length of the radius. The circle moves with the cursor until the circumference point is defined.

   You can create additional concentric circles by indicating additional points or radii.

Circle by Three Points

To draw the circle by indicating three points on the circumference:

1. Click **Circle** in **Geometry**.

2. Click **3 Points** in **Circle**.

3. Specify three circumference points.

   The circle drawn will vary with the movement of the cursor until the third point is defined.

Circle by Radius and Two Elements

To draw a circle of given radius tangential to two existing elements:

1. Click **Circle** in **Geometry**.

2. Click **Tangent** in **Circle**.

3. Specify the approximate tangent points on the two elements.

4. Enter the radius required.

The elements can be either lines, arcs or circles. Creo Elements/Direct Drafting will calculate the best solution or, if no solution is possible, it will suggest a valid radius.

Concentric Circles

To draw concentric circles:

1. Click **Circle** in **Geometry**.
2. Click **Concentric** in **Circle**.
3. Click the existing circle.
4. Specify the desired circumference point and the circle will be drawn.

   To draw further concentric circles, simply specify further circumference points.

Alternatively:

1. Click **Concentric**.
2. Click the existing circle.
3. Enter the radial offset.

Circle by Diameter

To draw a circle by diameter:

1. Click **Circle** in **Geometry**.
2. Click **Diameter** in **Circle**.
3. Specify the two diameter end points for the circle required.

Drawing Arcs

This section shows how to use the commands for drawing arcs. Arcs are normally drawn in the counterclockwise direction. The options are:

- **Offset**
- **3 Points**
- **Center & Radius & Angle**
- **Center & Radius & Ends**
- **Concentric**
- **Point & Radius & Angles**

- **Diameter**

- **Smooth**

Figure 35. Arc Geometry on page 124 shows arc geometry.

Figure 35. Arc Geometry
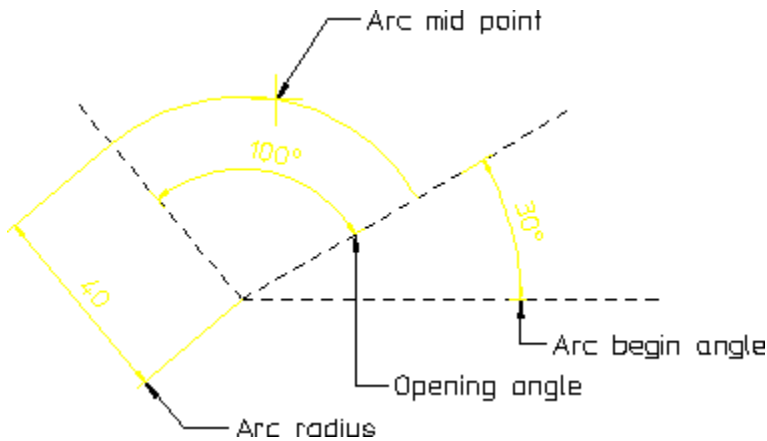
Specifying an Offset for Arcs

The **Offset** command enables you to specify an offset for a new arc. This offset is defined by the distance from two existing linear elements.

To define an offset for an arc:

1. Click **Arc** in **Geometry**.

2. Click on an arc option (such as **3 Points**) in **Arc**.

3. Click **Offset** in **Arc**.

4. Select the first linear element. Specify the required distance from this element. Click a point on one side of this element.

5. Select the second linear element. Specify the required distance from this element. Click a point on one side of this element.

6. The offset point thus defines a point of the arc you are creating. Complete the arc. Note that you can use the **Offset** command multiple times when creating the arc.

Alternatively:

1. Select the first element.

2. Select the second element.

3. Specify the offset point.

Arc by Three Points

To draw an arc through three points:

1. Click **Arc** in **Geometry**.
2. Click **3 Points** in **Arc**.
3. Specify the two end points of the required arc.
4. Specify a third point on the arc.

   The arc drawn will vary with the movement of the cursor until the third point is defined.

Arc by Center and Radius

To draw an arc with a given center and radius:

1. Click **Arc** in **Geometry**.
2. Click **Center & Radius & Angle** in **Arc**.
3. Specify the center point.
4. Enter the radius.
5. Enter the start angle and the end angle of the arc. Alternatively, you can define the angles by specifying two points.

Arc by Center and End Points

1. Click **Arc** in **Geometry**.
2. Click **Center & Radius & Ends** in **Arc**.
3. Specify the center of the arc.
4. Specify the radius (arc start point).
5. Specify a third point. It does not need to lie on the arc plane, but its line defines the end of the arc.

Concentric Arcs

To draw concentric arcs:

1. Click **Arc** in **Geometry**.
2. Click **Concentric** in **Arc**.
3. Click the existing arc.
4. Specify a point on the required arc.

   Simply specify further points to draw further concentric arcs.

Arc by Center and Radius and Angle

To draw an arc by specifying the arc center and radius:

1. Click **Arc** in **Geometry**.

2. Click **Point & Radius & Angles** in **Arc**.

3. Specify the center of the arc.

4. Enter the arc radius.

5. Enter the arc begin (or end) angle (angles are measured counterclockwise from the horizontal).

6. Enter the arc angle.

Arc by Diameter

1. Click **Arc** in **Geometry**.

2. Click **Diameter** in **Arc**.

3. Specify the diameter end points and an arc will be drawn between them.

   The resulting position of the arc will depend upon which diameter point is specified first.

Smoothly Connecting Arc

To draw a smoothly connecting arc:

1. Click **Arc** in **Geometry**.

2. Click **Smooth**.

3. Click an element end point (line, circle, and so on).

4. Specify the end point of the arc.

   The arc is drawn tangential to the element you have clicked. With each further point indicated, another arc is drawn tangential to the previous one. In Figure 36. Smoothly Connecting Arcs on page 126, points 1 to 4 have been specified in sequence.

Figure 36. Smoothly Connecting Arcs
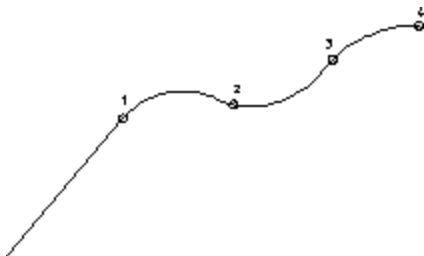
Drawing Fillets

To draw a fillet:

1. Click **Fillet** in **Geometry**.
2. Enter the radius.
3. Click the two elements between which the fillet should be drawn, or click the vertex.

   The fillet radius will be drawn and by default the corner point will be deleted. If you do not want the corner to be deleted, switch on **Keep Corner** before clicking the elements or vertex.

Changing Fillets

To change fillet radii you have already created:

1. Click **Fillet** in **Geometry**.
2. Enter the new fillet radius.
3. Click the individual fillets to be changed or enclose them in a box.

   You also can change all existing fillets by clicking **Fillet** and **All** in the **Select** menu. Then click **Confirm**. To remove all fillets, enter a radius of zero.

Drawing Chamfers

Four methods of drawing a chamfer are described in this section. By default the corner will be deleted after the chamfer is drawn. If you do not want the corner to be deleted, switch on **Keep Corner** before you choose a chamfer option.

Chamfer at Distance from Vertex

To draw a chamfer at a given distance from the vertex along each line:

1. Click **Chamfer** in **Geometry**.
2. Click **Vertex** in **Chamfer**.
3. Enter the chamfer distance.
4. Click the vertex (the point where the two lines intersect).

Chamfer by Two Points

To draw a chamfer by two points:

1. Click **Chamfer** in **Geometry**.

2. Click **2 Points** in **Chamfer**.

3. Click the start point and end point of the required chamfer.

Chamfer by Two Distances

To draw a chamfer by specifying two different distances:

1. Click **Chamfer** in **Geometry**.

2. Click **Dist. Dist.** in **Chamfer**.

3. Enter the distance for chamfer along the first element.

4. Enter the distance for chamfer along the second element.

5. Click the first element.

6. Click the second element.

Chamfer at Distance and Angle

To draw a chamfer at a given angle to the first line and at a given distance from the vertex:

1. Click **Chamfer** in **Geometry**.

2. Click **Dist. Ang.** in **Chamfer**.

3. Enter the angle between the chamfer and the first element.

4. Enter the chamfer distance.

5. Click the first element.

6. Click the second element.

Drawing Equidistant (Parallel) Contours

To create a contour equidistant to an existing contour:

1. Click **Equidist** in **Geometry**.

2. Enter the separation required for the equidistant/parallel contour.

3. Click the start point of the existing contour.

4. Select the elements to be copied (Creo Elements/Direct Drafting will allow you to skip to the final element, provided you do not pass more than one branch element that is not to be copied).

5. Indicate 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 **End**.

3. Specify the side on which the equidistant contour is to be drawn.

Using Overdraw

Use **Overdraw** to draw a contour between intersection points in the construction geometry. The lines will be 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 the **Overdraw** command:

1. Draw the construction geometry as shown in Figure 37. Using the Overdraw Command on page 129.

2. Click **Overdraw**.

3. Specify the start point (1).

4. Specify points 2 to 14 (end point).

Figure 37. Using the Overdraw Command
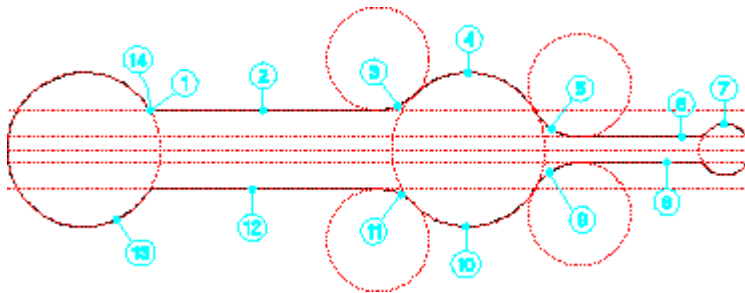
The **Overdraw** command also supports multiple **Undo** operations.

Creating a Centerline

This command lets you create a centerline for a circular element. The centerline becomes part of the circular element and is moved or deleted if the associated circular element is moved or deleted.

To create a centerline, click **Centerline** in the **Geometry** menu and click the circle. 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 38. Creating a Centerline
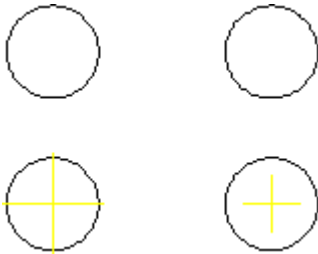
When you dimension centerlines and then make a modification, the dimension is automatically updated. When asked to click the first element, click the center point of the circle (use Catch Center).

Figure 39. Effect of Changing the Position
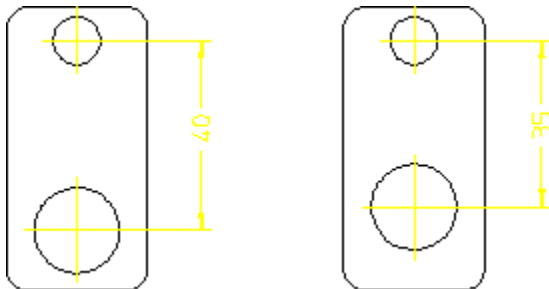
Click **Change** in the **Centerline** menu to change the position of the centerline.

To create a single slot centerline,

1. Go to **Geometry ▸ Centerline ▸ Slot Single**.

2. Specify a circular element of a slot. Creo Elements/Direct Drafting draws the single slot centerlines.
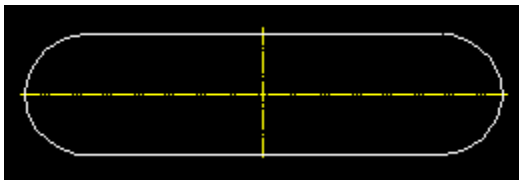
To create double slot centerlines,

1. Go to **Geometry ▸ Centerline ▸ Slot Double**.
2. Specify a circular element of a slot. Creo Elements/Direct Drafting draws the double slot centerlines.
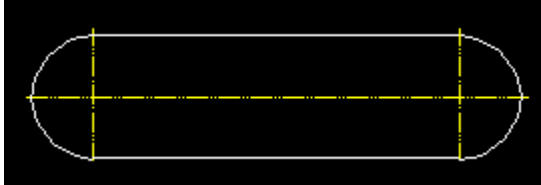
Limitations of slot centerlines:

- 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.

Creating Symmetric Elements

Use the **SymLine** command to create 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 40. Associated Symmetric Elements on page 132.
2. Create circle 1 having diameter 1 mm.
3. Create circle 2 having diameter 3 mm.
4. Click **SymLine** in the **Geometry** menu.
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 40. Associated Symmetric Elements
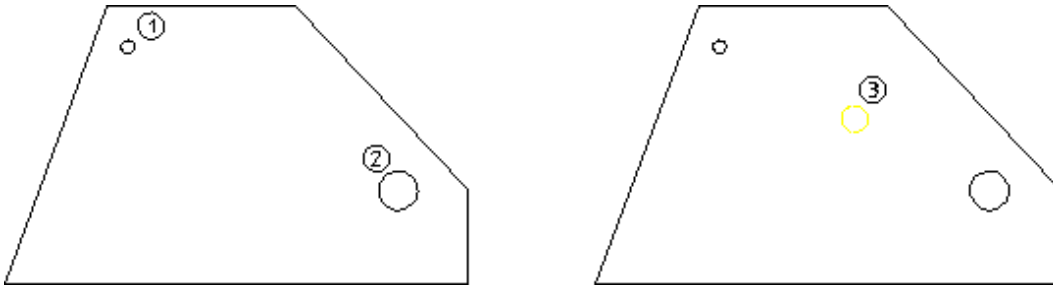
Click **Change** in the **SymLine** menu to change the position of the symmetry line.

Now try moving the circle at position 2 and notice how the associated element also moves.

Figure 41. Effect of Moving the Defining Circle
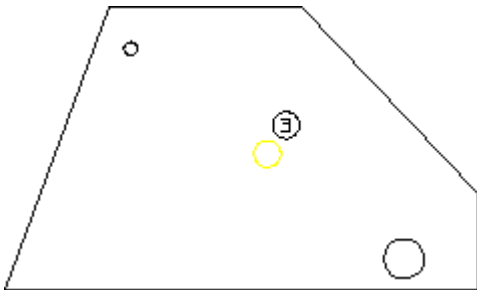
Figure 42. Finding Lines of Symmetry on page 132 shows another example of where to use the **SymLine** command.

Figure 42. Finding Lines of Symmetry
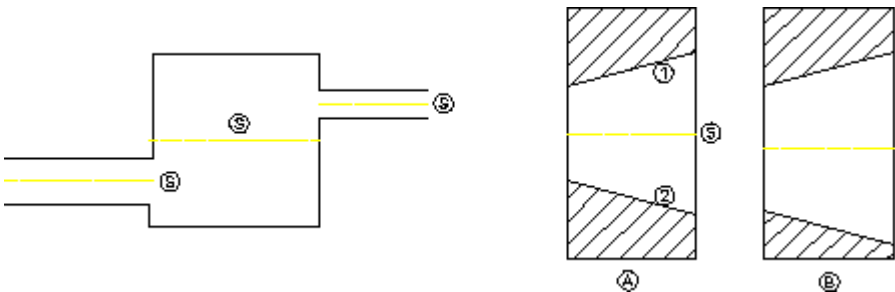
Indicating the

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 indicate this point in your drawing:

1. Click **Projected Reference Point** in **Geometry**.

2. Click the first element.

3. Click the second element.

The point where the traces of the two elements meet is now shown 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. Projected Reference Point on page 133 shows an example of a projected reference point.

Figure 43. Projected Reference Point
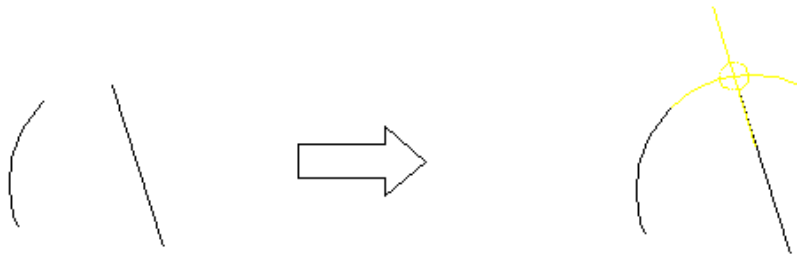
Changing the Settings of the Projected Reference Point

To change the settings of the projected reference point and its associated elements:

1. Click **Setup ▸ Edit Environment ▸ Projected Reference Points**.

2. In **Projected Reference Point Settings**, select the linetype, color, width, size and show option for the individual elements associated with the projected reference point.

Drawing Ellipses

The commands used for drawing an ellipse are:

- **Axis Ends & Peripheral Point**
- **Center & Angle & Radius**
- **Center & 2 Peripheral Points**
- **2 Focal Points & Circumference Point**
- **Focal Point & Angle & Ex_value & Radius**

Ellipse by Axis End Points and Peripheral Point

To draw an ellipse by axis end points and peripheral point:

1. Click **Ellipse** in **Geometry**.
2. Click **Axis Ends & Peripheral Point** in **Ellipse**.
3. Specify the two vertices of the major axis of the required ellipse.
4. Specify a point on the circumference.

Ellipse by Center, Angle and Radii

To draw an ellipse by center, angle and radii:

1. Click **Ellipse** in **Geometry**.
2. Click **Center & Angle & Radius** in **Ellipse**.

   See Figure 44. Ellipse by Center, Angle and Radii on page 134.

   Figure 44. Ellipse by Center, Angle and Radii
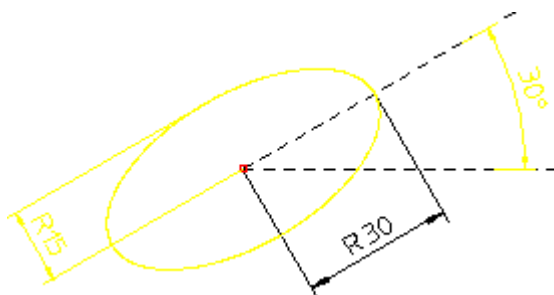
3. Specify the center of the required ellipse.
4. Enter the angle of the major axis with respect to the x axis (30° in Figure 44 on page 101).

5. Enter the major radius (30mm) in Figure 44. Ellipse by Center, Angle and Radii on page 134).

6. Enter the minor radius (15mm) in Figure 44. Ellipse by Center, Angle and Radii on page 134).

Ellipse by Center and Two Points

To draw an ellipse using the center and two points:

1. Click **Ellipse** in **Geometry**.
2. Click **Center & 2 Peripheral Points** in **Ellipse**.
3. Specify the center point of the ellipse.
4. Specify the first point on the circumference.
5. Specify the second point on the circumference.

Ellipse by Focus and Points

Use the **Ellipse** command options **2 Focal Points & Circumference Point** and **Focal Point & Angle & Ex_value & Radius** to create elliptical shapes by means of the focal points and the major radius.

Figure 45. Ellipses by Focus, Point, Angle
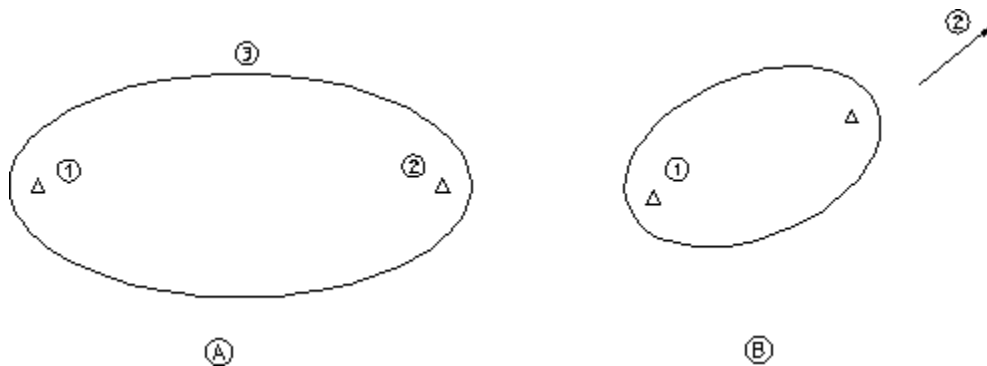
To create an ellipse by specifying the focus (drawing A):

1. Click **Ellipse** in **Geometry**.
2. Click **2 Focal Points & Circumference Point** in **Ellipse**.
3. Specify the first focal point (1).
4. Specify the second focal point (2).
5. Specify a point on the circumference (3).

To create an ellipse by specifying the focal points and a direction (drawing B):

1. Click **Focal Point & Angle & Ex_value & Radius** in **Ellipse**.

2. Specify the first focal point (point 1).

3. Specify the direction of the major axis (point 2)

4. Enter an Ex-value (for example, **0.8**)

5. Enter a value for the major radius (for example, **15**)

Drawing Hyperbolas

The commands used for drawing a hyperbola are:

- **Center & Focus & End Point**
- **Center & Tangential & Focus Point**

Hyperbola by Center, Focus and End Points

To draw a hyperbola using the center, focus and end points:

1. Click **Hyperbola** in **Geometry**.

2. Click **Center & Focus & End Point** in **Hyperbola**.

3. Specify the center of the hyperbola (point 1).

4. Specify the focus of the hyperbola (point 2).

5. Specify one end of the hyperbola (point 3). Note that the other end point of the hyperbola is computed by symmetry.

Figure 46. Hyperbola by Center, Focus, and End Point
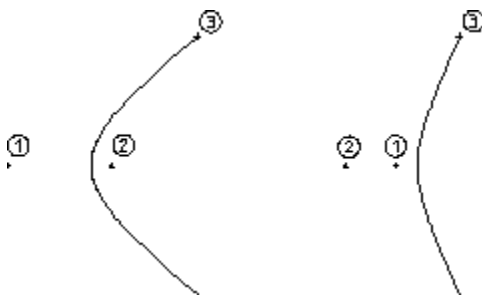
Hyperbola by Center, Tangent and Focus

To draw a hyperbola tangential to existing geometry:

1. Click **Hyperbola** in **Geometry**.

2. Click **Center & Tangential & Focus Point** in **Hyperbola**.

3. Specify the center of the hyperbola (point 1).

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4. Click a tangential point (point 2).

5. Specify the focus of the hyperbola (point 3).

Figure 47. Hyperbola by Center, Tangent, and Focus
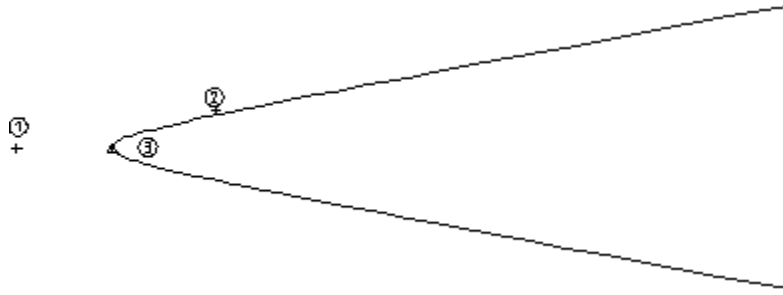
Creo Elements/Direct Drafting automatically adds the tangential construction geometry.

Drawing Parabolas

The commands used for drawing a parabola are:

- **Center & Focus**
- **Focus & Direction & End Point**

Parabola by Center and Focus

To draw a parabola using its center and focus:

1. Click **Parabola** in **Geometry**.

2. Click **Center & Focus** in **Parabola**.

3. Specify the center of the parabola (point 1)

4. Specify the focus of the parabola (point 2)

Figure 48. Parabola by Center and Focus
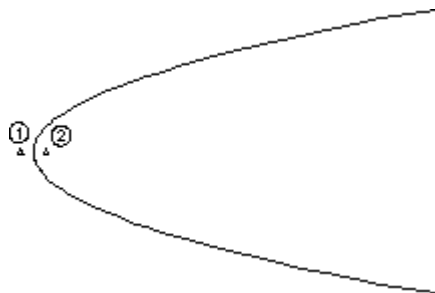
Parabola by Focus, Direction and End Points

To draw a parabola in a specific direction:

1. Click **Parabola** in **Geometry**.

2. Click **Focus & Direction & End Point** in **Parabola**.

3. Specify the focus of the parabola (point 1).

4. Specify the direction of the parabola (point 2).

5. Specify the end of the parabola (point 3). Note that the other end point of the parabola is computed by symmetry.

Figure 49. Parabola by Focus, Direction and End Points
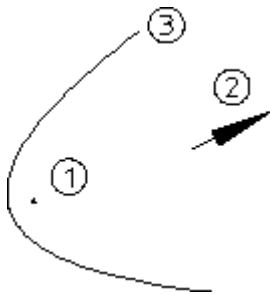
Drawing Splines

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 50. Typical Spline Curves: Engine Manifold, Pipe Tree, Racquet Head

Spline Components
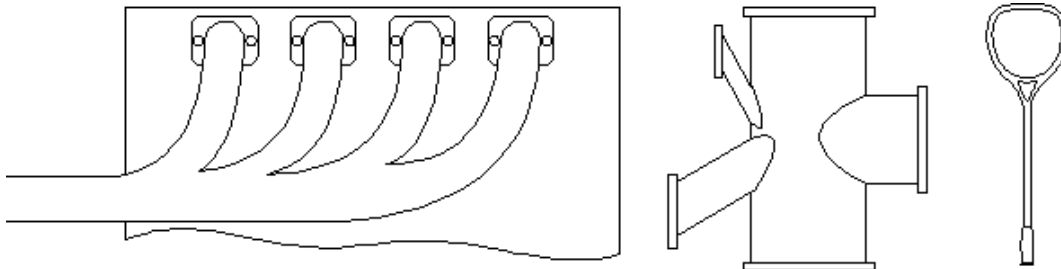
shows a simple spline with its control polygon visible. The control polygon can be thought of as a convex hull containing the spline. Note that both interpolation and control splines have a control polygon. Points on the interpolation spline curve are called interpolation points. Points on the control polygon are called control points.

Figure 51. Spline Components
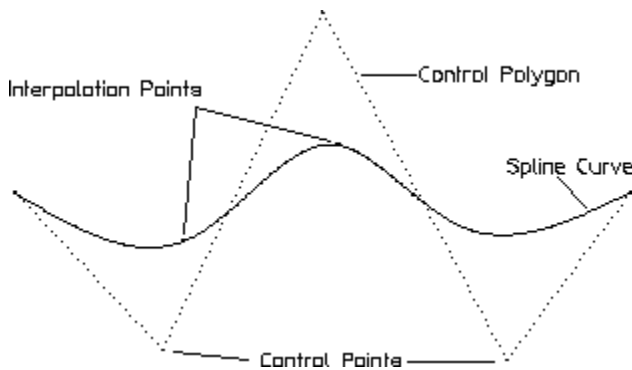
Interpolation Spline vs. Control Spline

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** option provided in **Interpolation Spline** and **Control Spline** enables you to create closed splines.

Figure 52. Open Spline vs. Closed Spline

Open Splines   Closed Splines
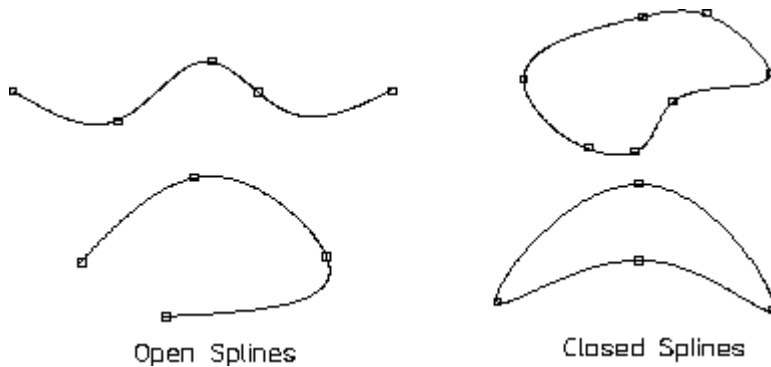
Specifying the Order of a Spline

The spline order controls the smoothness of the curve. The higher the order, the smoother the spline. Figure 53. Spline Order on page 140 shows two splines drawn through the same control points.

Figure 53. Spline Order
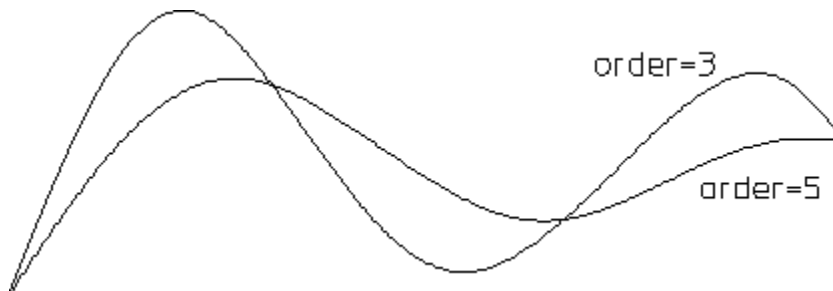
order=3

order=5

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.

Interpolation Splines

Points on an interpolation spline (I spline) are called interpolation points. A spline created with interpolation points will pass exactly through the interpolation points used to define it.

📋 Note

To display the control polygon of an interpolation spline, switch on Vertex in the Show menu.

In this section:

Drawing Interpolation Splines

In this section:

Spline by Interpolation Points

To draw an interpolation spline:

1. Click **Interpolation Spline** in **Geometry**.
2. Click **Create** in **Interpolation Spline**.
3. Specify a sequence of interpolation points.
4. Click **End**.

Creo Elements/Direct Drafting draws a spline curve through the points you specified.

Appending an Interpolation Spline

To append an interpolation spline to existing geometry:

1. Click **Interpolation Spline** in **Geometry**.
2. Click **Append** in **Interpolation Spline**.
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 **End**.

To append the begin-point and end-point of an interpolation spline:

1. Click **Interpolation Spline** in **Geometry**.
2. Click **Append** in **Interpolation Spline**.
3. Specify point A and a series of points B1, B2, B3, ..., Bn.
4. Click **Append** in **Interpolation Spline**.
5. Specify point C.
6. Click **End**.

Figure 54. Appending an Interpolation Spline
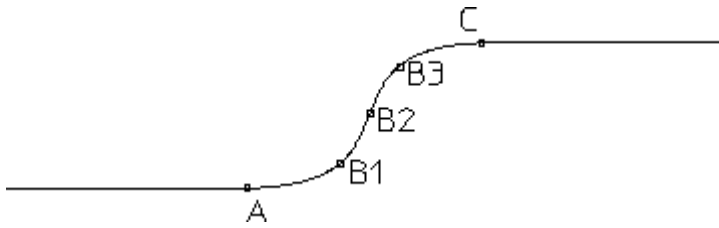
Specifying the Slope of an Interpolation Spline

When using **Create** to draw a new interpolation spline, you can use the **Tangent** option to specify the slope angle at which the interpolation spline leaves or approaches the current interpolation point. For example, to specify a slope of 70°:

1. Click **Interpolation Spline** in **Geometry**.

2. Use **Create** or **Append** in **Interpolation Spline** to define the first interpolation points (such as point A in Figure 55. Specifying the Slope of an Interpolation Spline on page 142).

3. Click **Tangent** in **Interpolation Spline**.

4. Type in the slope angle (such as 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 55. Specifying the Slope of an Interpolation Spline on page 142).

6. Click **End**.

Figure 55. Specifying the Slope of an Interpolation Spline
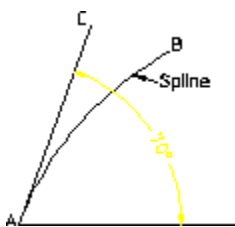
By specifying a slope at a given interpolation point, you add a tangency condition to the spline. This tangency condition can be modified (see Changing the Slope of an Interpolation Spline on page 145) or deleted (see Deleting a Tangency Condition of an Interpolation Spline on page 147).

Figure 56. Effects of Different Slope Angles on page 143 shows the effects of specifying different slope angles:

Figure 56. Effects of Different Slope Angles
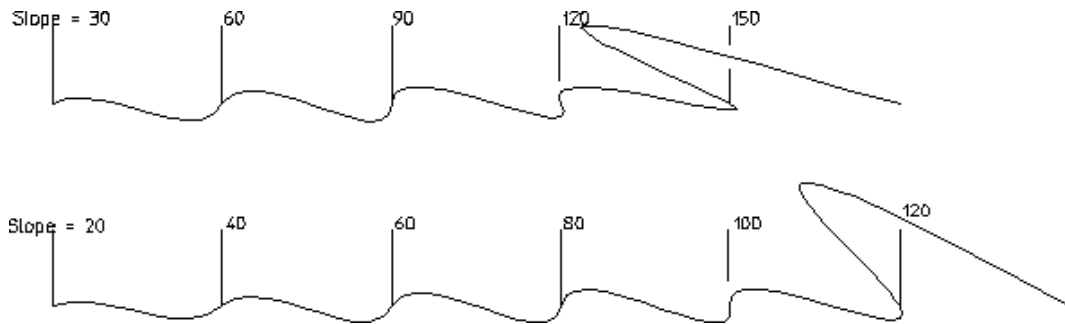
Being able to specify the slope of a spline is useful when joining unequal diameters. The following diagram shows the difference between curves created with the **Append** option (curve CD) and the **Tangent** option (curve AB).

Figure 57. Slope of Interpolation Splines
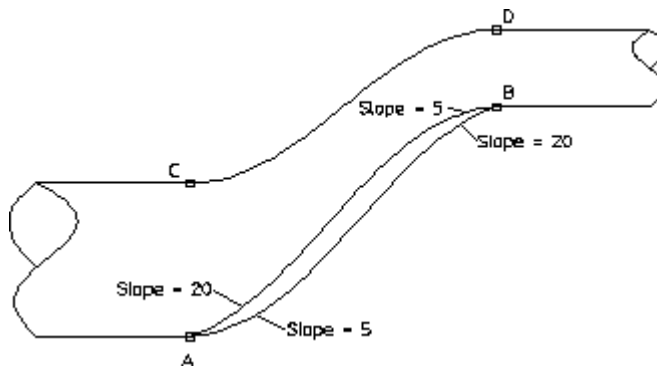
To draw curve C to D, do the following:

1. Click **Interpolation Spline** in **Geometry**.
2. Click **Append** in **Interpolation Spline**.
3. Click point **C**.
4. Click **Append** in **Interpolation Spline**.
5. Click point **D**.
6. Click **End**.

To draw the lower curve A to B, do the following:

1. Click **Interpolation Spline** in **Geometry**.
2. Click **Create** in **Interpolation Spline**.
3. Click point **A**.
4. Click **Tangent** in **Interpolation Spline**.
5. Enter **5**.

6. Click point **B**.

7. Click **Tangent** in **Interpolation Spline**.

8. Enter **20**.

9. Click **End**.

Modifying Interpolation Points

In this section:

Adding an Interpolation Point

To add an interpolation point to an interpolation spline:

1. Click **Interpolation Spline** in **Geometry**.

2. Click **Modify Point** in **Interpolation Spline**.

3. Click **Add** in **Modify Point**.

4. Click the spline to which you want to add an interpolation point.

5. 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 **Interpolation Spline** in **Geometry**.

2. Click **Modify Point** in **Interpolation Spline**.

3. Click **Change** in **Modify Point**.

4. Click the spline you want to modify.

5. Click the interpolation point you want to move.

6. Click the new position of the interpolation point.

When you move an interpolation point, other interpolation points will also be moved. The shape of the spline changes accordingly.

Deleting an Interpolation Point

To delete an interpolation point of a spline:

1. Click **Interpolation Spline** in **Geometry**.

2. Click **Modify Point** in **Interpolation Spline**.

3. Click **Delete** in **Modify Point**.

4. Click the spline you want to modify.

5. 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 **Interpolation Spline** in **Geometry**.

2. Click **Modify Point** in **Interpolation Spline**.

3. Click **Shape** in **Modify Point**.

4. Click the spline you want to modify.

5. Click the interpolation point you want to move.

6. 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

In this section:

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 58. Effect of Changing the Slope at an Interpolation Point on page 146 shows a simple spline and the same spline with the tangent of interpolation point A changed.

Figure 58. Effect of Changing the Slope at an Interpolation Point
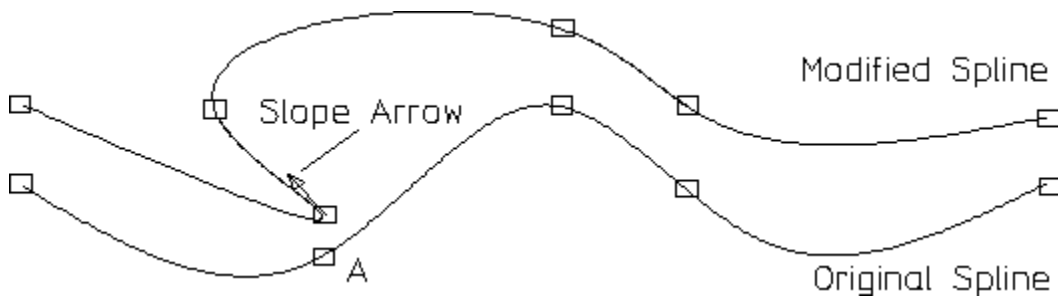
To change the spline slope at an interpolation point:

1. Make sure that the **Vertex** option is switched on (ticked) in the **Show** option box so that the interpolation points are displayed.

2. Click **Interpolation Spline** in **Geometry**.

3. Click **Modify Tangent** in **Interpolation Spline**.

4. Click **Add** in **Modify Tangent**.

5. Click the interpolation spline.

6. Click an interpolation point of the spline. Creo Elements/Direct Drafting displays an arrow to indicate the spline slope at that point.

7. Type in the required slope angle (such as 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 **Interpolation Spline** in **Geometry**.

2. Click **Modify Tangent** in **Interpolation Spline**.

3. Click **Change** in **Modify Tangent**.

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 in the required slope angle (such as 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 142, Changing the Slope at an Interpolation Point on page 145, and Changing the Slope at Any Point on an Interpolation Spline on page 146) by deleting the associated tangency condition.

To delete a tangency condition:

1. Make sure that the **Vertex** option is switched on (ticked) in the **Show** option box so that the interpolation points and any points where you changed the spline slope are displayed.

2. Click **Interpolation Spline** in **Geometry**.

3. Click **Modify Tangent** in **Interpolation Spline**.

4. Click **Delete** in **Modify Tangent**.

5. Click the spline.

6. Click the point where you defined or changed the slope angle.

The spline will return 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 **Interpolation Spline** in **Geometry**.

2. Click **Modify Tangent** in **Interpolation Spline**.

3. Click **Shape** in **Modify Tangent**.

4. Click the spline.

5. 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.

Control Splines

When you create a spline using a control polygon, the shape of the spline is influenced by the control polygon formed by the control points.

In this section:

Drawing Control Splines

To draw a control spline:

1. Click **Control Spline** in **Geometry**.

2. Click **Create** in **Control Spline**.

3. Specify a sequence of control points. Note that 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.

4. Click **End**.

Creo Elements/Direct Drafting draws a spline curve based on the control points you specified.

Modifying Control Points

You can modify the size and shape of a control spline by modifying its control points.

In this section:

Adding a Control Point

To add a control point to a control spline:

1. Click **Control Spline** in **Geometry**.

2. Click **Modify Point** in **Control Spline**.

3. Click **Add** in **Modify Point**.

4. Click the spline you want to modify.

5. Click a position between two existing control points.

6. Move the cursor to the position where you want to add the new control point.

7. 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 **Control Spline** in **Geometry**.
2. Click **Modify Point** in **Control Spline**.
3. Click **Change** in **Modify Point**.
4. Click the spline you want to modify.
5. Click the control point you want to move.
6. 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 **Control Spline** in **Geometry**.
2. Click **Modify Point** in **Control Spline**.
3. Click **Delete** in **Modify Point**.
4. Click the spline you want to modify.
5. 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 **Control Spline** in **Geometry**.
2. Click **Modify Point** in **Control Spline**.
3. Click **Shape** in **Modify Point**.
4. Click the spline.
5. 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

You can change the spline slope at any point on the control spline. This operation will move the control points of the spline.

To change the spline slope at any point on the spline:

1. Click **Control Spline** in **Geometry**.

2. Click **Modify Tangent** in **Control Spline**.

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 (such as 70) 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.

Modifying Splines

Splines can be converted to lines and arcs or subdivided (split) into individual spline segments.

Converting Splines

Splines need to be co-converted only if you want to export them to devices (such as numerical control machines) that do not recognize spline curves.

The **Convert** command converts splines to arcs and lines by approximation. This command provides the following options:

Arcs & Lines	Use this option to convert splines to arcs and lines.
Lines	Use this option to convert splines to lines.

Using the Trimming Function

Splines can be split in the same way as other line elements. However, you can set a **Trimming** function that affects the way in which a spline is split. The **Trimming** function has the options:

On	Two new (independent) splines are created, each with their own mathematical behavior.
Off	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 59. Splitting Splines on page 151 shows the effects of splitting a spline with **Trimming**:

On	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.
Off	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 59. Splitting Splines
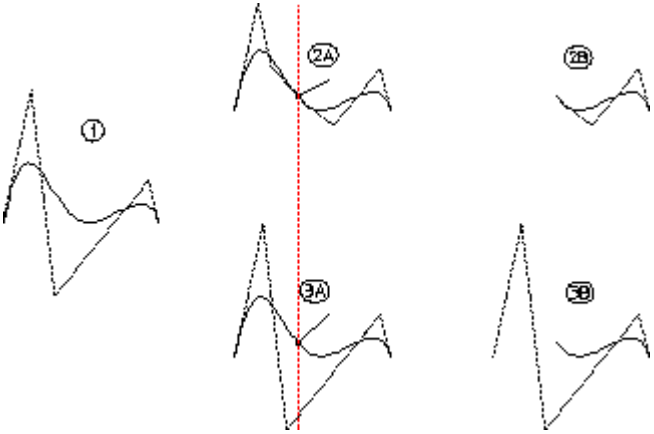
In drawings 2B and 3B, the left spline has been deleted. Notice in drawing 3B how the control polygon thinks that the spline starts at its original point (drawing 1).

To split and trim a spline:

1. Click **Interpolation Spline** or **Control Spline** in **Geometry**.

2. Switch on the **Trimming** option in **Interpolation Spline** or **Control Spline**.

3. Click **Subdivide** in **Interpolation Spline** or **Control Spline**.

4. Divide the spline. Proceed as described in Subdividing a Spline on page 152.

Subdividing a Spline

The **Subdivide** command either divides a spline into section of equal length (spline 1 in Figure 60. Dividing a Spline on page 152) or splits a spline at points along its length (spline 2 in Figure 60. Dividing a Spline on page 152).

Figure 60. Dividing a Spline
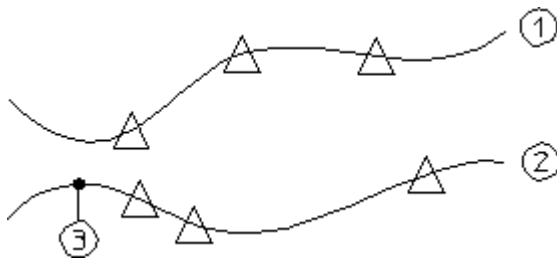
Dividing a Spline into Segments of Equal Length

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 **Interpolation Spline** or **Control Spline** in **Geometry**.

2. Click **Subdivide**.

3. Click the spline you want to split.

4. Enter an integer for the number of segments you want to produce (for example, 4).

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 will be split at the end point of the spline segment defined by this length.

To divide a spline by specifying a split point:

1. Click **Interpolation Spline** or **Control Spline** in **Geometry**.

2. Click **Subdivide**.

3. Click the spline you want to split.

4. Click any point on the spline you want to split.

5. Specify a length. A split point is created at the end point of the spline segment defined by this length.

If **Trimming** is switched on (see Using the Trimming Function on page 150), the spline segment "beyond" this split point will be hidden.

Example – Cam Profile

This example describes how to create a cam profile and then modify it.

Construct the following geometry (see Figure 61. Cam Profile - 1 on page 153):

- Horizontal and vertical lines at 0 0
- Circles at 35 40 45
- Lines at angles 30 60 120 150

Figure 61. Cam Profile - 1
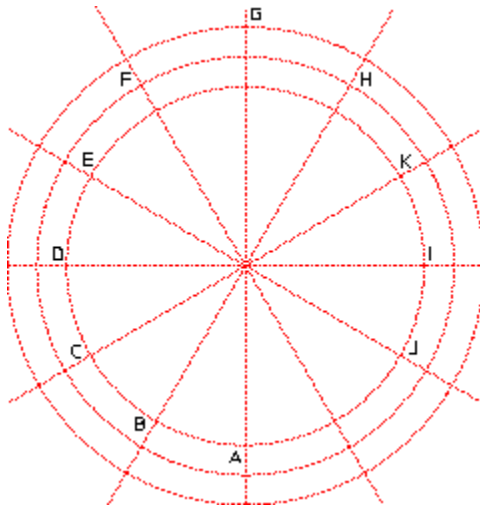
To create the initial profile:

1. Click **Interpolation Spline** in **Geometry**.

2. Click **Create** in **Interpolation Spline**.

3. Switch on **Closed**.

4. Specify points A, B, ... J.

5. Click **End**.

Figure 62. Cam Profile - 2
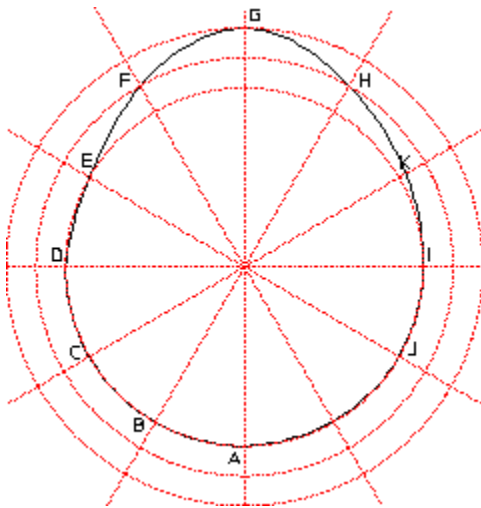
You now want to add an interpolation point at K:

1. Click **Add** in **Modify Point**.
2. Click the spline.
3. Click a point on the spline that is close to point K.
4. Click the intersection point at K.

Figure 63. Cam Profile - 3
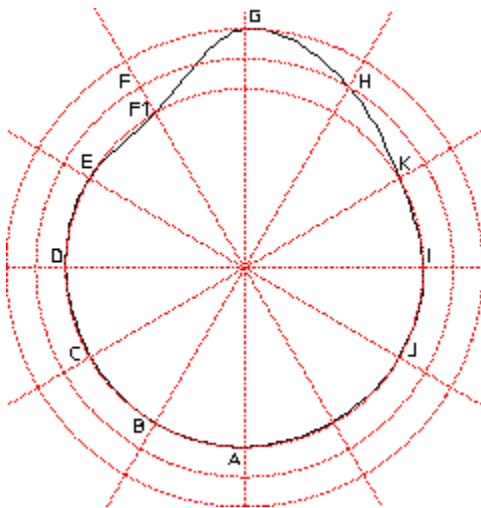
To delete and interpolation point:

1. Click **Delete** in **Modify Point**.
2. Click the spline.
3. Click point **C**.

The system deletes the point.

To change the position of an interpolation point:

1. Click **Change** in **Modify Point**.
2. Click the spline.
3. Click point **F**.
4. Move the point and click point **F1**.

To change the angle at point F1:

1. Click **Change** in **Modify Tangent**.
2. Click the spline.
3. Click point **F1**.

   The system displays an arrow indicating the slope direction. You can now either click a point to indicate the direction or enter a value. (This example uses an angle of 10°.)

4. Enter **10**

Figure 64. Cam Profile - Final Shape

7

Modifying a Drawing

This chapter shows you how to use the commands 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 .
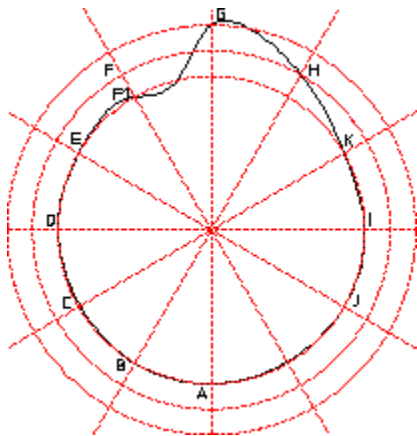
Keeping or Removing the Original Elements

For all modification commands/options (except **Isometric**) you can switch **Keep** on or off in the **Modify** menu. This determines whether the original drawing elements will be kept or deleted after the modification. For example, clicking **Move** with **Keep** copies the drawing elements concerned rather than just moving them. By default, **Keep** is switched off.

Repeating a Modification

You can use the **Repeat** function to repeat a modification a given number of times. For example, by switching on **Repeat** and specifying a repeat factor of 5, you can perform the same **Move** operation five times.

Modifying Using Selection

The **Selection** mechanism described in Commonly Used Commands on page 35 can also be used with all the **Modify** commands/options (except **Isometric** and **Stretch**).

Using **Select** gives you various ways to select elements to be modified and to see them highlighted on the screen before the modification is made.

To use **Select** with **Modify**:

1. Click the appropriate modification option (such as **Move** and **Horizontal**) in **Modify**.

2. Select the elements to be modified using the options available in the **Select** menu. For example, you can click **Circle** and **All** in the **Select** menu to perform the same modification on all circles.

3. Click **Confirm** to perform the modification on the selected elements.

> **⬛ 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 Global in Select).

   Refer to Handling Parts and Assemblies on page 335 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 **Setup ▸ Edit Environment ▸ System**.

2. Enter the appropriate number of elements in the **Max feedback** field in the **Environment** tab.

Moving Elements

Use the Handles or the **Move** command to move or copy elements to another position.

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.

3. Click **End**.

To move multiple elements with handles,

1. Hold the Shift key and click all the elements in the drawing area. The handles appear.

2. Release the Shift key and click one of the elements again.

3. Drag the elements to a new location.

4. Click **End**.

Click **Move** to move or copy drawing elements to another position.

Click **Move** with **Keep** switched on to produce a copy. The original remains unchanged.

With **Keep** switched on or off, switch on **Repeat** and enter a repeat factor. This will repeat the next operation the specified number of times.

By Two Points

You can click any two points to define a move vector.

To move or copy a drawing element(s) using a reference point:

1. Click **Move** in **Modify**.
2. Click **2 Points** in **Move**.
3. Select the element(s) to be moved.
4. To define the move vector, specify a reference point and then a destination point.

   Creo Elements/Direct Drafting moves the element(s) to the new position.

Horizontal Move/Copy

To move or copy a drawing element(s) to a new position in the horizontal direction:

1. Click **Move** in **Modify**.
2. Click **Horizontal** in **Move**.
3. Select the element(s) to be moved.
4. Enter the required horizontal distance. Alternatively, specify a reference point and then a destination point to define the horizontal distance.

   If a repeat factor has been specified, the same horizontal distance will be taken for each move or copy.

⬛ Note

You can interrupt the command with the Measure Length or Measure Distance function if the distance you need can be measured from existing features on the drawing. Alternatively, you can specify two points defining the horizontal distance required.

Vertical Move/Copy

To move or copy a drawing element(s) to a new position in the vertical direction:

1. Click **Move** in **Modify**.

2. Click **Vertical** in **Move**.

3. Select the element(s) to be moved.

4. Enter the required vertical distance. Alternatively, specify a reference point and then a destination point to define the vertical distance.

   If a repeat factor has been specified, the same vertical distance will be taken for each move or copy.

Multiple Move/Copy

To interactively move or copy a drawing element(s) to any number of new position required:

1. Click **Move** in **Modify**.

2. Switch on the **Keep** option in **Modify**.

3. Click **Multiple** in **Move**.

4. Select the element(s) to be moved.

5. Specify a reference point for the move.

6. Specify the destination point for the move.

   The selected element(s) will move to the new position defined by these two points.

7. Specify as many further destination points as required.

8. Terminate the move or copy by clicking another command, another option, or **End**.

Rotating Drawings

Elements can be rotated:

Both of these methods can be used with **Keep** switched on or off in **Modify**.

About a Center

To rotate a drawing or drawing element about a center:

1. Click **Rotate** in **Modify**.
2. Click **Center** in **Rotate**.
3. Specify the center of rotation.
4. Enter the angle of rotation. Alternatively, you can define the angle by specifying two points. A positive angle results in a counterclockwise rotation.
5. Select the element(s) to be rotated.

The **Center** option is especially useful when combined with a **Repeat** factor for repeated rotation. The flywheel shown in was drawn by rotating the holes with a repeat factor of 5.

Figure 65. Flywheel
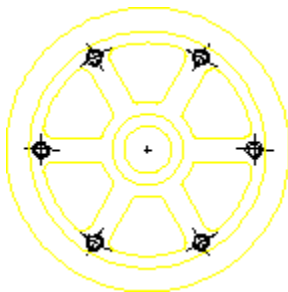
By Reference Point

To rotate a drawing or drawing element using a reference point:

1. Click **Rotate** in **Modify**.
2. Click **2 Points** in **Rotate**.
3. Specify a rotation reference point on the drawing or element.
4. Specify the destination for the rotation reference point.
5. Enter the angle of rotation. Alternatively, you can define the angle by specifying two points. A positive angle results in a counterclockwise rotation.
6. Select the element(s) to be rotated.

Mirroring

To save time when drawing symmetrical shapes you can mirror them about a line of symmetry.

Mirror is usually used with **Keep** in **Modify**. 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.

About Two Points

To mirror a drawing about two points:

1. Click **Mirror** in **Modify**.
2. Click **2 Points** in **Mirror**.
3. Specify two points to define the line of symmetry.
4. Select the elements to be mirrored.

    The line of symmetry in is the dashed line.

Figure 66. Line of Symmetry
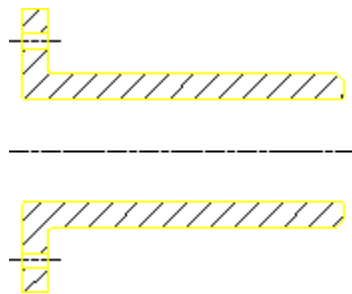
About a Horizontal Line

To mirror a drawing about a horizontal line:

1. Click **Mirror** in **Modify**.
2. Click **Horizontal** in **Mirror**.
3. Specify a point on the horizontal line of symmetry.
4. Select the elements to be mirrored.

About a Vertical Line

To mirror a drawing about a vertical line:

1. Click **Mirror** in **Modify**.
2. Click **Vertical** in **Mirror**.
3. Specify a point on the vertical line of symmetry.
4. Select the elements to be mirrored.

About a Center Point

To rotate a drawing through 180 degrees:

1. Click **Mirror** in **Modify**.
2. Click **Center** in **Mirror**.
3. Specify the center point of rotation.
4. Select the elements to be mirrored.

    The elements will be rotated 180 degrees in a clockwise direction.

Making Elements Larger or Smaller

Use **Scale** to make drawing elements and text larger or smaller. Note that dimensions will be changed accordingly. If you want to keep the original drawing elements, switch on **Keep** in **Modify**.

By Center

When drawing elements are scaled using the **Center** option, one point on the drawing remains fixed.

To scale your drawing using **Center**:

1. Click **Scale** in **Modify**.
2. Click **Center** in **Scale**.
3. Specify the center for the scaling operation.
4. Enter a scale factor or specify two points to define the scale factor.
5. Select the elements to be scaled.

By Reference Point

To scale your drawing using a reference point:

1. Click **Scale** in **Modify**.
2. Click **2 Points** in **Scale**.
3. Specify a reference point. Then specify the new position for this point.
4. Enter a scale factor or specify two points to define the scale factor.
5. Select the elements to be scaled. The selected elements will be moved and scaled.

Drawing Similar Elements

The **Similar** option is a combination of the **Move**, **Rotate** and **Scale** options. 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 **Similar** in **Modify**.
2. Click **2 Points** in **Scale**.
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 67. Fitting Drawing Elements Between Two Points on page 165 shows an example.

Figure 67. Fitting Drawing Elements Between Two Points
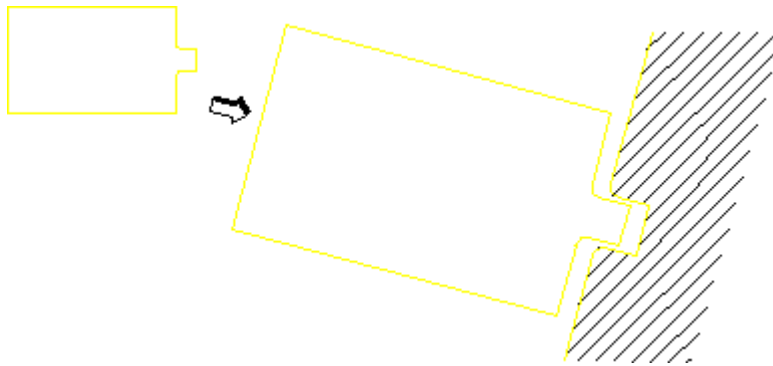
Creating a Parallel Perspective (Oblique) View

Use **Transform** to create a parallel perspective view.

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 68. Creating an Oblique View on page 166 shows an oblique view created by copying the end elevation, then using **Transform** command to create the side and top.

Points used for the first **Transform** operation are shown.

Figure 68. Creating an Oblique View
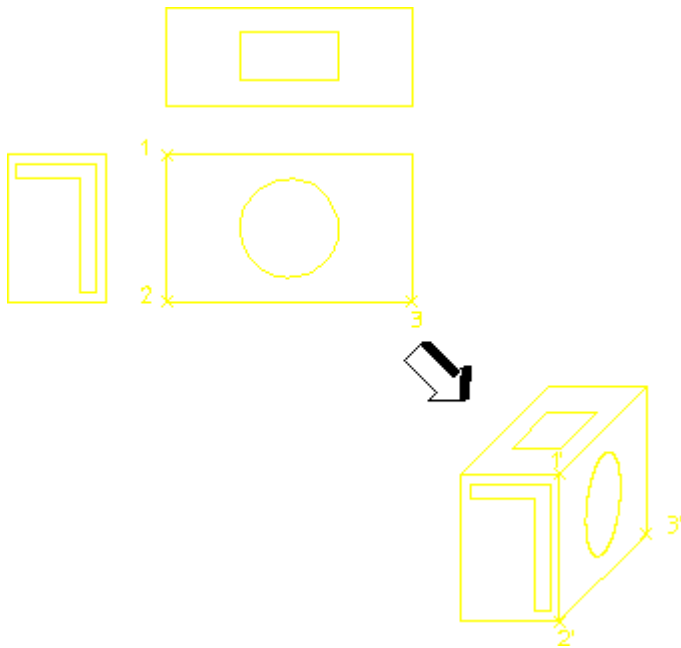
Position the End View

1. Click **Move** in **Modify**.
2. Click **2 Points** in **Move**.
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 68. Creating an Oblique View on page 166).

Position the Side View

1. Click **Transform** in **Modify**.
2. Click three reference points on the drawing (1, 2, and 3).
3. Click three destination points (1', 2', and 3') to which the three reference points will be mapped in the parallel perspective view.
4. Select the side view (marked by a circle in Figure 68. Creating an Oblique View on page 166).

Position the Top View

1. Click **Transform** in **Modify**.
2. Click three reference points on the drawing.

3. Click three destination points to which the three reference points will be mapped in the parallel perspective view.

4. Select the top view (marked by a rectangle in Figure 68. Creating an Oblique View on page 166).

Stretching a Drawing

Alter the shape of your drawing by stretching it in a certain direction. All the dimensioning, hatching, and finishing information will be modified to fit the new stretched shape.

Example

The following example shows how to use the **Stretch** command. Refer to Figure 69. Stretching a Drawing on page 167.

Figure 69. Stretching a Drawing
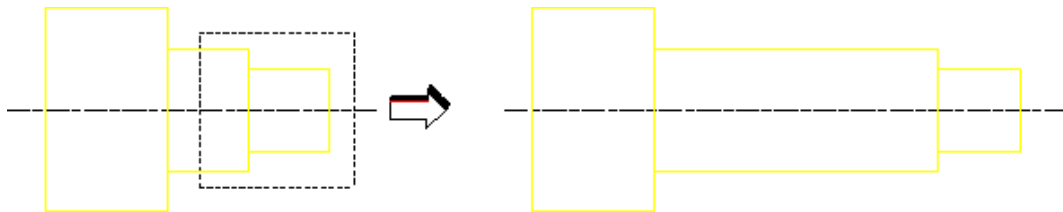
1. Switch off **Keep** in **Modify**.

2. Click **Stretch** in **Modify**.

3. Specify a reference point on the drawing.

4. Specify a destination point.

5. Select the elements to be stretched. You can select the elements by clicking a single vertex or boxing several (the vertex is moved and the element is stretched accordingly).

You can combine stretching with any of the other modify commands (except **Isometric**).

In Any Direction

To stretch drawing elements in an arbitrary direction:

1. Click **Stretch** in **Modify**.

2. Click **Free** in **Stretch**.

3. Select the drawing elements to be stretched.

4. Specify a reference point and a destination point for the stretch operation.

Horizontally

To stretch your drawing horizontally:

1. Click **Stretch** in **Modify**.

2. Click **Horizontal** in **Stretch**.

3. Select the drawing elements to be stretched.

4. Enter a horizontal distance. Alternatively, specify a reference point and a destination point for the horizontal distance.

Vertically

To stretch your drawing vertically:

1. Click **Stretch** in **Modify**.

2. Click **Vertical** in **Stretch**.

3. Select the drawing elements to be stretched.

4. Enter a vertical distance. Alternatively, specify a reference point and a destination point for the vertical distance.

📝 Note

You can interrupt the command by clicking Length in Measure, if the distance you need can be measured from existing features on the drawing.

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 70. Creating an Isometric View on page 169.

Figure 70. Creating an Isometric View
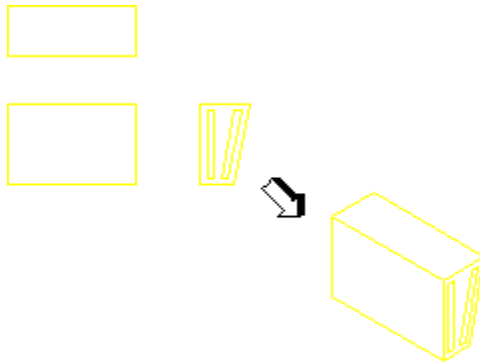
1. Click **Isometric** in **Modify**.

2. Select the front view by enclosing it in a box.

3. Click a reference point on the front view.

4. Specify the destination for the front view reference point.

5. Select the side view by enclosing it in a box.

6. Click a reference point on the side view.

7. Specify the destination for the side view reference point.

8. Select the top view by enclosing it in a box.

9. Click a reference point on the top view.

10. Specify the destination for the top view reference point.

Trimming Elements

Overlapping elements can easily be trimmed or extended with the **Trim/Expand** command. 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).

Trimming One Element

The **Trim One Element** option trims one overlapping element.

To trim one overlapping element:

1. Click **Trim/Expand** in **Modify**.

2. Click **Trim One Element** in **Trim/Expand**.

3. Click the first element (point **1**).

4. Click the intersecting element (point **2**).

Figure 71. Trimming One Element
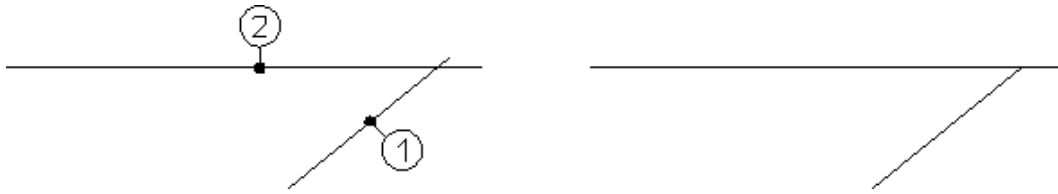
Trimming Two Elements

The **Trim Two Elements** option trims two overlapping elements.

To trim two overlapping elements:

1. Click **Trim/Expand** in **Modify**.
2. Click **Trim Two Elements** in **Trim/Expand**.
3. Click the first element (point **1**).
4. Click the intersecting element (point **2**).

Figure 72. Trimming Two Elements
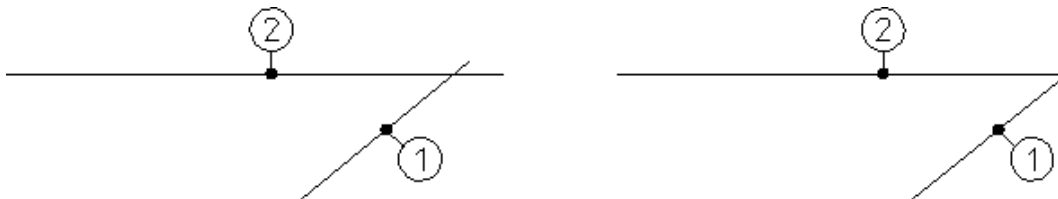
Chain Trimming

Use the option **Chain** if you are trimming a series of elements. After you click the second element, Creo Elements/Direct Drafting trims the line(s) and tells you to click the next element:

1. Click **Trim/Expand** in **Modify**.
2. Click **Chain** in **Trim/Expand**.
3. Click point **1**.
4. Click point **2**.
5. Click points **3, 4, 5, 6**.

Figure 73. Using the Chain Option

Creo Elements/Direct Drafting User's Guide: Windows User Interface
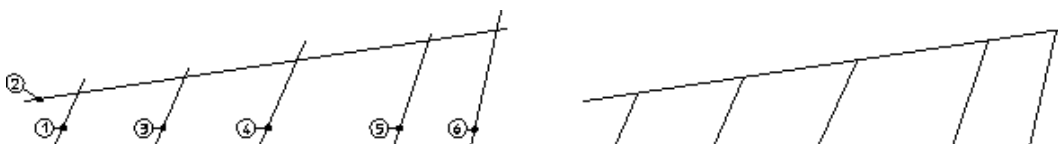
📝 **Note**

You cannot trim elements such as closed splines, circles, and closed polygons because split points are not defined.

Cutting Out the Middle Section

The **Cut Middle** option cuts the middle section out of intersecting elements.

To cut out the middle of an element:

1. Click **Trim/Expand** in **Modify**.
2. Click **Cut Middle** in **Trim/Expand**.
3. Click element (point **1**).
4. Click element (point **2**).
5. Click element (point **3**).

The option **Chain** takes the last intersecting elements as the new intersecting elements.

Figure 74. Cutting Out the Middle Section
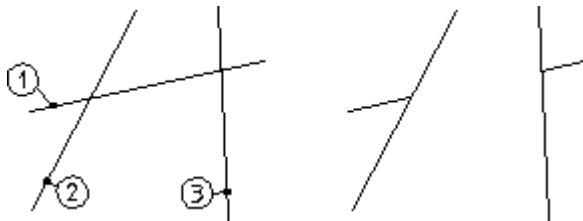
Creating a Contour

The **Contour** command creates a smooth contour by deleting overlapping elements.

To create a contour from a number of overlapping lines:

1. Click **Contour** in **Modify**.
2. Enclose the elements in a box.

Figure 75. Creating a Contour
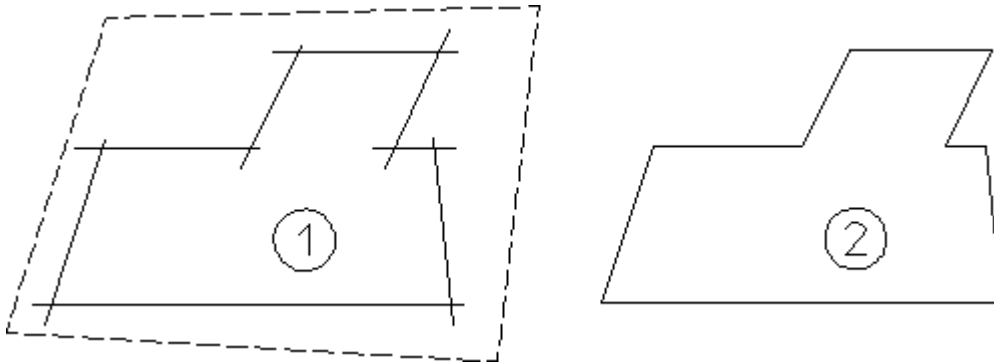
Splitting Elements

It is often useful to split an element (line, circle, and so on) into 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, switch on **Splitting** in the **Setup** menu to automatically split elements at their intersection points. This technique keeps contours clean, which is important for automatic hatching.

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 76. Splitting Elements on page 172 shows the principle of splitting elements.

Figure 76. Splitting Elements
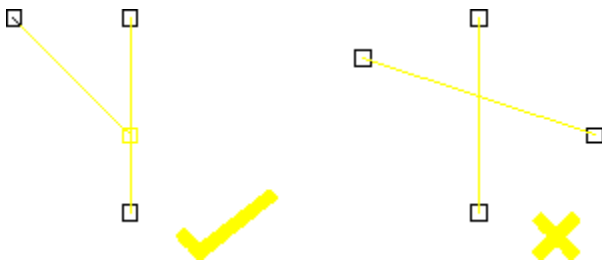
> **Note**
>
> If you are deleting an element whose end point lies on other elements and Splitting is active, Creo Elements/Direct Drafting will attempt to merge the remaining elements automatically into one.

Splitting Single Element/Elements in a Box

To split a single element:

1. Click **Split** in **Modify**.

2. Click the element to be split. Creo Elements/Direct Drafting highlights the chosen element.

3. Click the point where the element should be split.

Alternatively, to split several elements automatically at their intersection points:

1. Click **Split** in **Modify**.

2. Box the elements to be split (be sure to fully enclose the elements in the box).

To verify that all elements have been split, switch on **Vertex** in the **Show** menu. All existing points will be highlighted.

Merging Elements

To merge elements that have been split:

1. Click **Merge** in **Modify**.

2. 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 switch on **Vertex** in the **Show** menu.

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 several tools that can do limited cleaning of parts. The **Clean** commands are:

- **Points**
- **Duplicate**
- **Stacked**
- **Staggered**
- **Gaps**
- **Show 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, **Points** 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. **Points** is most useful for removing small gaps between elements. To use **Points**:

1. Click **Modify ▸ Clean ▸ Points**.
2. Enter a distance tolerance (current length units).
3. 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. The command 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, **Duplicate** should be used after **Points** to clean up duplicate elements that may result from merging model points. To use **Duplicate**:

1. Click **Modify ▸ Clean ▸ Duplicate**.
2. Enter a distance tolerance (current length units) for the removal of duplicate elements.
3. 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 ▸ Clean ▸ Stacked**.

2. 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 ▸ Clean ▸ Staggered**.

2. Enter a distance tolerance (current length units).

3. 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/Expand** functionality 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 ▸ Clean ▸ Gaps**.

2. Draw a selection box around the elements between which there are gaps that you want to close. Note that, if you use the **Global** selection mode, you may unintentionally close "correct" gaps between elements.

3. Enter a distance tolerance (in current length units). 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.

4. Be sure to check the "trivial gaps" preview for plausibility.

5. Click **Confirm** to close any "trivial" gap(s).

Figure 77. Closing Trivial Gaps on page 176 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 77. Closing Trivial Gaps
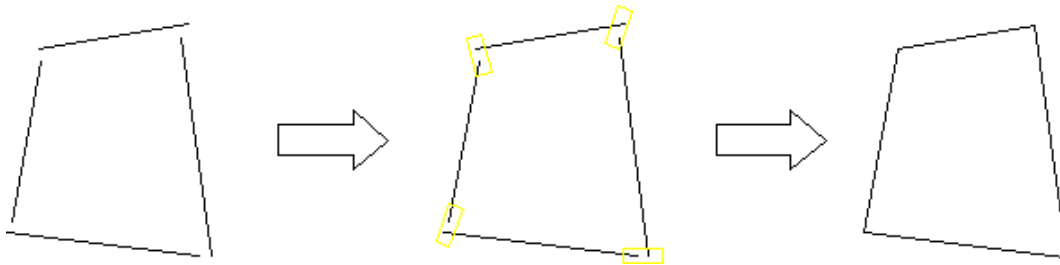
Show Gaps

Show Gaps can be used to box "non-trivial" gaps between two or more elements.

To mark "non-trivial" gaps:

1. Click **Modify ▸ Clean ▸ Show Gaps**.

2. Select a box color for "trivial" gaps.

3. Select a box color for "non-trivial" gaps.

4. Draw a selection box around the elements between which there are or may be gaps.

5. Enter a distance tolerance (current length units). 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.

6. Enter the number of "non-trivial" gaps.

Optionally, by incrementing this number, you can display one "non-trivial" gaps after the other for further examination. For example, you may want to enter **1** to box only a single "non-trivial" gap. Then enter **2** to box the next one, and so forth.

Optionally, enter the total number of "non-trivial" gaps to box all of these gaps at the same time.

7. If the **Show Gaps** command also indicates "trivial" gaps, you can optionally close these by clicking **Confirm**.

shows an example of a geometry that includes three "non-trivial" gaps.

Figure 78. Showing Non-Trivial Gaps
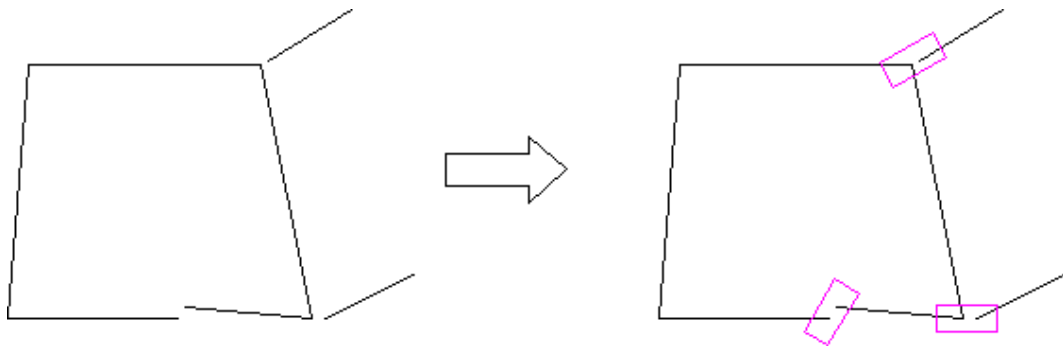
💡 **Tip**

Tip: To clean up most common problems in a part, run the Clean commands in the following sequence:

Points	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:

Enter `help modify` at the Creo Elements/Direct Drafting command line.

The screen will clear and Creo Elements/Direct Drafting will display further information about modifying your drawing.

8

Dimensioning a Drawing

This chapter shows 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

shows several examples of dimension text with the fields indicated.

Figure 79. Dimension Text Examples
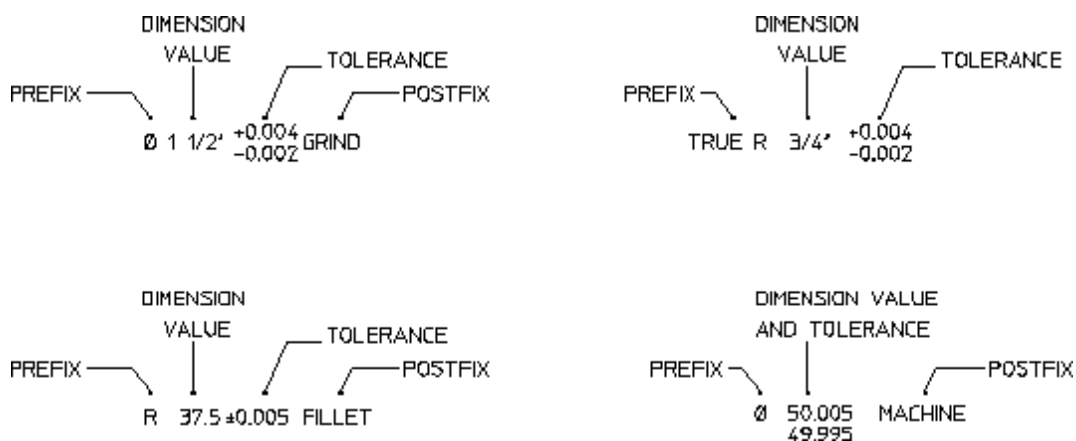
The fields prefix, tolerance, postfix, subfix and superfix are optional.

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 80. Limit Tolerancing on page 181 shows an example of limit tolerancing.

Figure 80. Limit Tolerancing
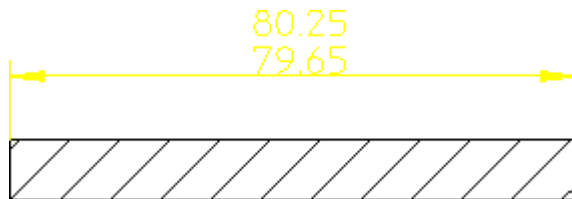
Plus and Minus Tolerances

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 81. Plus and Minus Tolerancing on page 181 shows an example of plus and minus tolerancing.

Figure 81. Plus and Minus Tolerancing
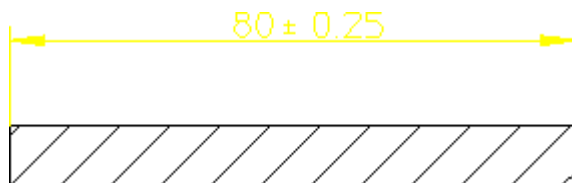
Upper and Lower Tolerances

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 82. Upper and Lower Tolerancing on page 182 shows an example of upper and lower tolerancing.

Figure 82. Upper and Lower Tolerancing
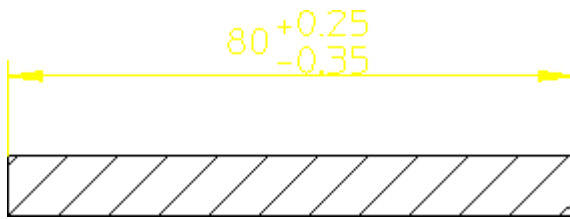
What is a Postfix?

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 83. Dimension Text plus Subfix plus Superfix on page 182 shows an example of a dimensioned part complete with dimension text, subfix and superfix.

Figure 83. Dimension Text plus Subfix plus Superfix
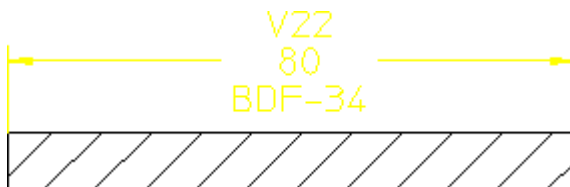
What is a Dimension?

Dimension marks the geometry feature corresponding to the dimension text. Dimension consists of:

- Dimension lines
- Extension lines
- A dimension text frame

Figure 84. Dimension on page 183 shows the three components of a dimension.

Figure 84. Dimension
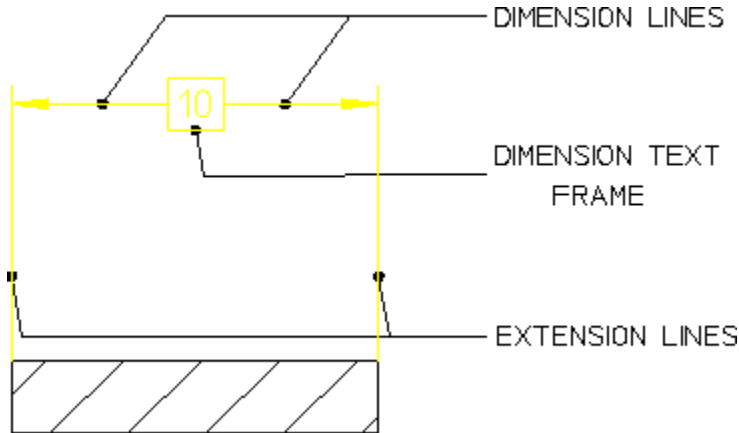
Dimension Lines

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 85. Dimension Line Terminators on page 183 shows dimension line terminators.

Figure 85. 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 120. Dimension Text Frame – Options on page 239).

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 86. Setting Dimension Distances on page 184 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 86. Setting Dimension Distances
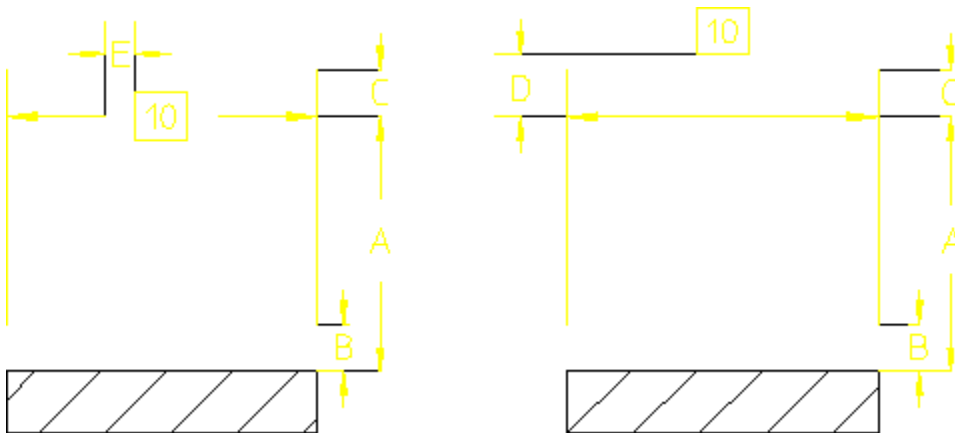
Table 1 on page gives the distance, the corresponding Creo Elements/Direct Drafting function that sets the distance, and a typical value.

Table 1. Setting Dimension Distances

Distance	Description
A	The minimum distance between the dimension line and the geometry feature. Define with the **Minimum Space** function (in **Lines** under **Dimension Settings**). The default minimum distance is 0 mm.
B	The distance between the extension line and the geometry

Distance	Description
	feature. Define with the **Offset Point** function (in **Lines** under **Dimension Settings**). 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** function (in **Lines** under **Dimension Settings**). The default distance is 2 mm.
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** function (in **Text** under **Dimension Settings**). 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** function (in **Line** under **Dimension Settings**). 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 via a screen menu 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 **End** 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 **Dimension**.
2. Click **Linear**.
3. Click **Single**.
4. Click and dimension more than one geometry feature.
5. When you have finished using **Single**, click **End** 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 87. Combining Dimensioning Commands and Functions

To create the dimension shown in Figure 87. Combining Dimensioning Commands and Functions on page 187:

1. Click **Dimension**.
2. Click **Circular**.
3. Click **Diameter**.
4. Click **Centerline**.
5. Click **Settings** in **Dimension**.
6. In **Dimension Settings**, click the **Text Ext** tab.
7. Select **Prefix**, then enter DIA into the **Text Ext** input field.
8. Enter an arbitrary name for the prefix (such as p1) into the entry field above the (originally empty) prefix list and click **Add**.
9. Select **Postfix**, then enter GRIND into the **Text Ext** input field.
10. Click **Upper/Lower** in **Tolerance**, then:

    a. Enter the upper tolerance value, **+0.004**.
    b. Enter the lower tolerance value, **-0.000**.

        Use the same procedure as above to add the tolerance values to your dimension text extensions.

11. Click the circle.
12. 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 System Status Bar for the entries **mm** and **deg**.

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 35.

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 **Linear** in **Dimension**.
    2. Click **Single** in **Linear**.
    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(s).

    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 asks 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 see how geometry-sensing assignment works, try the following:

1. Create a drawing with several different drawing elements.

2. Enter DA_DIM_GEO_SENSE ALL at the command line.

   Creo Elements/Direct Drafting creates dimensioning for all elements in the drawing.

3. To manually adjust the positions of the newly-created dimensions, you can use the DA_MOVE_DIMENSION command.

- 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

Dimension. The current filter can be enabled or disabled via the **On** check box in **Filter** under **Dimension Settings**.

- 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 System Prompt Line:

Warning; Detail part is not active. Dimension value reflects the drawing scale.

To make a detail part active:

1. Click **Parts**.
2. Click **Edit**.
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 405.

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 from the **Standard** menu at the bottom of the Creo Elements/Direct Drafting window. Creo Elements/Direct Drafting displays the current dimensioning standard in the **Standard** display field.

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 commands are provided in the **Modify** menu in **Dimension**.

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 via **Dimension Settings**. 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
- DIN
- ISO
- JIS
- Company (Creo Elements/Direct Drafting default)

To select a dimensioning standard:

1. Click on the selection arrow of the **Dimension Standard** list box at the bottom of the Creo Elements/Direct Drafting window to list the dimension standards available for selection.

2. Select 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 **Automatic** check box in **Options** under **Dimension Settings** or clear **Automatic** 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 226
- Dimension Text Settings on page 236
- Predefining Prefixes, Postfixes, Superfixes, Subfixes, and Tolerances on page 241
- Dimension Unit Settings on page 246
- Selection Filtering on page 250
- Dimension Options on page 255

Single Dimensioning

This section shows how to dimension single linear geometry features.

The following options are available to define the orientation of the single dimension relative to a geometry feature:

Parallel	Parallel to a line drawn between two dimension points.
Horizontal	In a horizontal plane.
Vertical	In a vertical plane.
Perpend to	Perpendicular to a reference line on your drawing.
Parallel to	Parallel to a reference line on your drawing.

To dimension single linear geometry features:

1. Click **Linear** in **Dimensions**.

2. Click **Single** in **Linear**.

3. Click an orientation option; the default is **Parallel**.

4. Complete the selection sequence required by the option. All options 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.

5. If you switched **Automatic** dimension placement off in **Dimension Settings**, 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(s). When multiple elements are selected, the trace shows the proposed dimensions for all of them at the same time.

   If you switched **Automatic** dimension placement on in **Options** under **Dimension Settings**, Creo Elements/Direct Drafting positions the new dimensions automatically. **Automatic** placement only applies when you have selected multiple elements for dimensioning.

6. Click **End** to end this command.

Example

Figure 88. Single Dimensioning – Options on page 194 shows the result of using each single dimensioning option.

Figure 88. Single Dimensioning – Options
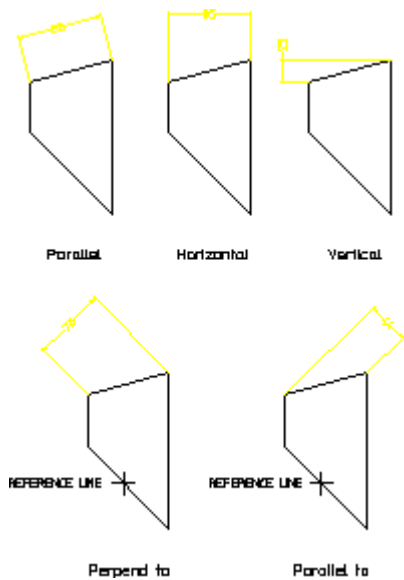
Parallel Dimensions

This is the default option. To dimension along an inclined plane:

1. Click **Linear** in **Dimensions**.

2. Click **Single** in **Linear**.

3. Click **Parallel** in **Single**.

4. Click the dimension points or elements to be dimensioned.

5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**,, click the desired location for the dimension text(s), using the dimension trace for guidance.

Horizontal Dimensions

To produce dimensions in the horizontal plane:

1. Click **Linear** in **Dimensions**.

2. Click **Single** in **Linear**.

3. Click **Horizontal** in **Single**.

4. Click the dimension points or elements to be dimensioned.

5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, specify the desired location for the dimension text(s), using the dimension trace for guidance.

Vertical Dimensions

To produce dimensions in the vertical plane:

1. Click **Linear** in **Dimensions**.

2. Click **Single** in **Linear**.

3. Click **Vertical** in **Single**.

4. Click the dimension points or elements to be dimensioned.

5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

Perpendicular Dimensions

To produce dimensions perpendicular to a reference line:

1. Click **Linear** in **Dimensions**.

2. Click **Single** in **Linear**.

3. Click **Perpend to** in **Single**.

4. Click a reference line. You can identify any line on the drawing as the reference line.

5. Click the dimension points or elements to be dimensioned.

6. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

Parallel to a Reference Line

To produce dimensions Parallel to a reference line:

1. Click **Linear** in **Dimensions**.

2. Click **Single** in **Linear**.

3. Click **Parallel to** in **Single**.

4. Click a reference line. You can identify any line on the drawing as the reference line.

5. Click the dimension points or elements to be dimensioned.

6. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

Symmetry-Style Dimensions

The commands **Symmetry Single** and **Symmetry Long** let you specify symmetry-style dimensions. Figure 89. Symmetry-Style Dimensioning on page 196 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 89. Symmetry-Style Dimensioning
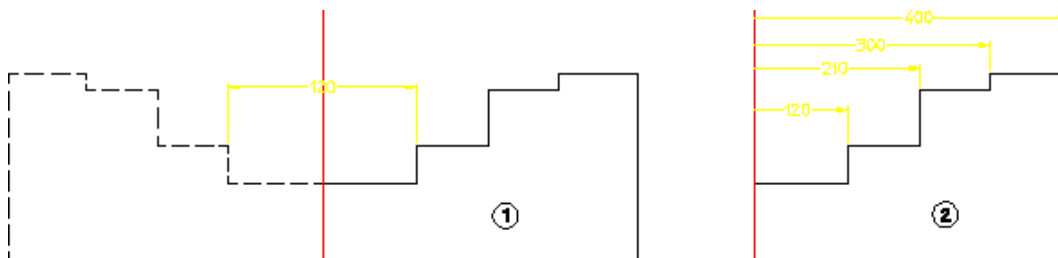
To produce a (single) symmetric dimension:

1. Click **Linear** in **Dimensions**.

2. Click **Symmetry Single** in **Linear**.

3. Click the first dimension point.

4. Click the second dimension point.

To produce a (datum) symmetric dimension:

1. Click **Linear** in **Dimensions**.

2. Click **Symmetry Long** in **Linear**.

3. Click the first dimension point.

4. Click the second dimension point.

5. 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 90. Dimensions and Centerlines
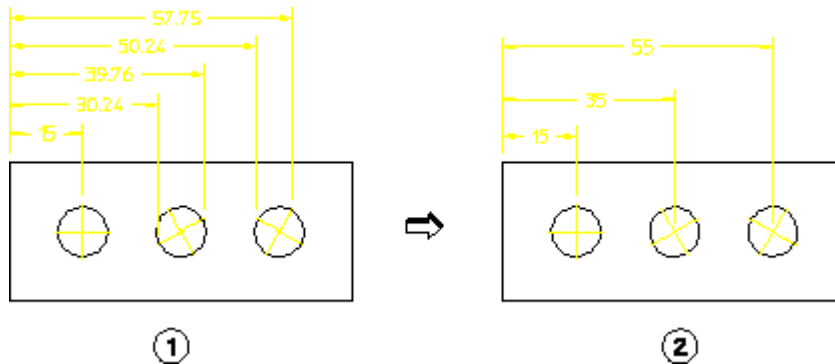
Modify a symmetry or center line

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.

Change the Handle Settings in **Setup ▸ Edit Environment ▸ System ▸ Handles** tab.

Chain Dimensioning

This section shows how to dimension groups of linear geometry features.

The following options are available to define the orientation of chain dimensions relative to geometry features:

Parallel	Parallel to a line drawn between two dimension points.
Horizontal	In a horizontal plane.
Vertical	In a vertical plane.
Perpend to	Perpendicular to a reference line on your drawing.
Parallel to	Parallel to a reference line on your drawing.

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.

To draw chained dimensions:

1. Click **Linear** in **Dimensions**.

2. Click **Chain** in **Linear**.

3. Click an orientation option (the default is **Parallel**).

4. Complete the selection sequence required by the option.

   All options allow you 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. Click 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.

5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, position the dimensions on your drawing. 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(s). When multiple elements are selected, they become the first members of the chain and the trace shows the proposed dimensions for all of them at the same time.

   If you switched **Automatic** dimension placement on in **Options** under **Dimension Settings**, Creo Elements/Direct Drafting automatically positions the dimensions. **Automatic** dimension placement only applies when you have selected multiple elements for dimensioning.

6. To continue chain dimensioning, click the next point in the dimension sequence.

7. Click **End** to end this command.

Figure 91. Chain Dimensioning on page 199 shows examples of chain dimensioning using each option.

Figure 91. Chain Dimensioning
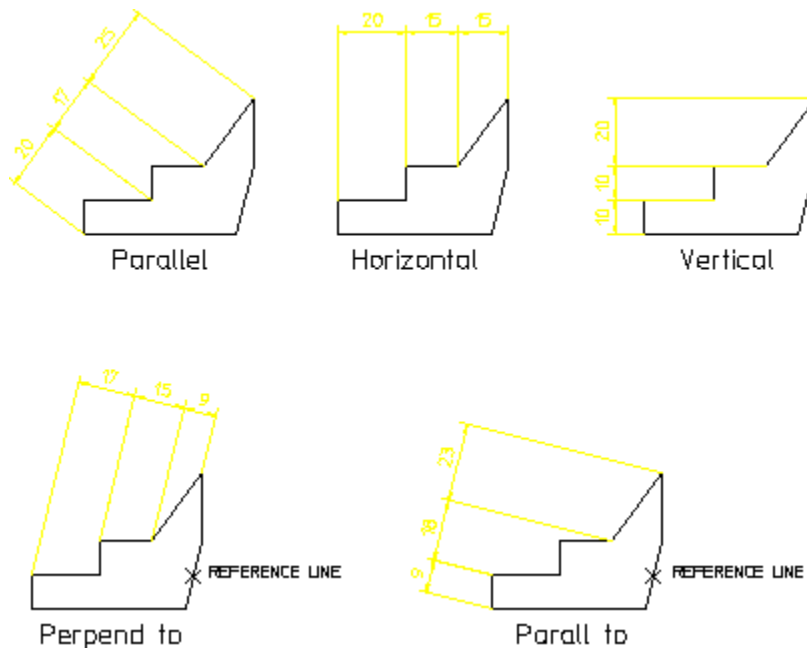
Parallel Dimensions

To chain dimensions along an inclined plane:

1. Click **Linear** in **Dimensions**.

2. Click **Chain** in **Linear**.

3. Click **Parallel** in **Chain**.

4. Click the first two dimension points. These define the angle for this and subsequent chain dimensions.

5. If your initial selection contains multiple drawing elements, you are prompted to identify one of them to provide the reference angle for the chain.

6. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, in **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

7. To continue the chain along the inclined plane, select additional linear elements.

Horizontal Dimensions

To chain dimensions in the horizontal plane:

1. Click **Linear** in **Dimensions**.

2. Click **Chain** in **Linear**.

3. Click **Horizontal** in **Chain**.

4. Click the dimension points or elements to be dimensioned. If your selection contains multiple drawing elements, they become the first members in the chain.

5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, in **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

6. To continue the chain, select additional linear elements.

Vertical Dimensions

To chain dimensions in the vertical plane:

1. Click **Linear** in **Dimensions**.

2. Click **Chain** in **Linear**.

3. Click **Vertical** in **Chain**.

4. Click the dimension points or elements to be dimensioned. If your selection contains multiple drawing elements, they become the first members in the chain.

5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

6. To continue the chain, select additional linear elements.

Perpendicular Dimensions

To create a dimension chain perpendicular to a given line:

1. Click **Linear** in **Dimensions**.

2. Click **Chain** in **Linear**.

3. Click **Perpend to** in **Chain**.

4. Click a reference line. You can use any line on the drawing as the reference line.

5. Click the dimension points or elements to be dimensioned. If your selection contains multiple drawing elements, they become the first members in the chain.

6. If **Manual** placement was selected, click the desired location for the dimension text(s), using the dimension trace for guidance.

7. To continue the chain, select additional linear elements.

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Parallel to a Reference Line

To create a dimension chain parallel to a reference line:

1. Click **Linear** in **Dimensions**.
2. Click **Chain** in **Linear**.
3. Click **Parallel to** in **Chain**.
4. Identify a reference line. You can identify any line on the drawing as the reference line.
5. Click the dimension points or elements to be dimensioned. If your selection contains multiple drawing elements, they become the first members in the chain.
6. If **Manual** placement was selected, click the desired location for the dimension text(s), using the dimension trace for guidance.
7. To continue the chain, select additional linear elements.

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.

The three datum dimension commands are:

- **Datum Long** – creates dimensions using long base lines.
- **Datum Short** – creates dimensions using short base lines.
- **Coordinate** – creates dimensions without dimension lines. Useful if the geometry of the part will cause the dimension lines to be too close together.

Creating Datum Dimensions

To create a datum dimension:

1. Click **Linear** in **Dimensions**.
2. In **Linear**, click the applicable datum dimension command (**Datum Long** or **Datum Short** or **Coordinate**).
3. Click an orientation option (the default is **Parallel**).
4. Complete the selection sequence required by the option. All options allow you apply dimensioning to a single drawing element or to a group of elements.
5. If you selected **Manual** placement, move the dimension trace around your drawing until the dimension text is in the desired location.

6. If multiple drawing elements are selected, the trace shows all proposed dimensions at the same time. Click the desired text location and Creo Elements/Direct Drafting draws the dimension(s).

   If you switched **Automatic** dimension placement on in **Options** under **Dimension Settings**, all the dimensions are positioned automatically and the trace is not used. **Automatic** dimension placement only applies when you have selected multiple elements for dimensioning.

7. To continue datum dimensioning, click the next dimension point.

8. Click **End** to end this command.

The orientation options are:

Parallel	Parallel to a line drawn between two dimension points.
Horizontal	In a horizontal plane.
Vertical	In a vertical plane.
Perpend to	Perpendicular to a reference line on your drawing.
Parallel to	Parallel to a reference line on your drawing.

Datum Dimension Examples

Example

Figure 92. Rectangular Plate – Dimensioned with Datum Long on page 202 shows a rectangular plate dimensioned using **Datum Long** and the **Parallel to** orientation option.
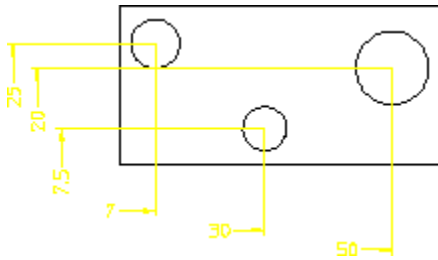
Figure 92. Rectangular Plate – Dimensioned with Datum Long
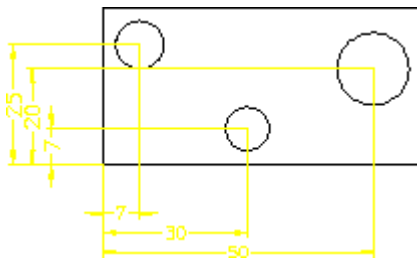
Example

Figure 93. Rectangular Plate – Dimensioned with Datum Short on page 203 shows a rectangular plate dimensioned using **Datum Short** with **Parallel** and **Vertical** orientation options.

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Figure 93. Rectangular Plate – Dimensioned with Datum Short

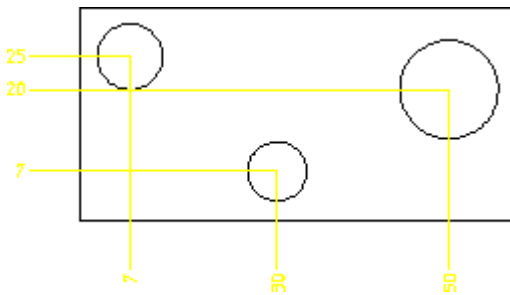## Example

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 94. Rectangular Plate – Dimensioned with Coordinate on page 203 shows the rectangular plate:

Figure 94. Rectangular Plate – Dimensioned with Coordinate

To add the dimensions measured along the vertical (from datum point):

1. Make sure that the **Vertex** option is switched on in the **Show** menu.
2. Click **Linear** in **Dimensions**.
3. Click **Coordinate** in **Linear**.
4. Click **Vertical** in **Coordinate**.
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** in **Linear**.
10. Click **Horizontal** in **Coordinate**.
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 in this chapter.

Parallel Dimensions

To datum dimension along an inclined plane:

1. Click **Linear** in **Dimensions**.
2. In **Linear**, click the applicable datum dimension command (**Datum Long** or **Datum Short** or **Coordinate**).
3. Click **Parallel**.
4. Click a single dimension point. This becomes the common datum point.
5. Click one or more additional dimensioning points.
6. If **Manual** placement was selected, click the desired location for the dimension text(s), using the dimension trace for guidance.
7. To add more dimensions relative to the common datum point, select additional drawing elements.

Horizontal Dimensions

You can dimension in the horizontal plane irrespective of the position of the dimension points.

To dimension in the horizontal plane:

1. Click **Linear** in **Dimensions**.
2. In **Linear**, click the applicable datum dimension command (**Datum Long** or **Datum Short** or **Coordinate**).
3. Click **Horizontal**.
4. Click a single dimension point. This becomes the common datum point.
5. Click one or more additional dimensioning points.
6. If you switched **Automatic** dimension placement off in **Options** under

Dimension Settings, click the desired location for the dimension text(s), using the dimension trace for guidance.

7. To add more dimensions relative to the common datum point, select additional drawing elements.

Vertical Dimensions

To dimension in the vertical plane:

1. Click **Linear** in **Dimensions**.

2. In **Linear**, click the applicable datum dimension command (**Datum Long** or **Datum Short** or **Coordinate**).

3. Click **Vertical**.

4. Click a single dimension point. This becomes the common datum point.

5. Select one or more additional dimensioning points.

6. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

7. To add more dimensions relative to the common datum point, select additional drawing elements.

Perpendicular Dimensions

To create a perpendicular datum dimension:

1. Click **Linear** in **Dimensions**.

2. In **Linear**, click the applicable datum dimension command (**Datum Long** or **Datum Short** or **Coordinate**).

3. Click **Perpend to**.

4. Click a reference line. You can use any line on the drawing as the reference line.

5. Click a single dimension point. This becomes the common datum point.

6. Select one or more additional dimensioning points.

7. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

8. To add more dimensions relative to the common datum point, select additional drawing elements.

Parallel to a Reference Line

To create a datum dimension parallel to a reference line:

1. Click **Linear** in **Dimensions**.

2. In **Linear**, click the applicable datum dimension command (**Datum Long** or **Datum Short** or **Coordinate**).

3. Click **Parall to**.

4. Click a reference line. You can use any line on the drawing as the reference line.

5. Click a single dimension point. This becomes the common datum point.

6. Select one or more additional dimensioning points.

7. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

8. To add more dimensions relative to the common datum point, select additional drawing elements.

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 **Circular** in **Dimension**.

2. Click **Radius** in **Circular**.

3. Click **Centerline** or **No Centerline** in **Radius**.

- Use **Centerline** to dimension the radius of a feature with the dimension line drawn from the feature center.

- Use **No Centerline** 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 95. Dimensioning Radii with and without a Centerline

4. Select one or more arc, circle, or fillet features.

5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

6. Click **End** to end this command.

📝 **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.

The following options are available to define the orientation of the tangential dimensions relative to geometry features:
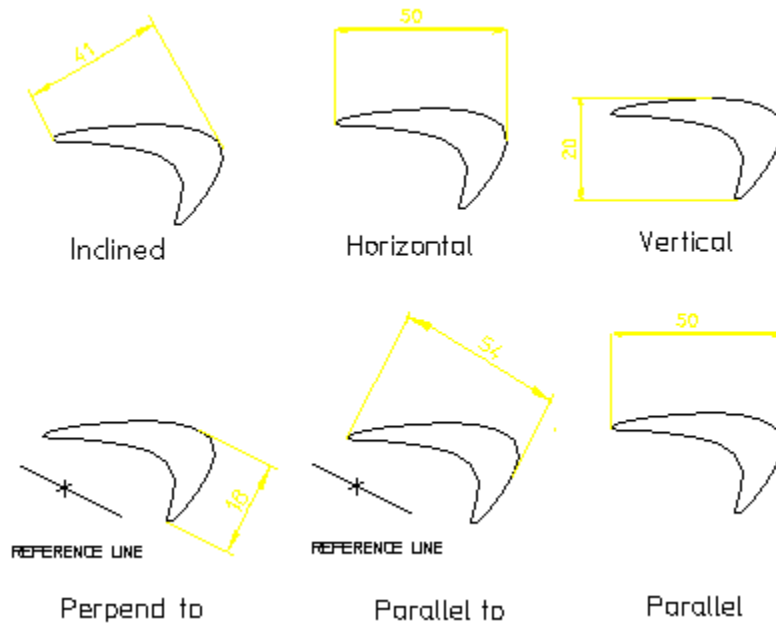
Inclined	In an inclined plane.
Horizontal	In a horizontal plane.
Vertical	In a vertical plane.
Perpend to	Perpendicular to a reference line on your drawing.
Parallel to	Parallel to a reference line on your drawing.
Parallel	Parallel to an imaginary line which passes through the selected extreme points on the geometry.

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. Tangential Dimensioning

Inclined Dimensions

To create tangential dimensions in an inclined plane:

1. Click **Circular** in **Dimension**.

2. Click **Tangential** in **Circular**.

3. Click **Inclined** in **Tangential**.

4. Select the two elements to be dimensioned. The dimension appears at the cursor.

5. Type the inclination angle of the dimension in the `User Input Line` or move the cursor to define the angle.

6. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, select the desired location for the dimension text(s), using the dimension trace for guidance.

7. Click **End** to end this command.

📝 **Note**

You can specify any value of the inclination angle, Creo Elements/Direct Drafting converts the angle between 360 and -360 degrees.

Horizontal Dimensions

To create tangential dimensions in the horizontal plane:

1. Click **Circular** in **Dimension**.
2. Click **Tangential** in **Circular**.
3. Click **Horizontal** in **Tangential**.
4. Select the two elements to be dimensioned. The dimension appears at the cursor.
5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, select the desired location for the dimension text(s), using the dimension trace for guidance.
6. Click **End** to end this command.

Vertical Dimensions

To create tangential dimensions in the vertical plane:

1. Click **Circular** in **Dimension**.
2. Click **Tangential** in **Circular**.
3. Click **Vertical** in **Tangential**.
4. Select the two elements to be dimensioned. The dimension appears at the cursor.
5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, select the desired location for the dimension text(s), using the dimension trace for guidance.
6. Click **End** to end this command.

Perpendicular Dimensions

To create a tangential dimension perpendicular to a given line:

1. Click **Circular** in **Dimension**.
2. Click **Tangential** in **Circular**.
3. Click **Perpend to** in **Circular**.
4. Select a reference line in the viewport. You can select any line on the drawing as the reference line.
5. Select the two elements to be dimensioned. The dimension appears at the cursor.
6. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, select the desired location for the dimension text(s), using the dimension trace for guidance.
7. Click **End** to end this command.

Parallel to a Reference Line

To create a tangential dimension parallel to a reference line:

1. Click **Circular** in **Dimension**.
2. Click **Tangential** in **Circular**.
3. Click **Parallel to** in **Tangential**.
4. Select a reference line in the viewport. You can select any line on the drawing as the reference line.
5. Select the two elements to be dimensioned. The dimension appears at the cursor.
6. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, select the desired location for the dimension text(s), using the dimension trace for guidance.
7. Click **End** to end this command.

Parallel Dimensions

To create a tangential dimension that is parallel to a line through the tangent points:

1. Click **Circular** in **Dimension**.
2. Click **Tangential** in **Circular**.
3. Click **Parallel** in **Tangential**.
4. Select the two elements to be dimensioned. The dimension appears at the cursor.
5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, select the desired location for the dimension text(s), using the dimension trace for guidance.
6. Click **End** to end this command.

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 **Circular** in **Dimension**.

2. Click **Diameter** in **Circular**.

3. Click **Centerline** or **No Centerline** in **Diameter**.

   - Use **Centerline** to dimension the diameter of a feature with the dimension line drawn from the feature center.

   - Use **No Centerline** 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 96. Dimensioning Circles with and without a Centerline

4. Select one or more circle, arc, or fillet features.

5. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

6. Click **End** to end this command.

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:

1. Click **Circular** in **Dimension**.

2. Click **Arc** in **Circular**.

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 with **Select** or by enclosing them in a box. The major arc will be dimensioned for all selected arcs.

4. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

5. Click **End** to end this command.

Figure 97. Dimensioning Arcs – Two Results on page 213 shows two possible results of length and angle dimensions obtained using the **Arc** Dimension UI. Each arc end point is marked in the order in which it has been identified.

Figure 97. Dimensioning Arcs – Two Results
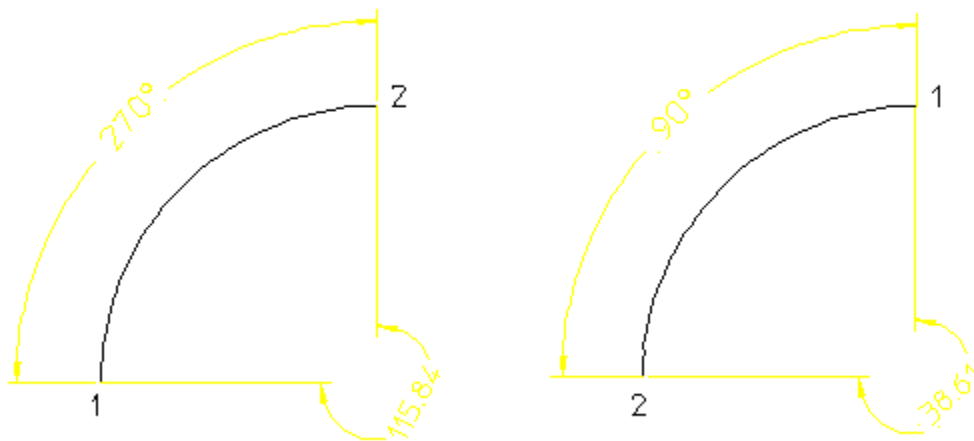
Angle and Chamfer Dimensioning

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** option. Creo Elements/Direct Drafting measures angles in a counterclockwise direction.

To dimension an angle:

1. Click **Angular** in **Dimension**.

2. Click the applicable **Angular** option.

3. Click the two lines that make up the angle.

4. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

5. Click **End** to end this command.

The **Angle** options are:

- **Direct**
- **Opposite**
- **Adjacent -**
- **Adjacent +**

For each of the above options you can achieve two results.

Figure 98. Identifying Angle Lines on page 214 shows how you identify the angle lines in the opposite order to achieve a different result.

Figure 98. Identifying Angle Lines
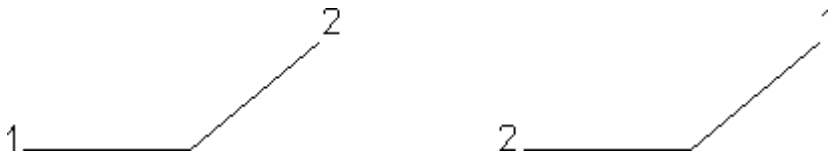
Between Two Lines

Figure 99. Angle Between Two Lines – Two Results on page 214 shows the two possible results.

Figure 99. Angle Between Two Lines – Two Results
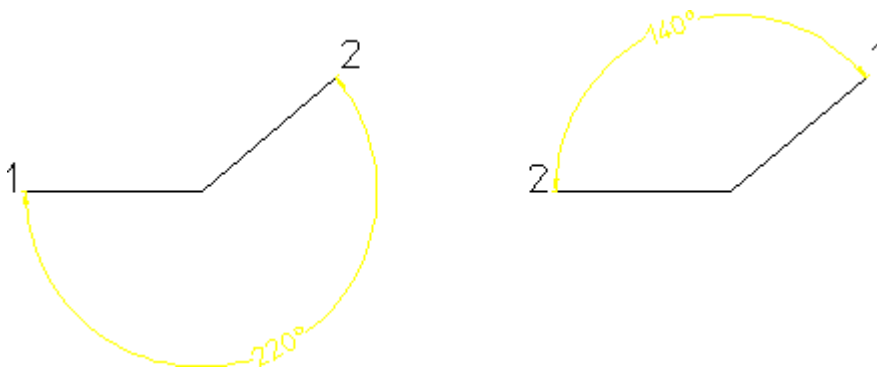
To dimension the angle between two lines:
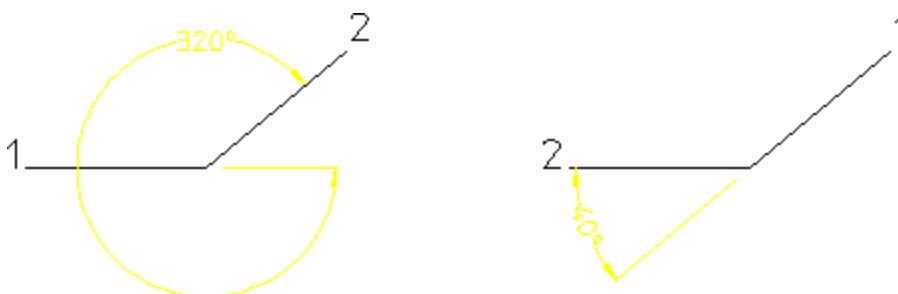
1. Click **Angular** in **Dimension**.

2. Click **Direct** in **Angular**.

3. Identify the two lines making up the angle (the angle dimensioned depends upon which of the two lines you identify first).

4. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

Between Intersections

Figure 100. Angle Between Intersections – Two Results on page 215 shows the two possible results.

Figure 100. Angle Between Intersections – Two Results
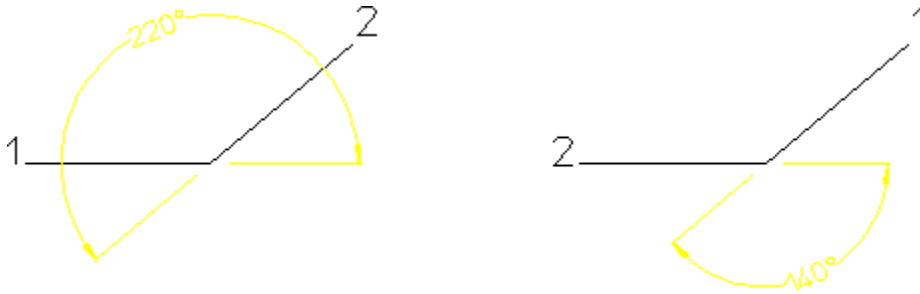
To dimension the angle between the projected intersection of two lines:

1. Click **Angular** in **Dimension**.

2. Click **Opposite** in **Angular**.

3. Click the two lines making up the angle (the angle dimensioned will depend upon which of the two lines you identify first).

4. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

Between First and Second Line

Figure 101. Angle Between First and Second Line – Two Results on page 216 shows the two possible results.

Figure 101. Angle Between First and Second Line – Two Results
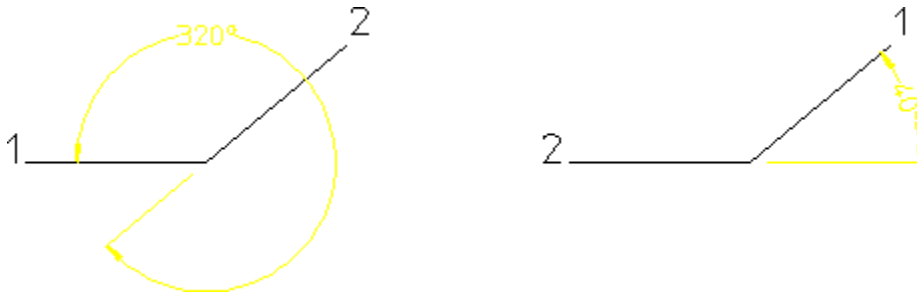
To dimension the angle between the first line you clicked and the projection of the second line:

1. Click **Angular** in **Dimension**.

2. Click **Adjacent -** in **Angular**.

3. Click the two lines making up the angle (the angle dimensioned depends upon which of the two lines you identify first).

4. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

Between Second and First Line

shows the two possible results.

Figure 102. 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 **Angular** in **Dimension**.

2. Click **Adjacent +** in **Angular**.

3. Click the two lines making up the angle (the angle dimensioned depends upon which of the two lines you identify first).

4. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

Dimensioning a Chamfer

Chamfered surfaces can be dimensioned using the **Chamfer** command in the **Dimension** menu. (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 103. Dimensioning a Chamfer
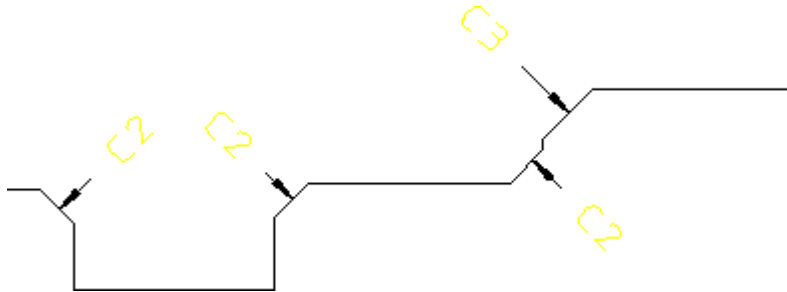
To dimension a chamfer:

1. Click **Linear** in **Dimension**.

2. Click **Chamfer** in **Linear**.

3. Click the chamfer you want to dimension.

4. If you switched **Automatic** dimension placement off in **Options** under **Dimension Settings**, click the desired location for the dimension text(s), using the dimension trace for guidance.

5. Click **End** to end this command.

📝 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 may not be visible after you stretch the chamfer.

Geometry-Sensing Assignment

The **Geo Sense** and **Scan** commands in the **Dimension** menu 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 257) 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 **Geo Sense** in **Dimension**.

2. Click any needed option.

3. Select a group of drawing elements, either with **Select** or by boxing them.

   Creo Elements/Direct Drafting generates and positions dimensions for the selected elements.

4. Click **End** to end this command.

## Dimensioning with Scan

Scan assigns dimensions to preselected types (see Scan Settings on page 258) 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 **Scan** in **Dimension**.

2. Click any needed option.

3. Select a group of drawing elements, which must have been previously given parametric constraints, either with **Select** or by boxing them.

   Creo Elements/Direct Drafting generates dimensions for the selected elements.

4. Click **End** to end this command.

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 **Match** menu which you can access via the **Dimension** menu.

Database Matching Example

1. Draw and dimension the flange shown in Figure 104. Initial Part for Database Matching on page 220.

Figure 104. Initial Part for Database Matching
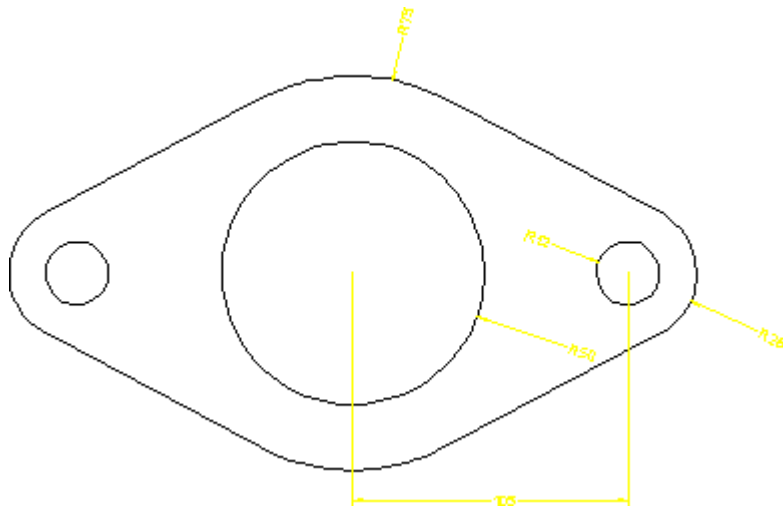
2. Add the flange to the dimension database:

   a. Delete the construction lines from the drawing.

   b. Click **Match** in **Dimension**.

   c. Click **Add** in **Match**.

   d. Enter the name 'Flange 001' when prompted.

   e. 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 **Modify**, make two or three copies of it. You can further modify each copy by scaling the whole part or resizing the holes (Figure 105. New Parts Needing Dimensions on page 221). Do not make modifications that alter the basic topology of the part.

Figure 105. New Parts Needing Dimensions
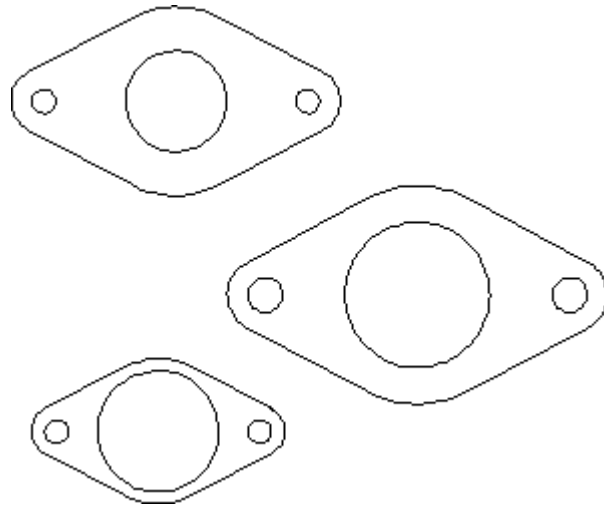
4. Return to **Match** and click the 'Flange 001' entry from the database table.

   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 106. Assigning Dimensions from the Database on page 222). 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 106. Assigning Dimensions from the Database
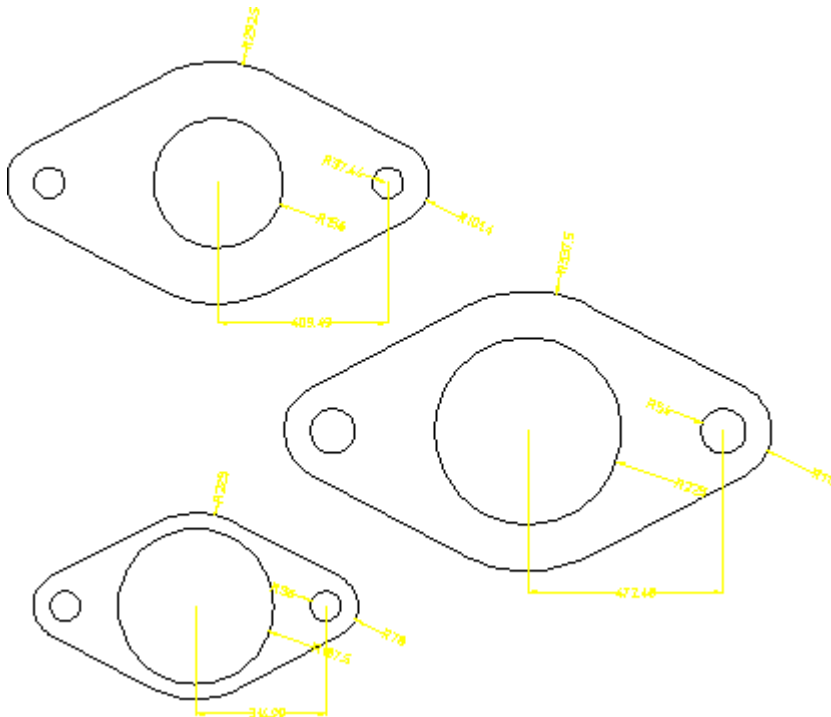
Adding Entries to the Dimension Database

1. Click **Match** in **Dimension**.

2. Click **Add** in **Match**.

3. Enter the name that will identify this entry ("flange 002", for example).

4. 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 **Match** in **Dimension**.

2. Click **Delete** in **Match**.

3. Enter the name of the entry to be deleted, either by clicking its entry in the dimension database table or by typing it on the command line.

Creo Elements/Direct Drafting deletes the selected entry from the dimension database.

Assigning Dimensions from the Database

1. Click **Match** in **Dimension**.

2. 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.

3. Click one end of a reference element on the database entry. Do this by clicking in the style display window.

4. Click the corresponding end of a target element in your drawing.

   Creo Elements/Direct Drafting dimensions your drawing to match the database entry.

5. Click **End** to end this command.

📝 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 View commands 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. Use **Vertex** in **Show** 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 switch on **Vertex** in the **Show** menu 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 the DA_DB_MATCH command. 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, then **Add** it back into the database.

Saving and Loading Dimension Databases

You can use the **Save** and **Open** commands 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 **Match** in **Dimension**.
2. Click **Save** in **Match**.
3. Enter a filename ("Flange_DB", for example).

   Creo Elements/Direct Drafting will store the current database under the given filename.

To load a dimension database from a file, appending to the current database:

1. Click **Match** in **Dimension**.
2. Click **Open** in **Match**.
3. Enter a filename ("Flange2_DB", for example).

Creo Elements/Direct Drafting will load 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 **Match** in **Dimension**.

2. Click **Unload** in **Match**.

   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 241 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 **Text Ext** in **Dimension**.

2. Click **Prefix** in **Text Ext**.

3. Define the prefix.

4. 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. Move the cursor to the prefix entry field.

3. Press the right mouse button and select **Paste** from the pop-up menu 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 245 for instructions.

Dimension Line Settings

The **Line** dialog box under **Dimension Settings** enables you to set attributes of dimension lines:

• Line Settings

• Terminator (Arrow) Settings

These settings will be applied to all subsequent dimensioning that you assign via the **Dimension** menu.

Line Settings

In the **Line** section of the **Line** dialog box under **Dimension Settings**, 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

1. Click the **Line** tab in **Dimension Settings**.

2. Enter the required value (such as **.25**) into the **Width** field.

Dimension a drawing element to see the effect of the changed dimension line width.

**Figure 107. Dimension Line Width**
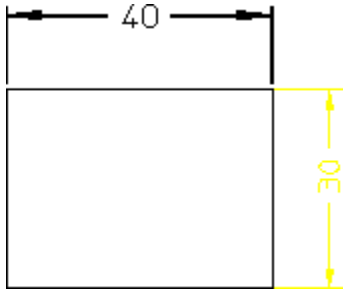
Offset Line

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 the **Line** tab in **Dimension Settings**.

2. Enter a positive value (such as `3) into the **Offset Line** field.

   Creo Elements/Direct Drafting sets this distance for all subsequent dimensions.

shows the dimension offset line distance.

Figure 108. Dimension Offset Line Distance
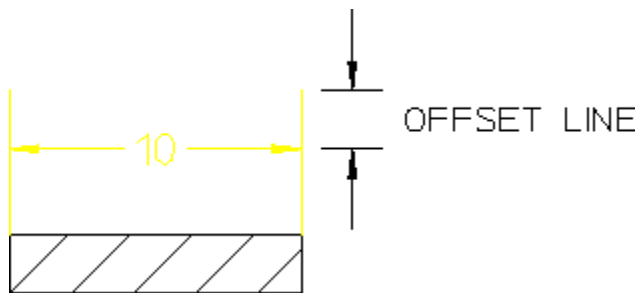
Space

The **Space** field 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 239.

To set this space:

1. Click the **Line** tab in **Dimension Settings**.
2. Enter a positive value (such as 2.3) into the **Space** field.

Figure 109. Distance between Dimension Lines and Text on page 228 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 109. Distance between Dimension Lines and Text
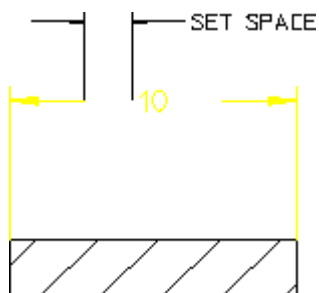
Datum Step

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** commands.

To specify a datum step:

1. Click the **Line** tab in **Dimension Settings**.

2. Enter a new distance value into the **Datum Step** field.

    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 110. Datum Stepping on page 229 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 110. Datum Stepping
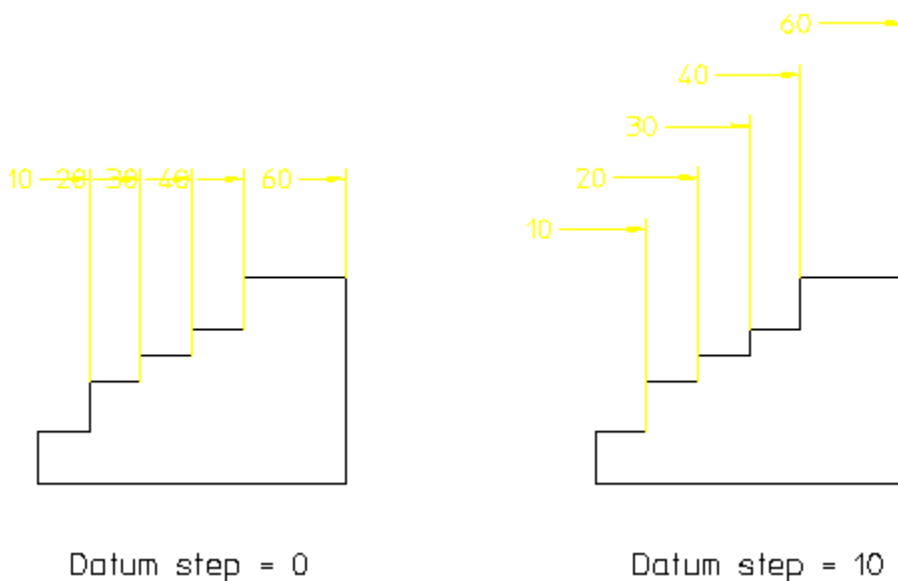
Dimension Line Color

To set the color of dimension lines and dimension extension lines:

1. Click the **Line** tab in **Dimension Settings**.

2. Select a color from the **Color** list. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

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 the **Line** tab in **Dimension Settings**.

2. Enter a positive value (such as **1.5**) into the **Offset Point** field.

Figure 111. Dimension Offset Point on page 230 shows the dimension offset point.

Figure 111. Dimension Offset Point
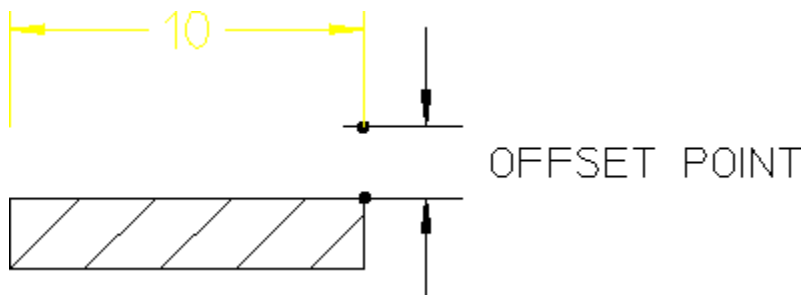
Minimum Space

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 the **Line** tab in **Dimension Settings**.

2. Enter a positive value (such as **10**) into the **Minimum Space** field.

Refer to Figure 112. Locating a Dimension – Closer Than the Minimum Distance on page 231. 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 112. Locating a Dimension – Closer Than the Minimum Distance
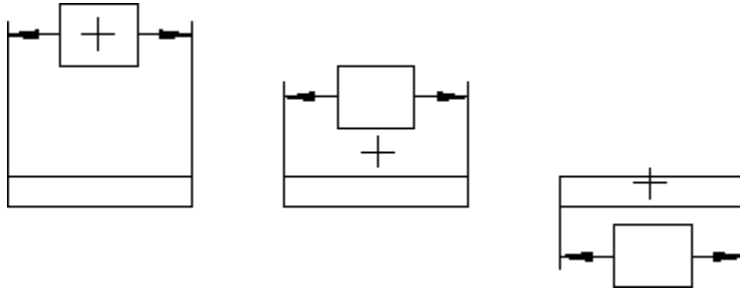
Catch Line

When you want to select a dimension, you can specify which components of the dimension can be selected. This is done by switching the **Catch Line** check box option on or off in **Line** under **Dimension Settings**.

On	A dimension may be selected by clicking one of the dimension components: • Dimension line • Extension line • Dimension text This is the default.
Off	A dimension may be selected by clicking the dimension text only.

Broken/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, switch the **Broken** check box option off in **Line** under **Dimension Settings**.

Figure 113. Dimension Lines Inside Extension Lines on page 232 shows dimension lines drawn inside the extension lines and dimension text outside.

Figure 113. Dimension Lines Inside Extension Lines
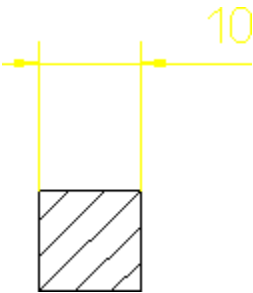
Drawing Dimension Lines Outside Extension Lines

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, switch the **Broken** check box option on in **Line** under **Dimension Settings**.

Figure 114. Dimension Lines and Text Outside Extension Lines on page 232 shows how dimension lines and text are drawn outside the extension lines.

Figure 114. Dimension Lines and Text Outside Extension Lines
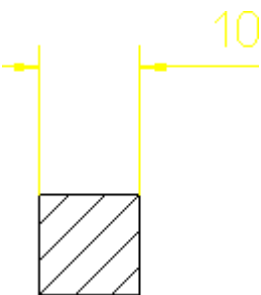
Catch Range

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 115. Dimension Catch Range on page 233 shows the catch range. Any point you indicate within the catch range results in Creo Elements/Direct Drafting centering the dimension text.

Figure 115. Dimension Catch Range
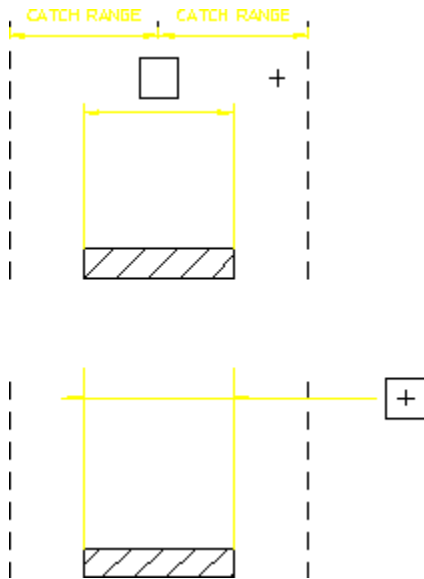
Terminator Settings

In the **Arrow** section of the **Line** dialog box under **Dimension Settings**, you can set the type, size, and fill attributes for all subsequent dimension line terminators.

Normally, the terminator is an arrowhead.

Figure 116. Dimension Line Terminators on page 233 shows the five possible dimension line terminators:

Figure 116. Dimension Line Terminators
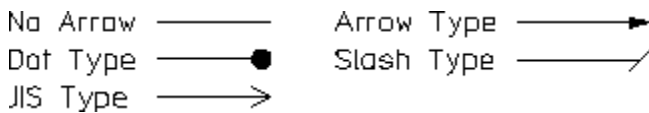
To set the dimension line terminator:

1. Click the **Line** tab in **Dimension Settings**.

2. Select the required terminator settings in the **Arrow** section.

   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**

- **First**

- **Second**

Both

Both is the default option. Click **Both** and then select the required settings. The settings you select will apply to both dimension line terminators.

First

Click **First** to apply the chosen terminator settings to the first point you click when identifying the dimension start-point.

Second

Click **Second** to apply the chosen terminator settings to the second point you click when identifying the dimension end-point.

Figure 117. Dimension Terminators – First and Second

A	**Both** (arrow type)
B	**First** (dot type)
C	**Second** (dot type)
D	**First** (dot type – notice which is the first point)
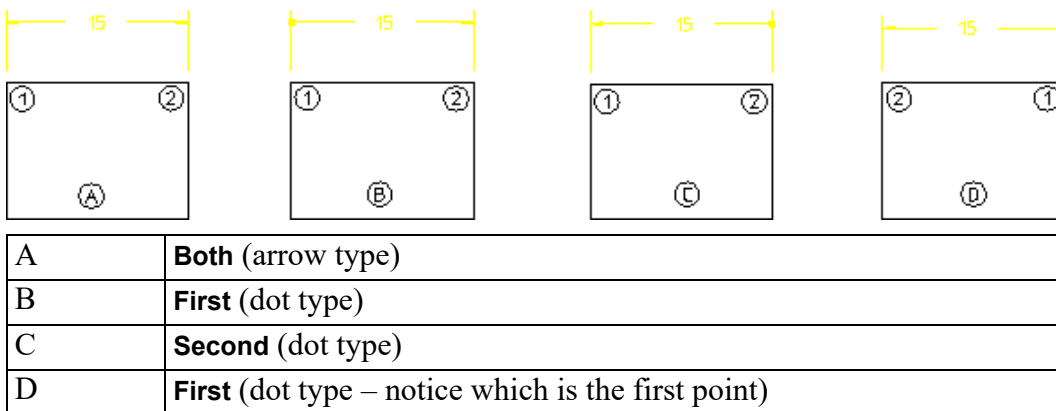
Because you specify a first point and a second point, it makes no difference if the dimensions are horizontal or vertical.

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 the **Line** tab in **Dimension Settings**.
2. Select **Auto** or **Inside** or **Outside** from the **Position** list.

You cannot change the direction of arrows in Broken on page 226 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 the **Line** tab in **Dimension Settings**.
2. Enter a value for the terminator size into the **Size** field.

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 the **Line** tab in **Dimension Settings**.
2. Switch on the **Relative** check box option.

Terminator Fill On/Off

By default, dimension line terminators are filled. Simply click on the **Fill** check box to switch the fill mode on or off in **Line** under **Dimension Settings**.

Example

Figure 118. Unfilled Dot Type Terminators on page 235 shows unfilled dot type terminators.

Figure 118. Unfilled Dot Type Terminators
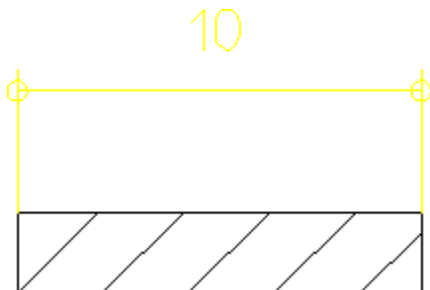
Dimension Text Settings

The **Text** dialog box under **Dimension Settings** enables 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 **Dimension** menu.

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 the **Text** tab in **Dimension Settings**.

2. Select a dimension text component from the **Set** list. You can choose any of the following:

   • **All** (prefix, value, tolerance and postfix)

   • **Lower**

   • **Upper**

   By default, the dimension text settings apply to **all** dimension text components.

3. 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 the **Text** tab in **Dimension Settings**.

2. Select a font from the **Font** list. 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 435.

Dimension Text Color

To set the color of dimension text:

1. Click the **Text** tab in **Dimension Settings**.

2. Select a color from the **Color** list. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

Dimension Text Orientation

The orientation of the dimension text can be one of the following:

Figure 119. Dimension Orientation
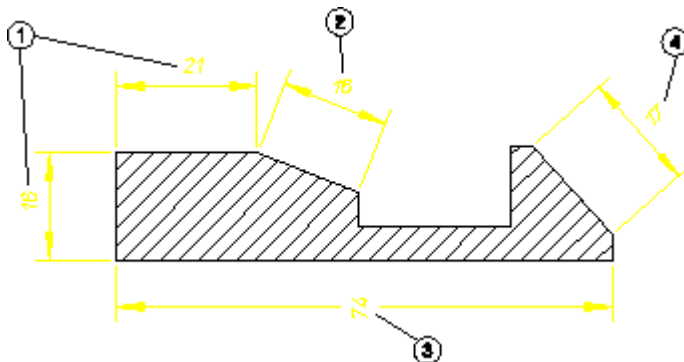
1 Parallel to the dimension line. This is the default.
2 Horizontal (2).
3 Vertical (3).
4 Perpendicular to the dimension line.

To set the orientation of dimension text:

1. Click the **Text** tab in **Dimension Settings**.

2. Select one of the following settings from the **Orientation** list:

   - **Parallel**

   - **Horizontal**

   - **Vertical**

   - **Perpend**

Dimension Text Frame

To set the dimension text frame:

1. Click the **Text** tab in **Dimension Settings**.

2. Select one of the following settings from the **Frame** list:

   - **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 120. Dimension Text Frame – Options on page 239 shows the dimension text frame options.

Figure 120. Dimension Text Frame – Options

Dimension Text Location

The location of dimension can be in one of three positions:

Figure 121. Dimension Location
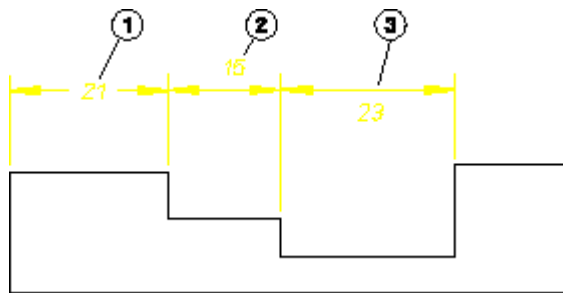
1 On the dimension line.
2 Above the dimension line. This is the default.
3 Below the dimension line.

To set the location of dimension text:

1. Click the **Text** tab in **Dimension Settings**.

2. Select one of the following settings from the **Location** list:

    •   **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 the **Text** tab in **Dimension Settings**.

2. Make sure that the **Relative** check box option is switched off.

3. Specify 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 the **Text** tab in **Dimension Settings**.

2. Make sure that the **Relative** check box option is switched on.

3. Enter the relative dimension text size.

Dimension Text Offset

📋 **Note**

This function only works if you have previously selected either Above or Below from the Location list.

Figure 122. Dimension Text Offset on page 240 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 122. Dimension Text Offset
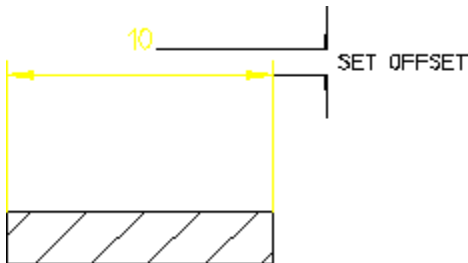
To set the dimension text offset:

1. Click the **Text** tab in **Dimension Settings**.

2. Enter a positive offset into the **Offset Line** field.

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 the **Text** tab in **Dimension Settings**.

2. Enter a new slant angle into the **Slant** field.

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 the **Text** tab in **Dimension Settings**.
2. Enter the new value into the **Ratio** field.

Dimension Text Brackets

Primary and secondary dimension text can be enclosed in brackets [ ] or not. By default, brackets are not on.

To switch dimension text brackets on or off:

1. Click the **Text** tab in **Dimension Settings**.
2. Select either **Upper** or **Lower** in the **Set** list.
3. Click on the **Brackets** check box to toggle this option on or off.

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 the **Text** tab in **Dimension Settings**.
2. Under **Specific Texts**, click or clear **Coordinate Dimension - Leading Zero**.

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 **Dimension Settings**. Once defined, they can easily be selected from a table. Apart from making the process quicker, this method also improves consistency.

The **Text Ext** dialog box under **Dimension Settings** enables you to add, delete, save, open and clear the following:

- **Prefix**
- **Postfix**
- **Superfix**
- **Subfix**
- **Tolerance**

See 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 the **Text Ext** tab in **Dimension Settings**.
2. Select **Prefix** from the **Set** list in the **Extension** section.
3. Enter a name for the prefix (such as p1) into the entry field above the (originally empty) prefix list.
4. Enter the prefix text (such as the following example) into the **Ext Text** field. Examples of prefixes are:

   • DIA (diameter)

   • R (radius)

   • CHAMFER

   • BC (base circle)

   Optionally, you can select a special character to serve as the dimension text extension:

5. Click **Spec Char** to display a graphics list of special text characters. Note that these are based on the `hp_symbols` and `hp_symbols2` fonts.
6. In **Special Characters**, 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 **Text** field in **Set Text Extension**.
8. In the **Ext Text** box, type the basic dimension prefix text string in the **Extension Text** section.
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(s) 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 **Single** in **Dimension**.
2. Click **Prefix**.
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 the **Text Ext** tab in **Dimension Settings**.
2. Select **Prefix** from the **Set** list in the **Extension** section.
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 the **Text Ext** tab in **Dimension Settings**.
2. Select **Prefix** from the **Set** list in the **Extension** section.
3. Click **Save** to display the **Save as** dialog.
4. Specify a filename and select a file type.
5. Click **OK** to save the dimension text extension(s).

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 the **Text Ext** tab in **Dimension Settings**.
2. Select **Prefix** from the **Set** list in the **Extension** section.
3. Click **Open** to display the **Open** dialog.
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. Select **Prefix** from the **Set** list in the **Extension** section.

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** section of the **Text Ext** dialog box under **Dimension Settings** 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 the **Text Ext** tab in **Dimension Settings**.

2. Select a font from the **Font** list. 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 435.

Dimension Text Extension Color

To set the color of dimension text extensions:

1. Click the **Text Ext** tab in **Dimension Settings**.

2. Select a color from the **Color** list. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

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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 the **Text Ext** tab in **Dimension Settings**.
2. Enter the new value into the **Ratio** field.

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 the **Text Ext** tab in **Dimension Settings**.
2. Make sure that the **Relative** check box option is switched off.
3. Specify the absolute dimension text extension size.

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 the **Text Ext** tab in **Dimension Settings**.
2. Make sure that the **Relative** check box option is switched on.
3. Enter the relative dimension text extension size.

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 the **Text Ext** tab in **Dimension Settings**.
2. Enter a new slant angle into the **Slant** field.

Special Characters in Dimension Text Extensions

To add a special character to a dimension text extension:

1. Click the **Text Ext** tab in **Dimension Settings**.
2. Under **Extension**, 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. See Editing a Font on page 305 for further information.

4. In **Special Characters**, 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 the **Text Ext** tab in **Dimension Settings**.

2. Under **Extension Text**, in the **Basic Ext Text** box, type the basic dimension extension text.

3. Click **Ok** to save the basic dimension text extension.

All the dimensions created after setting the basic dimension text extension inherit the text extension.

📋 **Note**

   To change the basic dimension text extension, in the Basic Dimensions Extensions area, click the Change box adjacent to the Prefix or Postfix boxes.

Dimension Unit Settings

The **Units** dialog box under **Dimension Settings** enables 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 via the **Dimension** menu.

To select the dimension text component(s) to which you want to apply the unit settings:

1. Click the **Units** tab in **Dimension Settings**.

2. Select a dimension text component from the **Set** list. You can choose any of the following:

   • **Upper**

   • **Lower**

Upper Dimension

This is the default option. Select **Upper** from the **Set** list in **Units** to set the units for the main dimension text.

Lower Dimension

Select **Lower** from the **Set** list in **Units** to set the units for the lower dimension text (if applicable).

Linear Units

First select the **Upper** or **Lower** dimension text component from the **Set** list in **Units**, then select the linear units for that component from the options available:

- **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

This is the default option. Select **Millimeters** to set the dimension units to millimeters.

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

First select the **Upper** or **Lower** dimension text component from the **Set** list in **Units**, then select the angular units for that component from the options available:

- **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

First select a dimension unit from the **Unit** list in **Display**, then enter an associated number format into the **Decimal Places** field 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, enter:

```
0.01
```

The format string in the **Decimal Places** field 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. Selection filtering commands are located under **Filter** in **Dimension Settings**.

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 turned on or off at any time via the **Filter On** check box under **Filter** in **Dimension Settings**.

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 the **Filter** tab in **Dimension Settings**.

2. Switch on **Filter On** in **Filter**.

3. Select a filter type:

   • Geometry Type

   • Line Slope

   • Linetype

   • Pensize

   • Color

   • Linesize

4. Select the desired 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 will be excluded by this filter.

To set the geometry-type filter:

1. Click the **Filter** tab in **Dimension Settings**.

2. Switch on **Filter On** under **Filter** in **Dimension Settings**.

3. Select the **Geometry Type** filter.

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 the **Filter** tab in **Dimension Settings**.
2. Switch on **Filter On** under **Filter** in **Dimension Settings**.
3. Select the **Line Slope** filter.
4. Switch on **Horizontal** under **Exclude**.
5. Enter the lower bound (in current angular units).
6. Enter 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 switch on **Horizontal** or **Vertical** under **Include**.

Saving from The Line-Slope Filter

1. Click the **Filter** tab in **Dimension Settings**.
2. Select the **Line Slope** filter under **Filter** in **Dimension Settings**.
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 the **Filter** tab in **Dimension Settings**.

2. Select the **Line Slope** filter under **Filter** in **Dimension Settings**.

3. Select 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 the **Filter** tab in **Dimension Settings**.

2. Select the **Pensize** or **Linesize** filter under **Filter** in **Dimension Settings**.

3. Enter the lower bound.

4. Enter 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 the **Filter** tab in **Dimension Settings**.

2. Select the **Pensize** or **Linesize** filter under **Filter** in **Dimension Settings**.

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 the **Filter** tab in **Dimension Settings**.

2. Select the **Color** filter under **Filter** in **Dimension Settings**.

3. Select a color from the **Add Color** list. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

4. Click **Add**.

   Creo Elements/Direct Drafting creates a new entry in the filter table.

5. Click **End** to finish the operation.

Deleting from the Color Filter

1. Click the **Filter** tab in **Dimension Settings**.

2. Select the **Color** filter under **Filter** in **Dimension Settings**.

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 the **Filter** tab under **Dimension Settings**.

2. Click **Reset**.

Saving and Loading Selection Filters

You can use the **Save** and **Load** commands to save selection filters to files and later reload them into Creo Elements/Direct Drafting.

To save the current selection filter:

1. Click the **Filter** tab in **Dimension Settings**.

2. Click **Save**.

3. Enter a filename (`"Select_filter.1"`, for example).

   Creo Elements/Direct Drafting will store the selection filter under the given filename.

To load a new selection filter from a file (overwriting any current filter):

1. Click the **Filter** tab in **Dimension Settings**.

2. Click **Load**.

3. Select a filename (`"Select_filter.1"`, for example) from the list.

   Creo Elements/Direct Drafting will load the named selection filter, replacing any current filter.

Dimension Options

The **Options** dialog box in **Dimension Settings** enables you to set dimension location options as well as well as automatic dimensioning settings. These settings will be applied to all subsequent dimensioning that you assign via the **Dimension** menu.

Dimension Location Settings

In the **Dimension Location** section of the **Options** dialog box, you can set the following:

•   Automatic Dimension Positioning

•   Auto-Placement Position

•   Dimension Strategy

Automatic Dimension Positioning

You can switch on the **Automatic** check box option to automatically place dimensions rather than use the dimension trace (**Automatic** switched off).

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** list in **Options** under **Dimension Settings**, you can ask Creo Elements/Direct Drafting to place dimensions either above and to the right or below and to the left (see Figure 123. Automatic Dimension Positioning — Above (a.) and Below (b.) on page 256).

Figure 123. Automatic Dimension Positioning — Above (a.) and Below (b.)
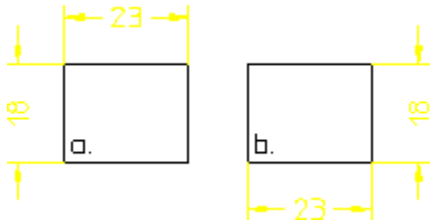
To set the auto-placement position:

1. Click the **Options** tab in **Dimension Settings**.

2. Tick the **Automatic** check box under **Dimension Location**.

3. Select **Above** or **Below** from the **Position** list.

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 **Dimension Settings**.

2. Tick the **Automatic** check box under **Dimension Location**.

3. Select **Off**, **Geo**, or **Geo+Dim** from the **Strategy** list box.

   • **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 boxes at the bottom of the **Options** dialog box in **Dimension Settings**:

• **Delete Dimension Check**

   Sets the check option for the **Dimension** - **Delete** command.

• **Insert Dimension Check**

   Sets the check option for the **Dimension** - **Modify** - **Insert** command.

Automatic Dimensioning Settings

In the **Automatic Dimensioning Settings** section 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 **Dimension Settings**.
2. Select **Geo Sense** from the **Type** list under **Automatic Dimensioning Settings**.
3. 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 124. Geo Sense Options on page 258 illustrates the effects of these options on the resulting dimensions.

Figure 124. Geo Sense Options
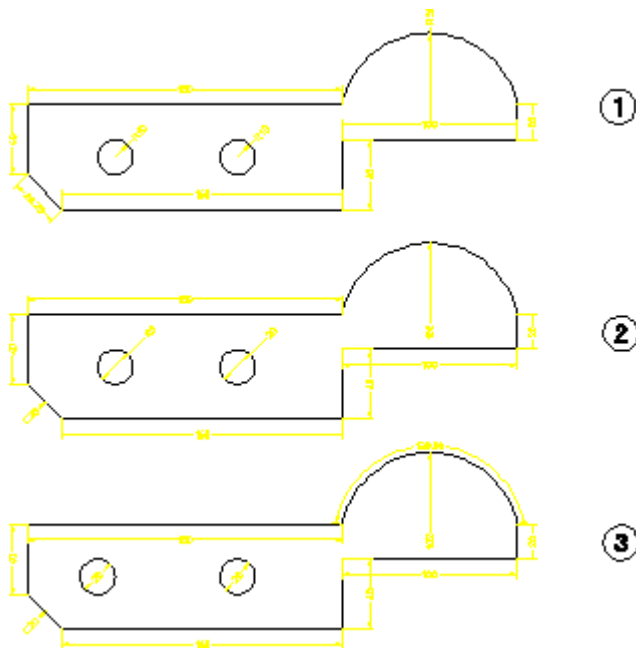
1 Defaults
2 Diameter and Chamfer On
3 Diameter, Chamfer, and Arc On

Scan Settings

To preselect the geometry items to be included in automatic dimensioning with **Scan**:

1. Click **Options** in **Dimension Settings**.

2. Select **Scan** from the **Type** list under **Automatic Dimension Settings**.

3. 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, the **Style** radio button is depressed to indicate that you are using your own dimensioning styles. Optionally, you can press the **Standard** radio button 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** to do this. The style display window shows the current settings.

2. Click **Style** in **Dimension**.

3. Enter a name for the new table entry in the name entry field above the (originally empty) style table.

4. Click **Add** in **Dimension Style**.

   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 **Dimension**.

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 via the **Style** menu.

1. Set up the desired dimensioning style, using either **Dimension Settings** or the **Dimension Style** table.

2. Click **Style** in **Dimension**.

   The style display window should reflect the settings you want to apply.

3. Click **Apply** in **Style**.

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

The **Grab** command 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 **Style** in **Dimension**.

2. Click **Grab** in **Style**.

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 **End**.

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 **Dimension**.

2. Select the table entry you wish to change.

3. Click **Edit** in **Dimension Style**. 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 **Dimension**.

2. Click the table entry you wish to delete.

3. Click **Delete** in **Dimension Style**.

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 **Dimension**.

2. Click **Store** in **Dimension Style**.

3. In the **Store** dialog, Specify a filename and select a file type.

4. Click **OK** to save the dimension text extension(s).

   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. Click **Style** in **Dimension**.

2. Click **Open** in **Dimension Style** to display the **Open** dialog.

3. 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 **Change...** in **Dimension**.

2. Select the dimension(s) to be changed. Either click on a single dimension or box multiple dimensions or use **Select**.

3. In **Change Dimension**, click the tab of the dimension items you want to change:

   • Line

   • Text

   • Units

Note that the settings in **Dimension Settings** apply to all dimensions to be added to the drawing, while the settings in **Change Dimension** apply to existing dimensions you selected.

Changing Dimension Line Attributes

The **Change Dimension** dialog box enables you to change the attributes of dimension lines:

- Line Settings
- Terminator (Arrow) Settings

Changing Dimension Line Settings

In the **Line** section of the **Line** dialog box under **Change Dimension**, 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	Changes the color of extension lines, dimension lines, and text frames.
Offset Point	Changes the offset distance between a geometry feature and the dimensioning extension lines.

See Dimension Line Settings on page 226 for an explanation of these dimension line attributes.

Changing Dimension Line Terminator Settings

In the **Arrow** section of the **Line** dialog box under **Change Dimension**, you can change the type, size, and fill attributes of dimension line terminators. See Terminator Settings on page 233 for an explanation of these terminator attributes.

Changing Dimension Text Settings

In the **Text** dialog box under **Change Dimension**, you can change the attributes of dimension text.

1. Begin by selecting the dimension text component(s) 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 pull down the **Set** list to select any of the following:

- ○ **Upper**
- ○ **Lower**
- ○ **Prefix**
- ○ **Postfix**
- ○ **Superfix**
- ○ **Subfix**

Then select **Value** or **Tolerance** from the **Subset** list or tick 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 236 for an explanation of these attributes.

Changing Dimension Unit Settings

In the **Units** dialog box under **Change Dimension**, you can change the settings of dimension units. See Dimension Unit Settings on page 246 for an explanation of dimension unit settings.

Example

The example below shows how to change dimension units from millimeters to fractional inches.

Figure 125. Changing Millimeters to Fractional Inches on page 263 shows the dimension before and after the change:

Figure 125. Changing Millimeters to Fractional Inches

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 with the **Reset** command.

• 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 the Shift key 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.

Change the Handle settings in **Setup ▸ Edit Environment ▸ System ▸ Handles** tab.

To modify a dimension line:

1. Click **Modify** in **Dimension**.

2. Click **Line** in **Modify**.

3. Click one of the following options in **Line**:

   • **Break**

   • **Stagger**

   • **Change Vertex**

   • **Reset**

Breaking Dimension Lines

To break a dimension extension line around obstructing features:

1. Click **Modify** in **Dimension**.

2. Click **Line** in **Modify**.

3. Click **Break** in **Modify**. Refer to .

Figure 126. Broken Extension Line
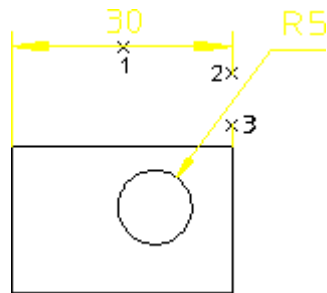
4. Click the dimension text for the dimension (1).
5. Specify the start point for the break (2).
6. Specify the end point for the break (3).

📝 **Note**

The break will be 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** in **Dimension**.
2. Click **Line** in **Modify**.
3. Click **Slant** in **Modify**. Refer to the following figure.

Slanted Extension Lines
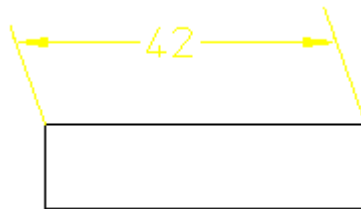
4. Type the slant angle in the User Input Line.
5. Select the dimension to slant.

The dimension extension lines are slanted by the specified angle.

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** in **Dimension**.
2. Click **Line** in **Modify**.
3. Click **Stagger** in **Modify**. Refer to Figure 127. Staggered Extension Lines on page 266.

Figure 127. Staggered Extension Lines
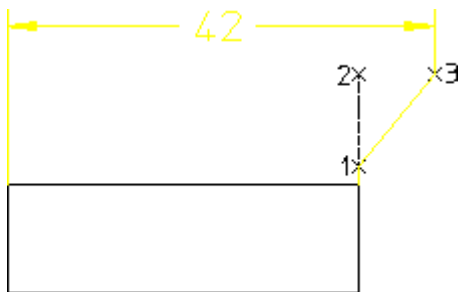
4. Click the dimension text for the dimension.
5. Specify the start point for the stagger on the extension line (1).
6. Specify the end point for the stagger on the extension line (2).
7. Specify the point where you want the stagger to go (3).

Figure 128. Effect of Moving Staggered Extension Lines
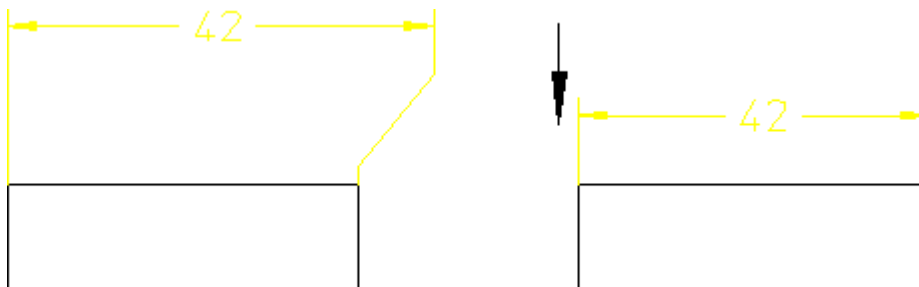
Note

If a dimension contains several staggers, only the first is retained following a move. The staggers will be lost after any modification of the geometry to which the dimension is attached.

Figure 129. Multiple Staggered Extension Lines
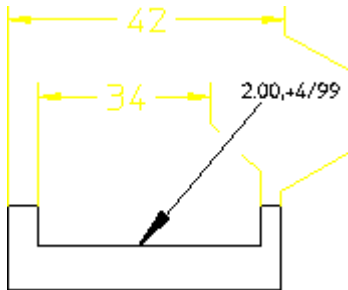
Modifying a Dimension Vertex

The dimension vertex can be relocated to a new position without the need to click the first point again.

Figure 130. Relocating a Dimension Vertex
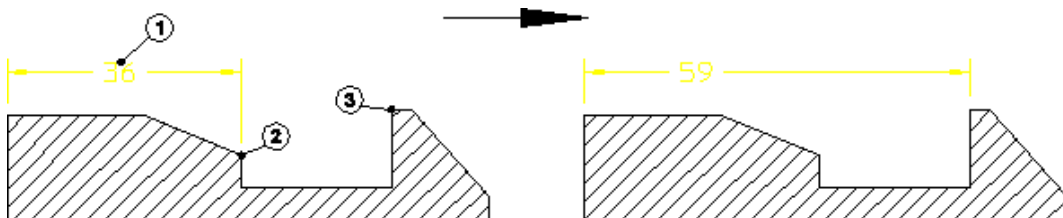
To modify a dimension vertex:

1. Click **Modify** in **Dimension**.
2. Click **Line** in **Modify**.
3. Click **Change Vertex** in **Modify**.
4. Click **Keep Text** in **Change Vertex**.
5. Click the dimension (point 1).
6. Click the old reference element (point 2).
7. Click the new reference element (point 3).

Creo Elements/Direct Drafting relocates the dimension but leaves the dimension text in its original position. If you click **Move Text** in **Change Vertex**, you can also relocate the dimension text.

Resetting Dimension Lines

To reset a broken or staggered dimension line:

1. Click **Modify** in **Dimension**.
2. Click **Line** in **Modify**.
3. Click **Reset** in **Modify**.
4. Identify the dimension for the dimension 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.

To move or rotate dimension text:

1. Click **Modify** in **Dimension**.
2. Click **Text** in **Modify**.
3. Click one of the following options in **Text**:

   • **Move Text**
   • **Move Text & Line**
   • **Rotate Text**

Moving Dimension Text

To move dimension text without moving the dimension or extension lines:

1. Click **Modify** in **Dimension**.
2. Click **Text** in **Modify**.
3. Click **Move Text** in **Text**.
4. Select the dimension text items you want to move.

Creo Elements/Direct Drafting replaces each selected dimension text item with a trace box.

5. Click the desired location for the dimension text.

    Creo Elements/Direct Drafting relocates the selected dimension text items.

Moving Complete Dimensions

To move one or more complete dimensions:

1. Click **Modify** in **Dimension**.
2. Click **Text** in **Modify**.
3. Click **Move Text + Line**.
4. Click the dimension text of the dimension(s) you want to move.

    Creo Elements/Direct Drafting draws a trace of the dimensions.

5. Click the desired location for the dimension text.

    Creo Elements/Direct Drafting relocates the selected dimensions.

Moving Dimension Text and Leader Line

To move a dimension text and its associated leader line:

1. Click **Modify** in **Dimension**.
2. Click **Text** in **Modify**.
3. Click **Move Text + Leader**.
4. 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.
5. Click the desired location for the dimension text. Creo Elements/Direct Drafting relocates the selected dimension text and its leader line.

Figure 131. Moving a Dimension Text and Its Leader Line
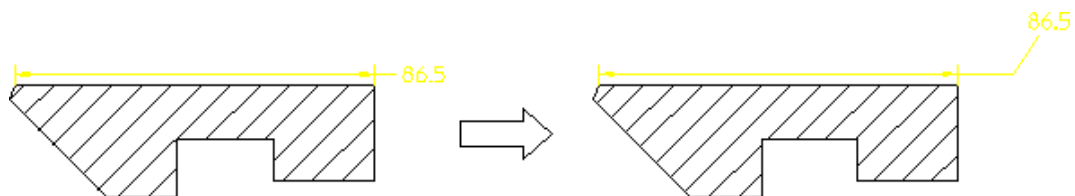
Rotating Dimension Text

Horizontal is the zero angle and positive rotation is counterclockwise.

To rotate dimension text:

1. Click **Modify** in **Dimension**.

2. Click **Text** in **Modify**.

3. Click **Rotate Text** in **Text**.

4. Enter an absolute angle of rotation. A positive angle causes a counterclockwise rotation.

5. Select the dimension text(s) you want to rotate.

    Creo Elements/Direct Drafting rotates the selected dimension text.

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 132. Aligning Dimensions to Other Dimensions
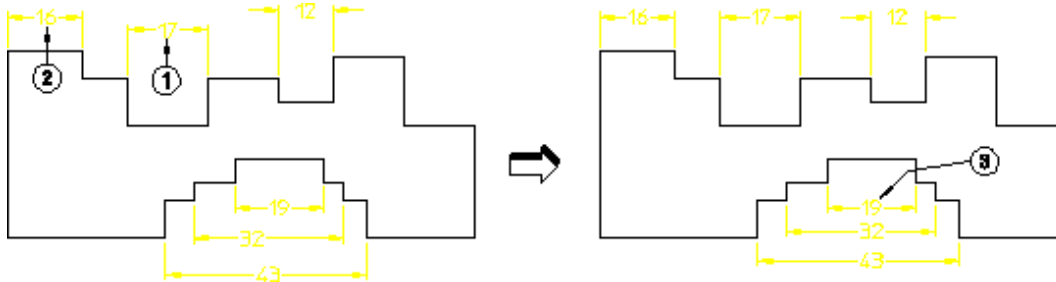
To align a dimension:

1. Click **Modify** in **Dimension**.

2. Click **Text** in **Modify**.

3. Click **Move Text & Line**.

4. Click the dimension you want to move (point 1).

5. Click the reference dimension (point 2).

6. Drag the cursor to the new location.

7. Release the mouse button.

8. 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 133. Effects of Freezing a Dimension
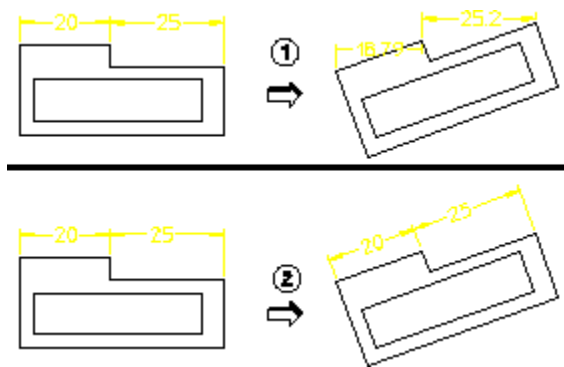
1 Dimension created by **Single**, **Horizontal**. When the part is rotated 20°, the dimensions remain horizontal giving a correct but meaningless result.
2 Same as item 1 but the dimensions were frozen with **Freeze** before the part was modified.

To freeze a dimension:

1. Click **Modify** in **Dimension**.

2. Click **Freeze** in **Modify**.

3. Click the dimension you want to incline.

Note the following:

- If you use the select **Box** option when selecting dimensions, only those dimensions with the attribute **Horizontal** or **Vertical** are affected.

- There is no **Undo** for this command.

Editing 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, then press **[Enter]**.

Creo Elements/Direct Drafting will redraw the edited dimension text.

To edit existing dimension text:

1. Click **Modify** in **Dimension**.

2. Click **Edit Dim Text** in **Modify**.

3. Click the dimension text to edit.

   Creo Elements/Direct Drafting displays the dimension text in the User Input Line.

4. Edit the text, then press **[Enter]**.

5. Click **End** to end this command.

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** in **Dimension**.

2. Click **Edit Dim Text** in **Modify**.

3. Click the dimension to be underlined.

4. Press **[Enter]** to accept the current value. Optionally, you can edit this value.

Creo Elements/Direct Drafting displays the edited dimension value with an underline.

Figure 134. Dimension Underlining
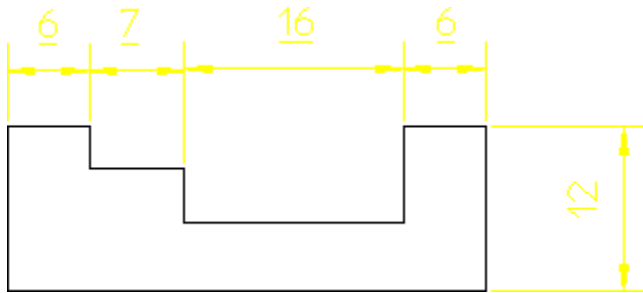
To remove an underline:

1. Click **Modify** in **Dimension**.

2. Click **Cancel Edit** in **Modify**.

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** in **Dimension**.

2. Click **Cancel Edit** in **Modify**.

3. Click the dimension text you want to reset to its original version.

   Creo Elements/Direct Drafting displays the original dimension text.

4. Click **End** to end this command.

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.

Deleting a Segment from a Chain or Datum Dimension Group

To delete a segment from an existing chain or datum dimension group:

1. Click **Delete** in **Dimension**.

2. Click the dimension text of the segment to be deleted.

3. If the segment you are deleting is part of a dimension chain, click the dimension text of the next segment in the chain. This segment will grow to fill the space created by the segment you delete.

4. When you have finished deleting dimensions, click **End** to end this command.

Inserting a New Chain or Datum Dimension Segment

To insert a new segment into an existing chain or datum dimension group:

1. Click **Modify** in **Dimension**.

2. Click **Insert** in **Modify**.

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 **End** to end this command.

Grouping

You can group annotations together. Groups on page 505 may consists 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:

Enter `help da_dimension` at the Creo Elements/Direct Drafting command line.

The screen will clear and Creo Elements/Direct Drafting will display further information about dimensioning.

9

Hatching a Drawing

This chapter shows 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 **Enter** commands 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 for details.

Hatching a Drawing Area

You have a choice of two hatch methods:

Automatic Hatching

You can hatch any area fully enclosed by drawing lines or already hatched.

To automatically hatch a drawing area:

1. Click **Enter** in **Hatch**.

2. Specify a point inside the area to be hatched.

   The cursor will automatically track around the enclosing polygon and then search for any inner holes. Having established the inner and outer contours, Drafting will hatch the bounded area. If Drafting finds any gaps or incorrectly split lines in the enclosing polygon, the command is aborted and 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 135. Hatch Pattern Aligned with Match Point on page 276 shows the hatch pattern aligned with an external match point.

Figure 135. Hatch Pattern Aligned with Match Point

To set the hatch match point:

1. Click **Enter** in **Hatch**.

2. Click **Enter Point** in **Hatch**. Then specify a point with you want to align the hatch to be created.

3. Specify a point inside the area to be hatched.

One hatch line of all subsequent hatching will pass 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 **Hatch ▸ Enter ▸ Manual**.

2. Click the start point on the existing outer contour of the area to be hatched. See Automatic Splitting on page 278 for further contour creation options.

3. Continue by clicking successive points around the outer contour.

4. 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.

5. Click **End** and Drafting will hatch the area.

   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 136. Indicating the Area for Hatch on page 277 illustrates the above techniques.

Figure 136. Indicating the Area for Hatch
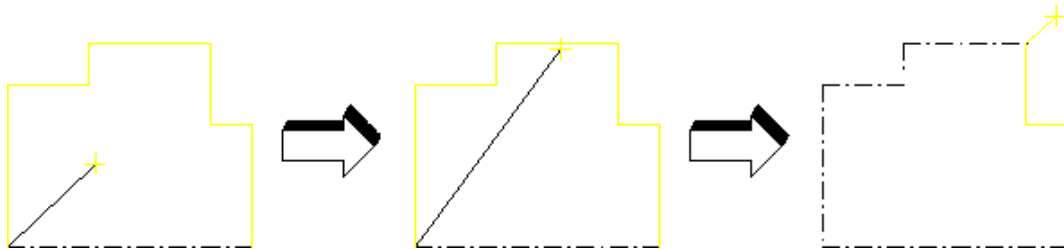
Not all the sides have to be drawing lines. An imaginary line can be used to separate free space or existing points. See Figure 137. Hatching — Leaving an Unhatched Area on page 278.

Figure 137. Hatching — Leaving an Unhatched Area
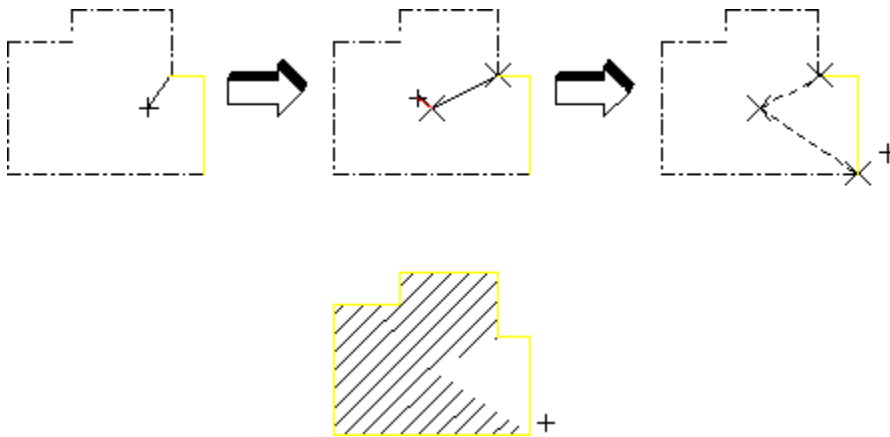
Automatic Splitting

You can use automatic splitting options for extended contour creation functionality. To enable these options, **Hatch ▸ Settings ▸ Automatic Split for Hatch Manual** click. 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:

To display the **Hatch Settings** dialog box:

- Click **Settings** in **Hatch**.

Or:

- Pull down the **Setup** menu. Click **Edit Environment** and then click **Hatch**.

When you have defined your hatch settings, either click **Hatch** to apply these settings to the next face(s) you click, or click **OK** to apply these settings to subsequent hatching (via **Enter**).

Simple Hatch Settings

Press the **Hatch** radio button in the **Hatch Settings** dialog box to switch to **Hatch** mode. In this mode, you can select or change the following hatch settings:

- **Distance**
- **Angle**
- **Linetype**
- **Color**

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, enter the required value into the **Distance** field in **Hatch Settings**. This value is interpreted in terms of the length unit (such as mm) currently set up for your Drafting environment. Figure 138. Hatch Line Distance on page 279 shows an example of a hatch line distance.

Figure 138. Hatch Line Distance

Hatch Angle

To set the hatch angle, enter the required value into the **Angle** field in **Hatch Settings**. The angular units will be those currently defined for your Drafting environment. A zero value implies horizontal hatch lines and a positive angle results in counterclockwise rotation.

Figure 139. Positive Hatch Angle on page 280 shows a positive hatch angle.

Figure 139. Positive Hatch Angle
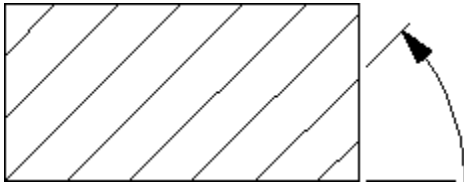
Hatch Linetype

To set the hatch linetype, select a linetype from the drop-down list box next to the **Linetype** field in **Hatch Settings**.

Hatch Color

To set the hatch color, select a color from the drop-down list box next to the **Color** field in **Hatch Settings**. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

Setting Hatch Patterns

Patterns consist of any number of simple hatch lines built up in layers. When you have created a pattern, it will be active every time you click **Enter** in **Hatch**. The pattern will be 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 **Enter Point** in **Hatch**). The perpendicular offset is the entered value multiplied by the distance. See Figure 140. Hatch Perpendicular Offset on page 281.
Distance	Any Number	The distance between each of the layer's hatch lines will be the entered value

Name	Allowed Value	Description
		multiplied by the distance value specified in **Hatch** mode.
Angle	0 to ± 360	The entered angle will be added to the angle specified in **Hatch** mode 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 59 for instructions on defining custom colors.
Linetype		Any of the eight Drafting standard linetypes can be selected from a drop-down list box.

Figure 140. Hatch Perpendicular Offset
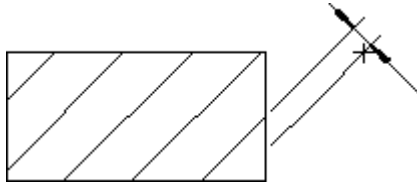
Creating a Hatch Pattern

To create a hatch pattern:

1. Press the **Pattern** radio button in the **Hatch Settings** dialog box to switch to **Pattern** mode.

2. Specify the first layer's values for **Offset**, **Distance**, **Angle**, **Color** and **Linetype**.

3. Enter 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 286. 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 **Enter** commands in **Hatch** will revert to the hatch parameters set using the linetype and color specified in the **Hatch** mode (see Simple Hatch Settings on page 279).

To cancel a hatch pattern:

1. Press the **Pattern** radio button in the **Hatch Settings** dialog box to switch to **Pattern** mode.

2. Click **Cancel**.

   The previously created hatch pattern will be canceled.

Changing Hatch Settings

This section shows how to change existing hatching.

- Changing Individual Hatch Settings on page 282
- Changing Multiple Hatch Settings on page 284
- Changing a Hatch Pattern on page 284

Changing Individual Hatch Settings

Via the **Change** menu in **Hatch**, you can change individual hatches in your drawing.

- Changing the Hatch Angle on page 282
- Changing the Hatch Line Distance on page 282
- Changing the Hatch Color on page 283
- Changing the Hatch Linetype on page 283
- Changing the Match Point on page 283
- Changing a Hatch Pattern on page 284
- Automatic Splitting on page 278

Changing the Hatch Angle

To change the hatch angle:

1. Click **Change** in **Hatch**.
2. Click **Angle** in **Change**.
3. Enter the required hatch angle.
4. Click the hatching to be changed. You can click individual areas of hatch or box several hatch areas or click **All**. Drafting will change the selected hatching.

   Alternatively, use the **Selection** block as you would for a **Delete** command, and finish by clicking **Confirm**.

Changing the Hatch Line Distance

To change the line distance of an existing hatch:

1. Click **Change** in **Hatch**.
2. Click **Distance** in **Change**.
3. Enter the required line distance.
4. Identify the hatching to be changed. You can click individual areas of hatch or box several hatch areas or click **All**. Drafting will change the selected hatching.

   Alternatively, use the **Selection** block as you would for a **Delete** command, and finish by clicking **Confirm**.

Changing the Hatch Color

To change the color of an existing hatch:

1. Click **Change** in **Hatch**.
2. Click **Color** in **Change**.
3. Select a color from the **Color** list box. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.
4. Identify the hatching to be changed.

   You can only identify individual areas of hatch to be changed. When there are two hatch patterns superimposed on each other, the pattern nearest to the cursor will be changed.

   Alternatively, use the **Selection** block as you would for a **Delete** command, and finish by clicking **Confirm**.

Changing the Hatch Linetype

To change the hatch linetype:

1. Click **Change** in **Hatch**.
2. Click **Linetype** in **Change**.
3. Select a linetype from the **Linetype** list box.
4. Identify the hatching to be changed.

   You can only identify individual areas of hatch to be changed. When there are two hatch patterns superimposed on each other, the pattern nearest to the cursor will be changed.

   Alternatively, use the **Selection** block as you would for a **Delete** command, and finish by clicking **Confirm**.

Changing the Match Point

To change the hatch match point:

1. Click **Change** in **Hatch**.

2. Click **Match Point** in **Change**.

3. Enter the required point for the hatch lines to match.

4. Identify the hatching to be changed. You can click individual areas of hatch or box several hatch areas or click **All**. Drafting will change the selected hatching.

   Alternatively, use the **Selection** block as you would for a **Delete** command, and finish by clicking **Confirm**.

Automatic Splitting

Read about Automatic splitting in the Manual Hatching on page 278 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 **Change** in **Hatch**.

2. Click **Multiple** in **Change**. Drafting returns the **Change Hatch** dialog box.

3. Change the hatch settings as required. The method is the same as described in Simple Hatch Settings on page 279.

4. Click the hatch to which you want to apply these changes.

   Alternatively, use the **Selection** block as you would for a **Delete** command, and finish by clicking **Confirm**.

Changing a Hatch Pattern

To change a hatched area to a new pattern:

1. Click **Change** in **Hatch**.

2. Click **Pattern** in **Change**.

3. Specify or select the settings for the first layer of the pattern (see Setting Hatch Patterns on page 280 for details).

4. If there are further layers, continue by defining the next layer(s) in the same way.

5. Finish defining the pattern by clicking **OK** in **Hatch Settings**.

6. Identify any hatched areas and Drafting will change them to the new pattern.

Hatching Around Text and Dimension Text

When text lies in a hatched area, 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.

The **Text Window** command 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. You use the **Text Window** command to punch a window through the overlapping hatching.

Figure 141. Text and Dimension Text: Hatched Area
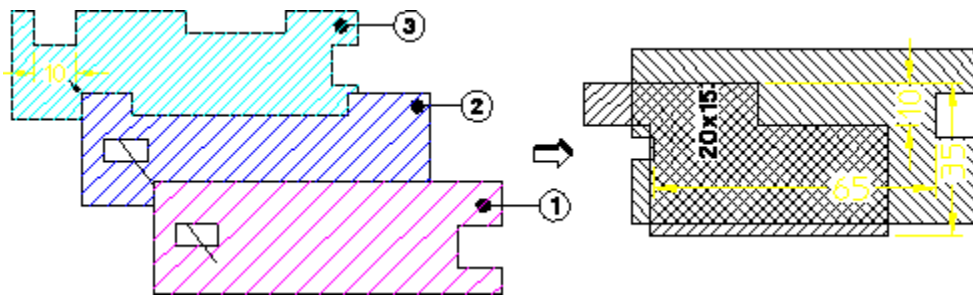
1. Hatch the required area in automatic or manual mode.
2. Click **Change** in **Hatch**.
3. Click **Text Window** in **Change**.
4. Click the dimension text around which you want to hatch.

Figure 142. Hatching Around Dimension Text
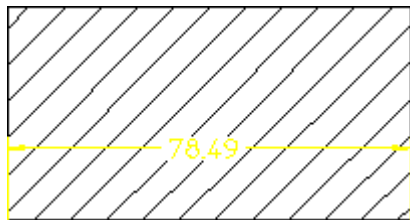
To move dimension text when you have used the **Text Window** command to create the hole around the text:

1. Move the dimension text to its new position.

2. Delete the old hole in the hatching (Drafting deletes the hole and regenerates the hatch).

3. Use the **Text Window** command to create a new hole around the dimension text.

Standard Hatch Patterns

This section shows how to set up standard hatch patterns which will be used by all subsequent **Enter** commands in **Hatch**.

To set up a standard hatch pattern:

1. Display the **Hatch Settings** dialog box:

   • Click **Settings** in **Hatch**.

   Or:

   • Pull down the **Setup** menu. Click **Edit Environment** and then click **Hatch**.

2. Press the **Standard** radio button in the **Hatch Settings** dialog box to switch to **Standard** mode.

3. Select the desired option (**Iron**, **Steel**, **Copper**).

   The selected standard hatch pattern will become active.

The three standard options are shown below together with the values of offset, distance, angle, color, and linetype used (refer to Setting Hatch Patterns on page 280). All the examples shown are drawn with the following hatch parameters: hatch angle 45, hatch distance 5, and 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 143. Iron
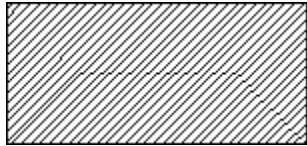
Standard Hatching — Steel

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 144. Steel
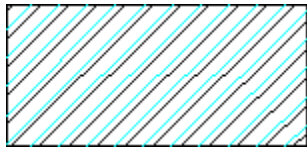
Standard Hatching — Copper

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 145. Copper
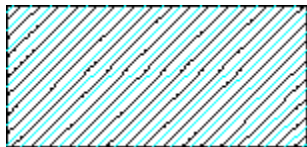
Converting Hatch to Geometry

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 make sure your hatch does not change appearance, convert it to geometry.

1. Click **Hatch**.

2. Click **Hatch to Geometry**.

3. "Identify hatch to convert it to the geometry" prompt appears. Click in your viewport window or use the **Select** function to select hatches in the current part. **Global** selection is not available.

4. Click **Confirm**.

5. Click **End**.

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.

Completely filled hatches will not convert. 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** and **Redo** in combination with **Hatch to Geometry**.

Deleting Hatching

This section shows how to delete areas of hatching.

To delete an area of hatching:

1. Click **Delete** in **Hatch**.

2. Click individual areas of hatching to be deleted, (or enclose a group of hatched areas in a box). Alternatively:

   • Use the **Selection** block as you would for a normal deletion.

   • Finish by clicking **Confirm** to delete the selected groups of hatching.

3. If you delete an area by mistake, click **Undo** to undo the last delete operation.

4. Click **End** to end the command.

Delete hatch that has been converted to geometry the same way you would delete geometry, using the **Delete** command.

Further Information

For further technical information about hatching, refer to the Drafting help system:

Enter `help hatching` at the Drafting command line.

The screen will clear and Drafting will display further information about hatching.

10

Adding Text to a Drawing

Entering Text

Creo Elements/Direct Drafting provides the following options for entering text:

- Entering Text Via a Text Box on page 292
- Entering Text Directly in the Drawing on page 293

📒 **Note**

If Creo Elements/Direct Drafting draws boxes instead of the text, you must input a text font. Refer to Text Font on page 298. To load additional fonts for selection, follow the instructions in Importing a Font on page 435.

Entering Text Via a Text Box

To add text to your drawing via a text box:

1. Click **Enter Text** in **Text**. Creo Elements/Direct Drafting returns the **Text** box.
2. Enter the required text into the **Text** box.
3. Click **OK** to add the text to the drawing.
4. 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 296.

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 292
- Text Settings on page 296
- Loading an Existing Text File on page 293
- Storing Text in a Text File on page 293

Entering Special Characters

To add a special character to your Creo Elements/Direct Drafting drawing:

1. In **Text**, click **Enter Text**.
2. In the **Text** box, 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. See Editing a Font on page 305 for further information.

3. In **Special Characters**, 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.

4. Click **Insert** to copy the selected special character to the **Text** box. Note that you can insert the special character(s) into existing text in the **Text** box.

5. When the **Text** box contains the required special character(s), click **OK** to add the text to your drawing.

6. Click the position for the text.

Loading an Existing Text File

To load an existing text file for use in your Creo Elements/Direct Drafting drawing:

1. In **Text**, click **Enter Text**.

2. In the **Text** box, click **Load**.

3. Use the **Load** browser to find and load a text file.

4. In the **Text** box, edit the text file as required.

5. Click **OK** to add the text to your drawing.

6. Click the position for the text.

Storing Text in a Text File

To store text in a text file:

1. Click **Text**, **Enter Text** to display the **Text** box. Either type in the text to be saved or load an existing text file (see Loading an Existing Text File on page 293) and edit it.

2. In the **Text** box, click **Store**.

3. In the **Store** browser, select a directory and enter the name of the text file in which you want to store the text.

4. Click **OK** to store your text in this file.

Entering Text Directly in the Drawing

To enter text directly in your drawing:

1. In **Text**, click **Inplace**, **Enter**.

2. Click the position for the text.

3. Enter 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 via Text Settings on page 296.

4. Click **End**.

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 294)

- Change the settings of this text (see Changing Text Settings on page 300)

- Modify this text just like any geometric element (see Modifying a Drawing on page 157)

- Add a reference line to this text (see Creating a Reference Line on page 300)

- Convert the text to normal geometry (see Converting Text to Geometry (DXF/ DWG and IGES 2D Translation) on page 300)

Editing and Deleting Text

Creo Elements/Direct Drafting provides the following options for editing and deleting text:

- Editing Text Via a Text Box on page 294

- Editing Text Directly in the Drawing on page 295

- Moving Text on page 295

- Deleting Text on page 295

🗩 Note

If Creo Elements/Direct Drafting draws boxes instead of the text, you must input a text font. Refer to Text Font on page 298. To load additional fonts for selection, follow the instructions in Importing a Font on page 435.

Editing Text Via a Text Box

To edit existing text via a text box:

1. Click **Edit Text** in **Text**.

2. In the Creo Elements/Direct Drafting drawing, click the text to be edited. Creo Elements/Direct Drafting displays this text in the **Edit Text** box.

3. Use the keyboard and edit keys to make the required changes to the text.

4. Click **OK**. The normal display will reappear and the changed text is displayed in the drawing.

Editing Text Directly in the Drawing

To edit text directly in your drawing:

1. In **Text**, click **Inplace**, **Edit**.

2. Click the text to be edited.

3. 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 296).

4. Click **End**.

Moving Text

Quickly move text by moving the text box.

1. Hover over the text. Creo Elements/Direct Drafting highlights a box around the text (if **Enable Fly-By Highlighting** is on).

2. Drag the text to a new location.

Deleting Text

There are several ways to delete text from your drawing.

Use the Delete key on your keyboard:

1. Click the text. A frame appears around the text.

2. Press the **Delete** key on your keyboard.

Creo Elements/Direct Drafting deletes text in the same way as normal drawing lines. To delete text without using the selection block:

1. Click **Delete**.

2. Box the text to be deleted, or click a single text to be deleted. Creo Elements/ Direct Drafting will now delete the selected text.

If you use the **Text** option in **Select**, 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 block:

1. Click **Delete**.

2. Select text for deletion using the **Selection** block options just as you would for a normal **Delete** command. You can click **Undo** to correct any mistakes.

3. Click **Confirm** to delete the selected text block(s).

Text Settings

You can display the **Text Settings** dialog box in the following ways:

• Click **Enter Text** in **Text**. Then click **Settings** in the **Text** box.

• Click **Settings** in **Text**.

• Pull down the **Setup** menu. Click **Edit Environment** and then click **Text**.

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 296

• Text Font on page 298

• Text Options on page 298

Text Parameters

Use the items in the **Parameters** section of the **Text Settings** dialog box to set the following text parameters:

• **Size**

• **Angle**

• **Ratio**

• **Linespace**

• **Slant**

• **Fill**

Text Character Size

To set the text size, enter the required value into the **Size** field in **Text Settings**. This value is interpreted in terms of the length unit (such as mm) currently set up for your Creo Elements/Direct Drafting environment. The Creo Elements/Direct Drafting default text size is 3.5 mm.

Text Angle

To set the text angle, enter the required value into the **Angle** field in **Text Settings**. The angular units will be those currently defined for your Creo Elements/Direct Drafting environment. A zero value implies horizontal text and a positive angle results in counterclockwise rotation. Figure 146. Setting the Text Angle on page 297 illustrates the text angle concept.

Figure 146. Setting the Text Angle

Text Character Ratio

To set the width-to-height ratio of text characters, enter the required value into the **Ratio** field in **Text Settings**. The default text ratio is 1.0.

Linespace

To set the line spacing of text lines, enter the required value into the **Linespace** field in **Text Settings**. The default linespace in Creo Elements/Direct Drafting is 2.2 mm. The distance between lines of text in a block is computed as follows:

Actual line spacing = Linespace * Text Size

Text Character Slant

To set the slant angle of text characters, enter the required value into the **Slant** field in **Text Settings**. The angular units will be those 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 147. Text with a 15° Slant on page 297 shows text with a 15° slant.

Figure 147. 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. Simply click on the **Fill** check box to switch the fill mode on or off. Figure 148. Unfilled and Filled Text on page 298 shows an example (unfilled and filled).

Figure 148. Unfilled and Filled Text

Text Font

To select a font for text, simply select that font from the **Font** list in **Text Settings**. 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 435.

OSD's different internal and external encoding schemes were Unicode on page 513 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 fontssubdirectory below the Creo Elements/Direct Drafting installation directory.

Text Options

Use the items in the **Options** section of the **Text Settings** dialog box to set the following text options:

• Color

• Frame

• Origin Vertex

Text Color

To set the text color, select a color from the **Color** list in **Text Settings**. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

Text Frame

By default, text in Creo Elements/Direct Drafting is displayed without a frame. Optionally, you can select the following frame forms from the **Frame** list in **Text Settings**:

• **Balloon**

Figure 149. Text with a Balloon Frame on page 299 shows text with a balloon frame.

Figure 149. Text with a Balloon Frame

• **Box**

Figure 150. Text with a Box Frame on page 299 shows text with a box frame.

Figure 150. Text with a Box Frame

• **Flag**

Figure 151. Text with a Flag Frame on page 299 shows text with a flag frame.

Figure 151. Text with a Flag Frame

Text Origin Vertex

To set the position of the origin of any subsequent text, click **Origin Vertex** and specify the associated position number (an integer between 1 and 9). Figure 152. Cursor Positions in the Text Box on page 300 shows you the relative position associated with each of these adjust values.

Figure 152. Cursor Positions in the Text Box
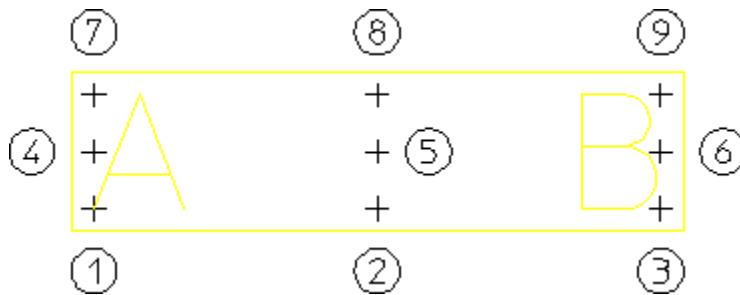
Changing Text Settings

To change the settings of existing text:

1. Click **Change Text** in **Text**.

2. Select the text to be changed. Either click on a single text block or box multiple text blocks or use **Select**.

3. In the **Text Change** dialog box, change the settings as required. See the Text Settings on page 296 section for an explanation of these settings. Note that the settings in **Text Settings** apply to all text to be added to the drawing, while the settings in **Change Text** apply to existing text you selected.

Converting Text to Geometry (DXF/DWG and IGES 2D Translation)

The **Text to Geometry** command converts text entities to geometry entities. 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).

Note the following:

*   Only text entities can be converted, not dimension texts.
*   Any text-windows inside hatched areas are retained.

Creating a Reference Line

The **Enter Refline** command lets 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 will remain attached to the target element like a "rubber band" (see Figure 153. Effect of Moving the Target Element on page 301).

To draw a reference between two existing elements:

1. Click **Enter Refline** in **Text**.
2. Click the required option (if any) in **Enter Refline**. See Reference Line Options on page 304 for further information.
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 302.

Figure 153. Effect of Moving the Target Element
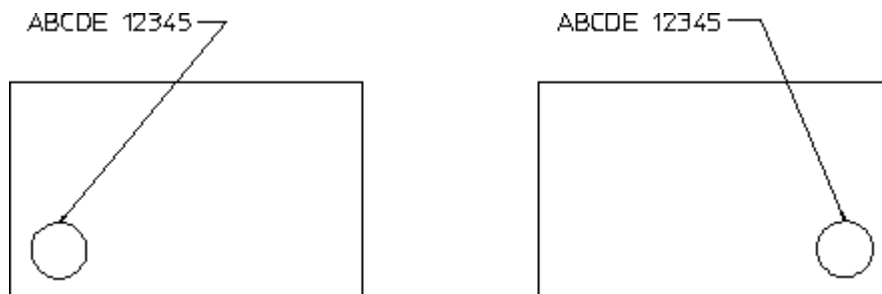
For deletion purposes, even a multi-segment reference line is considered as a single element and can be deleted in one operation.

Reference Line Options

When you create a reference line between two elements, you can choose any (or none) of the following options.

Reference Line Option	Result
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.
Perpendicular	The reference line is drawn perpendicular to the orientation direction of the source element.
Reference Angle	This options prompts you for an angle. Enter the required angle into the command input line. The

Reference Line Option	Result
	angular units will be those 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 option enables you to attach the reference line to its source element. When you move the source element, the reference line will remain attached to this element like a "rubber band".
Underline	The source element is underlined.
Invisible	An invisible reference line is drawn.

When you draw a multi-segment reference line (by clicking points between the source element and its target element), note that these options apply to the first reference line segment (between the source element and the first reference line point you click) only.

Modify a Reference Line

Use the handles of a reference line to quickly modify it:

Reattach the reference line	Click the handle on the start or end of the refline and drag.
Stretch reference line segments	Click inside the refline handles and drag.

Change the Handle Settings in **Setup ▶ Edit Environment ▶ System ▶ Handles**.

Delete a Reference Line

Click the reference line. Press **Delete** on your keyboard.

Reference Line Settings

To display the **Reference Line Settings** dialog box:

• Click **RefLine Settings** in **Text**.

Or:

• Pull down the **Setup** menu. Click **Edit Environment** and then click **Refline**.

The **Reference Line Settings** dialog box enables you to define:

Line Settings

Use the items in the **Line** section of the **Reference Line Settings** dialog box to set the following:

- **Type**
- **Color**
- **Width**

Type

To set the linetype of reference lines, Select a linetype from the **Type** list in the **Line** section of **Reference Line Settings**.

Color

To set the reference line color, select a color from the **Color** list in the **Line** section of **Reference Line Settings**. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

Width

To set the reference line width, enter the required width into the field next to **Width** in the **Line** section of **Reference Line Settings**. Refer to Line Width vs. Line Size on page 56 for further information on line width.

Reference Line Terminator

Use the items in the **Terminator** section of the **Reference Line Settings** dialog box to set the following:

- **Type**
- **Size**

Type

To set the reference line terminator type, select one of the following options from the **Type** list in the **Terminator** section of **Reference Line Settings**:

- **None**
- **Arrow**

- **Dot**

- **Slash**

- **Triangle**

- **Jis**

Size

To set the reference terminator size width, enter the required value into the field next to **Size** in the **Terminator** section of **Reference Line Settings**. 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

Use the items in the **Options** section of the **Reference Line Settings** dialog box to 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, enter the required value into the **Source Gap** field in the **Options** section of **Reference Line Settings**.

Destination Gap

To set a distance between the end point of the reference line and its target element, enter the required value into the **Destination Gap** field in the **Options** section of **Reference Line Settings**.

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, enter the required value into **Offset** field 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 296 and Changing Text Settings on page 300) can be assigned to another text block.

To assign the properties of a text block to another text block:

1. Click **Get** in **Tools ▸ Properties**.

2. Click the text block whose properties you want to get.

3. Click **Put** in **Tools ▸ Properties**.

4. Click the text block to which you want to assign these properties.

Font Editor

The Creo Elements/Direct Drafting Font Editor enables you modify existing Creo Elements/Direct Drafting fonts and create new fonts. This special module 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 **View** menu 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 calling the Font Editor, 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. Pull down the **Tools** menu, click **Font Editor** and **Font**.

2. In the **Select Font** dialog box, select a font and click **Edit**.

3. Pull down the **Tools** menu, click **Font Editor** and **Show Font** to display all of the characters included in this font in the Creo Elements/Direct Drafting drawing area.

4. Pull down the **Tools** menu, click **Font Editor** and **Edit Character**.

5. Enter the character number (for example: 174) of the character you want to edit in the User Input Line. Press [Enter] to display this character.

📝 Note

If you use this method to add further 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.

Creo Elements/Direct Drafting displays the character and a yellow character-frame.

6. Modify or delete existing character geometry.

7. Pull down the **Tools** menu, click **Font Editor** and **Draw**.

8. Select the **Draw** option you want to use to create the new character. Note that the maximum number of unconcatenated elements in a character is 84.

Figure 154. Character Frame
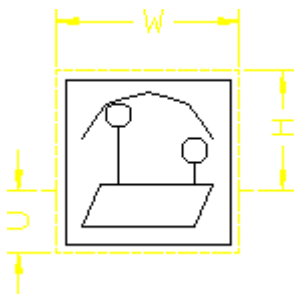
9. Pull down the **Tools** menu, click **Font Editor** and **Update Character**.

10. Enter the character number (you can either keep this number or assign a new number).

11. Repeat this procedure for other characters if/as required.

12. To save the modified font, pull down the **Tools** menu, click **Font Editor** and **Store Font**.

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 it can be re-used following an Creo Elements/Direct Drafting update.

Changing the Character Spacing

To change the spacing between characters:

1. Pull down the **Tools** menu, click **Font Editor** and **Change**.

2. Select one of the following options:

Normal Distance	to change the distance between constant-spaced characters.
Constant Width	to change the distance between variable-spaced characters.

The next time you enter text, Creo Elements/Direct Drafting draws the text block with the new character spacing.

Figure 155. Characters with Constant Spacing

Figure 156. Characters with Variable Spacing
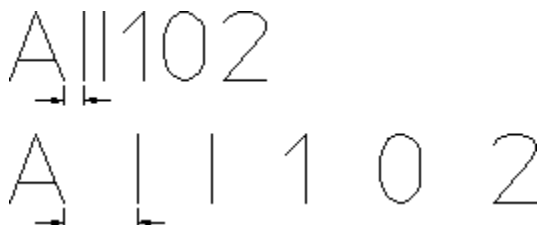
Grouping

You can group annotations together. Group on page 505 may consists 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:

Enter: `help text` at the Creo Elements/Direct Drafting command line.

11

Adding Symbols to a Drawing

Geometric Tolerancing Symbols

This section shows 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.

Click **Symbols** to display the **Symbols** menu.

Setting Symbol Size

The size of symbols and associated text is determined by the **Size** in **Text Settings**. Therefore, before you start adding symbols to your drawing, set the desired text size as described in Adding Text to a Drawing on page 291.

To change the size of symbols and associated text, use **Change Text** as described in Adding Text to a Drawing on page 291.

Deleting and Moving Symbols

Deleting Symbols

There are two ways to delete a symbol:

1. Click the symbol. A frame appears around the symbol. Press the **Delete** key on your keyboard.
2. Click **Delete** in the menu and identify the symbol to be deleted.

Moving Symbols

There are two ways to move a symbol:

1. Move the symbol box.

   • Hover over the symbol. Creo Elements/Direct Drafting highlights a box around the symbol (if **Enable Fly-By Highlighting** is on).

   • Drag the symbol to a new location.

2. Click **Move** in **Modify** and proceed as you would for a normal element such as text, a line, or an arc.

Building a Geometric Tolerance Group

Click **Create** in **Symbols** to display the **Symbols** dialog box. The items in this dialog box allow you to quickly build up the desired geometric tolerance group.

The main items are:

Item	Description
Tolerance	Sets **Normal** or **Composite** symbol mode.
Type	Enables you to select a symbol type.
Symbol	This rectangular area provides a preview of the current symbol definition.
Symbol Browser Bar	This bar displays predefined symbols of the selected **Type**.
Add Text	Displays a text 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
Undo	The **Undo** arrow button undoes the last addition to the symbol.
Apply	This button 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 **Symbols**.

2. In the **Symbols** dialog box, select a symbol mode from the **Tolerance** list:

    *   **Normal**

    *   **Composite**

3. Select a symbol type from the **Type** list:

    *   **Geometric Symbols**

    *   **Modifying Symbols**

    *   **Draws Datum Target and Revision Symbols**

    Symbols of the selected type are displayed on the Symbol Browser Bar for selection.

4. Click on a symbol on the Symbol Browser Bar. This symbol is displayed in the **Symbol** preview area.

5. Optionally, click **Add Text** to display a text box in which you can enter text to be added to the symbol. By convention, only UPPER CASE characters should be entered. Click **OK** to close the text box and to append the text to the symbol.

6. Click the appropriate button(s) to append vertical and/or horizontal lines and/ or] characters to the symbol.

7. When the symbol displayed in the preview area is complete, click **Apply**. Then position the symbol in the Creo Elements/Direct Drafting drawing. Note that another copy of the same symbol is added to the drawing every time you click a point in the drawing.

8. Click **End** to terminate the positioning of the symbol.

9. Indicate the symbol location.

10. Optionally, attach a reference line that connects the symbol to the associated feature in the Creo Elements/Direct Drafting drawing. See Creating a Reference Line on page 300 for instructions.

Normal or Composite Symbols

Figure 157. Position Geometric Symbol on page 312 shows an example of a **Position** geometric symbol. Both the **Normal** and **Composite** options are shown.

Figure 157. Position Geometric Symbol
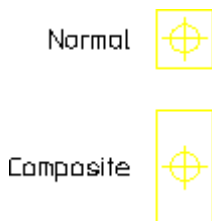
Drawing a Geometric Symbol

Select **Geometric Symbols** from the **Type** list in **Symbols** to draw geometric symbols.

Form Symbols

Figure 158. Symbol Type and Form Geometric Symbols on page 313 shows the four form symbols.

Figure 158. Symbol Type and Form Geometric Symbols

1 Straight
2 Flat
3 Round
4 Cylinder

Profile Symbol

Figure 159. Profile Geometric Symbols on page 313 shows the two profile symbols.

Figure 159. Profile Geometric Symbols

Location Symbols

Figure 160. Location Geometric Symbol on page 313 shows the three location symbols.

Figure 160. Location Geometric Symbol

1 Position
2 Symmetry
3 Concentric

Alignment Symbols

Figure 161. Alignment Geometric Symbols on page 314 shows the three alignment symbols.

Figure 161. Alignment Geometric Symbols

    1 Angular
    2 Perpendicular
    3 Parallel

Runout Symbols

shows the two runout symbols.

Figure 162. Runout Geometric Symbols

    1 Circular
    2 Total

Appending Text

Text can be added to symbols with the **Add Text** command. Use this command to interrupt the symbol group, add text, and then proceed with the symbols. By convention, only UPPER CASE characters should be entered. The two examples in show you the correct sequence of steps.

Figure 163. Adding Symbol Text
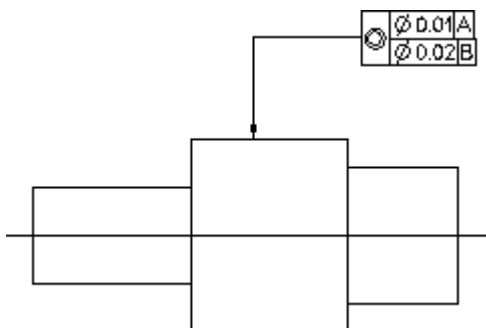
Drawing Modifiers

Select **Modifying Symbols** from the **Type** list in **Symbols** to draw any of the symbols shown in Figure 164. Modifier Geometric Symbols on page 315.

Figure 164. Modifier Geometric Symbols

Diameter	Ø	
M	(M)	At Maximum Material Condition
L	(L)	At Least Material Condition
S	(S)	Regardless of Feature Size
P	(P)	Projected Tolerance Zone

Drawing Datum Target and Revision Symbols

Select **Draws Datum Target and Revision Symbols** from the **Type** list in **Symbols** to draw any of the symbols shown in Figure 165. Target and Revision Symbols on page 315.

Figure 165. 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 291 for details).

Example 1: Single Line Symbol Group

Figure 166. Symbol Group – Example 1

To add symbol text:

1. Select **Normal** from the **Tolerance** list in **Symbol**.
2. Select **Geometric Symbols** from the **Type** list.
3. Click the Total Runout symbol on the Symbol Browser Bar.
4. Select **Modifying Symbols** from the **Type** list.
5. Click the **(S)** symbol on the Symbol Browser Bar.
6. Click **Add Text**. Enter 0.02 into the text box.
7. Click **OK** in the text box.
8. Click **|**.
9. Click **Add Text**. Enter A into the text box.
10. Click **OK** in the text box.
11. Click **|**.
12. Click **Add Text**. Enter 5R into the text box.
13. Click **OK** in the text box.
14. Click **]**.
15. Click **Apply**.
16. In the Creo Elements/Direct Drafting drawing, specify a point where you want to position the symbol.

Example 2: Multiline Symbol Group

Figure 167. Symbol Group – Example 2
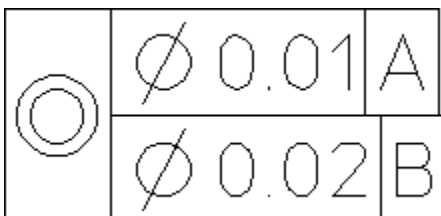
To add symbol text:

1. Select **Composite** from the **Tolerance** list in **Symbol**.
2. Select **Geometric Symbols** from the **Type** list.
3. Click the **Concentric** location symbol on the Symbol Browser Bar.
4. Select **Modifying Symbols** from the **Type** list.
5. Click the diameter symbol on the Symbol Browser Bar.
6. Click **Add Text**. Enter `0.01` into the text box.
7. Click **OK** in the text box.
8. Click **|**.
9. Click **Add Text**. Enter `A` into the text box.
10. Click **OK** in the text box.
11. Click **]**.
12. Click **–** to initiate a second line in the symbol.
13. Click the diameter symbol on the Symbol Browser Bar.
14. Click **Add Text**. Enter `0.02` into the text box.
15. Click **OK** in the text box.
16. Click `|`.
17. Click **Add Text**. Enter `B` into the text box.
18. Click **OK** in the text box.
19. Click `]`.
20. Click **Apply**.
21. In the Creo Elements/Direct Drafting drawing, specify a point where you want to position the symbol.

Example 3: Editing a Symbol Group

Symbol groups can be edited using the **Edit** command. This command 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 **Edit** in **Symbol**.
2. Click the symbol you want to edit.

   Creo Elements/Direct Drafting displays the text in screen mode, similar to the following:
   ```
   \030\151e0.01|A}
   ```

```
\030fe0.02|B}
```

3. Edit the text and select **Save** from the **File** menu in the text box to save your changes.

📝 **Note**

Use the font list for special characters created by FONT EDITOR.

Surface Symbols

📝 **Note**

The Surface Symbols and Welding Symbols menus are available in the Classical User Interface of Creo Elements/Direct Drafting only. To run Creo Elements/Direct Drafting with this user interface, set MEPELOOK to 1 or 3 in the me10.inifile.

To display the **Surface Symbols** menu, click **Menu 2** in the **SYMBOLS** menu.

Symbol Set

Click this icon to load or save a symbol set.

Symbols

Click this icon to handle (e.g. load, save or delete) individual symbols in the current symbol set.

Positioning

Click this icon to display the positioning menu for the current symbols.
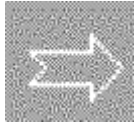
Option	Description
	The symbol is drawn horizontally from the positioning point to the right.
	The symbol is drawn vertically upwards from the positioning point.
	The symbol is drawn vertically downwards from the positioning point.
	The symbol is drawn from the positioning point (first point) towards the second point.
	The symbol is drawn from the positioning point (first point) at the specified angle.

📝 Note

The positioning point marks the foot point of the symbol.

Modifying

	Click this icon to remove, read or replace existing symbols in the drawing.

Setting Up Surface Symbols

	Click this icon to display the setup menus for surface symbols.

Symbol Setup

Menu Item	Description
File	Filename of the currently loaded symbol set or, following changes, the name of the symbol set to be loaded.
Font	Click **Font** to display a submenu for selecting the font size for surface symbols. If you prefer to use your own font size, specify the applicable number. Note that italic fonts are marked by the letter i preceding the `fnt` extension. For example, the italic version of the `hp_symbols2` font is named `hp_symbols2_i.fnt`.
Color	Click **Color** to display a submenu for selecting the color for surface symbols. Optionally, you can select RGB values.
Text Angle	Click **Text Angle** and then specify the text character slant angle.

Reference Lines

Menu Item	Description
Color	Click **Color** to display a submenu for selecting the color for reference lines. Optionally, you can select RGB values.
Width	Click **Width** to display a submenu for selecting the reference line width. Optionally, you can specify a numerical value for the reference line width. 📋 **Note** The Width applies to both reference lines and symbols.
Automatic	Click **Automatic** if you want reference lines to be added automatically (`YES`) or not (`NO`).

Reference Line Terminator

Menu Item	Description
Shape	Click **Shape** in **Reference Lines Symbol** to select a dot, an arrow, a triangle or a slash as the terminator for reference lines.
Size	Click **Size** in **Reference Lines Symbol** to display a submenu from which you can select the reference line terminator size. Optionally, you can specify a numerical value for this size.

Angle Correction
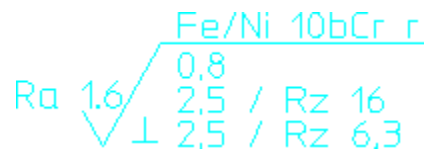
Menu Item	Description
Angle Correction	Click **Angle Correction**. Then specify the maximum deviation angle (in degrees) at which the reference line will still snap to the horizontal or vertical direction. The default value of 45 degrees implies that the reference line is always drawn horizontally or vertically. If you want to draw reference lines at arbitrary angles, set the **Angle Correction** to 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
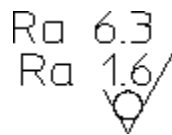
Use **Load** to load this symbol set.

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.

Figure 174. Casting plus Surface Symbols
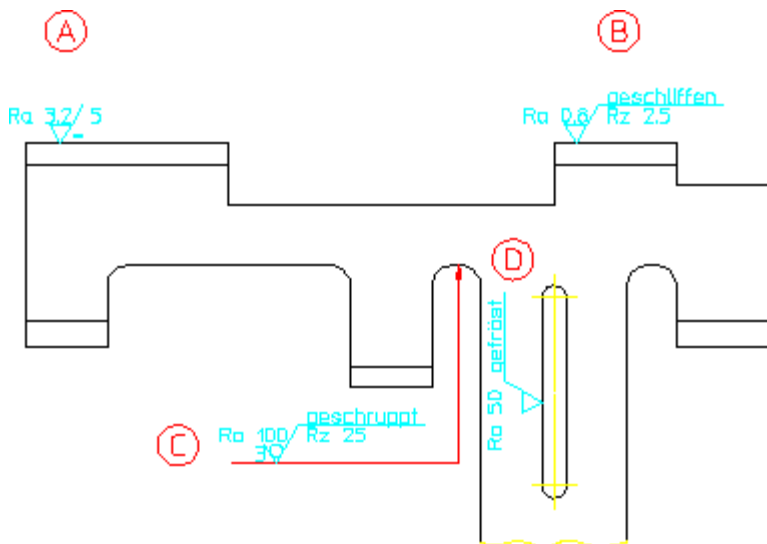
The following exercise illustrated the procedure for creating the surface symbols ((A)), ((B)), ((C)) and ((D)). This exercise assumes that the **Surface Symbols** menu is displayed.

Symbol ((A))

Proceed as follows:

1. Click **S** in the first column of the **Compose Menu** to display the current symbol definitions. Then click the following symbol shape icon:

📝 Note

The extension line running to the right is drawn only if you specify additional symbol-related data.

---

2. Click **a2** in the first column of the **Compose Menu**. Then click Ra 3.2 in the associated submenu.

3. Click **c** in the first column of the **Compose Menu**. Then click **5** in the associated submenu.

4. Click **d** in the first column of the **Compose Menu**. Then click **=** in the associated submenu.

5. Click the following icon:

You do not need to specify any positioning options because you want to draw the symbol in the default direction (from left to right).

6. Specify the positioning point (the foot point) for symbol ((A)) in your drawing.

Symbol ((B))

Proceed as follows:

1. Load the smooth surface symbol.

2. Set **d** to **EMPTY**.

3. Set **a2** to `Ra 0,8`.

4. Remove the entry for **c** by specifying a blank in apostrophes: `''`

5. Specify `Rz 2,5` (in apostrophes) for **f1**.

6. Position symbol ((B)) in your drawing.

Symbol ((C))

Proceed as follows:

1. Click the following icon:

> **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 click BLANK.

2. Set **d** to **EMPTY**.

3. Specify `Ra 100` for **a2**.

4. Specify `grind` for **b**.

5. Set **e** to **3**.

6. Set **f1** to `Rz 25`.

7. Position the symbol at a suitable point to the left of and below the area to marked for grinding.

8. To select the begin point of the reference line, click on a point to the left of the symbol foot point.

9. To select the flexion point of the reference line, click on a point close to a line perpendicular to the area to be marked for grinding.

10. To select the begin point of the reference line arrow, click on a point on the area to be marked for grinding.

11. Click **CONFIRM**.

Symbol ((D))

Proceed as follows:

1. Load the milled surface symbol.
2. Set **a1** to `Ra 50`.
3. Click on the "vertically upwards" positioning arrow icon to position symbol ((D)) in your drawing.

Welding Symbols

Note

The Surface Symbols and Welding Symbols menus are available in the Classical User Interface of Creo Elements/Direct Drafting only. To run Creo Elements/Direct Drafting with this user interface, set `MEPELOOK` to `1` in the `me10` startup file.

To display the **Welding Symbols** menu, click **MENU 3** in the **SYMBOLS** menu.

Composing a Welding Symbol

To compose a welding symbol, click on the applicable field(s) in the first column of 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

Click **Text** in the **Welding Symbols** menu to display the associated **Text** menu. 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).

Please note that the following applies to fillet weld offsets:

• Specify the following if you want the `Z` to cross the reference line:

`\v`

- If you do not want the Z to cross the reference line, specify the following:

\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
Ring	Ring weld ("weld all around")
Assembly	Assembly weld
Fork	Fork
- -	Relative position of the reference extension line. The following options are available:

above: the reference extension line is located above the reference line.

below: the reference extension line is located below the reference line.

none: no reference extension line.

Positioning Welding Symbols

Proceed as follows to position a welding symbol:

1. Click the following icon:

2. Select the applicable positioning option.
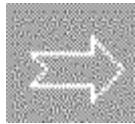
3. Click a point in your drawing where you want to add the welding symbol.
4. Draw the reference line for your welding symbol. Follow the instructions in the following section.

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.

Welds

This icon selects fillet weld symbols.
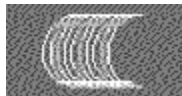
Use this function to indicated weld surfaces in your drawing.

Text Functions

Use the functions of this menu section to add text in the font size of the current drawing. Click the applicable icon and then enter the text (in apostrophes) into the user input line.

Adds text without a frame.
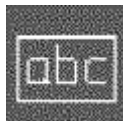
Adds text in a rectangular box.

Adds text in a balloon.

Enables you to move text added by one of these three text frame options.

Reference Line

This function enables you to manually create a reference line for the current symbol.

The reference line always runs from the object to the symbol. Proceed as follows:

1. Click the following icon in **Menu 2**:

2. The begin point of the reference line is automatically attached to the symbol. Click any points along the reference line as well as any bend points.

3. Click the end point of the reference line.

4. To complete the reference line, either:

    • Double-click on its end point, or

    • Click **CONFIRM**.

Note

If you want to break (interrupt) the reference line, click Free.
The Back option enables you to delete the last point on the reference line.

Stretching Symbols
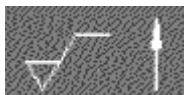
Stretches symbols vertically.

Stretches symbols horizontally.

Construction Lines
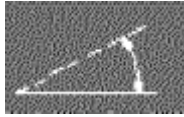
Creates a construction line at an angle.
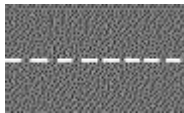
Creates a horizontal construction line.

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 319.

Weld

Menu Item	Description
Color	Click **Color** to display a submenu from which you can select the color of fillet weld and weld surface symbols. Optionally, you can specify RGB values.
Hatch Dist	Click **Hatch Dist** to display a submenu from which you can select the hatch distance for fillet weld symbols. If you want to draw fillet weldÇ symbols "filled", set this distance to 0.
Linetype	Linetype for fillet welds and weld surfaces.

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.

Figure 179. Sheet Metal Design plus Welding Symbols
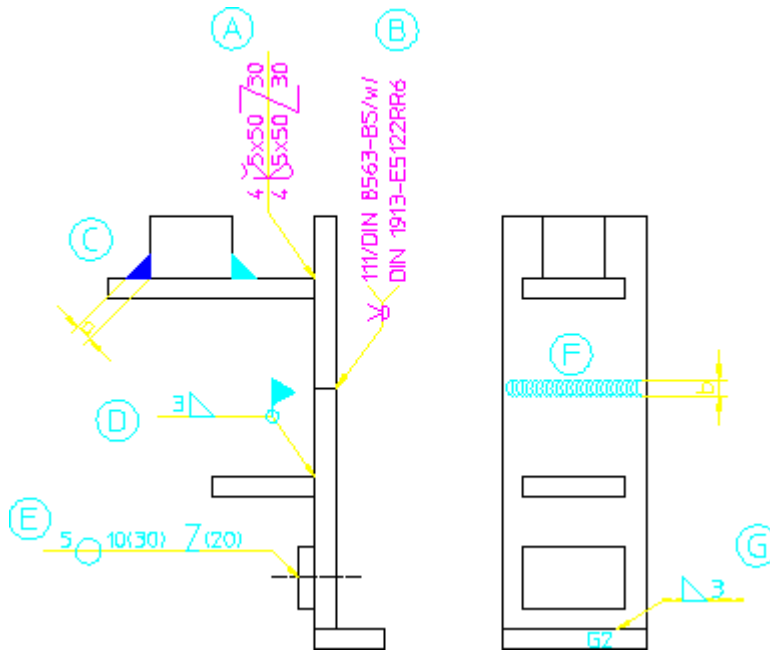
The following exercise illustrated the procedure for creating the welding symbols ((A)), ((B)), ((C)), ((D)), ((E)), ((F)) and ((G)). This exercise assumes that the **Welding Symbols** menu is displayed.

Symbol ((A))

Proceed as follows:

1. Load the `kehl_gegen` welding symbol.
2. Click the following icon:

3. Click the following positioning icon:

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 `V_back` welding symbol.
2. Position this welding symbol and its reference line in your drawing.

Symbol ((C))

1. In the **SETUP** menu for welding symbols, set the hatch distance to `0`.
2. Display **Menu 2** for welding symbols.
3. Click the following icon:

4. Specify the fillet weld width: `4`
5. Click the vertex for the "left" leg of the fillet weld.
6. Click the first leg.
7. Click the second leg.
8. Repeat these four steps to define the "right" fillet weld.

Symbol ((D))

1. Specify `3` for **T 1**.

   Set **Sy 2** to:

   ```
   delta
   ```

2. Set **R Sm** to:

3. Set **M Sm** to:

4. Position this symbol and its reference line.

Note

The SETUP menu enables you to select defaults for the size of the symbol elements.

Symbol ((E))

1. Load the `zwr_pnt_n` welding symbol.
2. Click **Text**.
3. Replace the entry for **T 1** by: 5
4. Replace the entry for **T 2** by:
   `10(30) \v (20)`
5. Click the following positioning icon:
6. Click the positioning points for the symbol.
7. To define the reference line direction, click on a point horizontally above and to the left of the symbol positioning points.
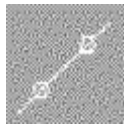
8. Click the reference line end point.

Symbol ((F))

1. Click **Menu 2** in **Welding Symbols**.
2. Click the following icon:
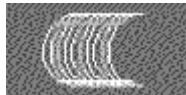
3. Click the begin point of the weld.
4. Specify the weld contour width: 5
5. Specify a weld contour spacing factor: 4
6. Click the end point of the weld.

Symbol ((G))

1. Specify 3 for **T 1**.
2. Set **Sy 2** to:

```
delta
```

3. Set **- -** to `above`.

4. Position the symbol such that its reference line runs to the left.

Grouping

You can group annotations together. Groups on page 505 may consists of a single dimension, multiple text elements and multiple symbols.

12

Handling Parts and Assemblies

This chapter shows how to use parts in your drawing.

The **Parts** menu contains the following items:

- **Edit**
- **New**
- **Create**
- **Gather**
- **Show**
- **Smash**
- **Share**
- **Unshare**
- **Symbol**
- **View**
- **Set Ref Point**
- **Scale**
- **Part List**
- **Spotlight**

Introduction to Parts

Parts are most useful in large complex drawings. Therefore, in your first few weeks using ME10 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 System Status Line. In a drawing without parts you will see the name associated with the top part (default name `Top`) displayed in the System Status Line; 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 180. Single-Level Parts Structure
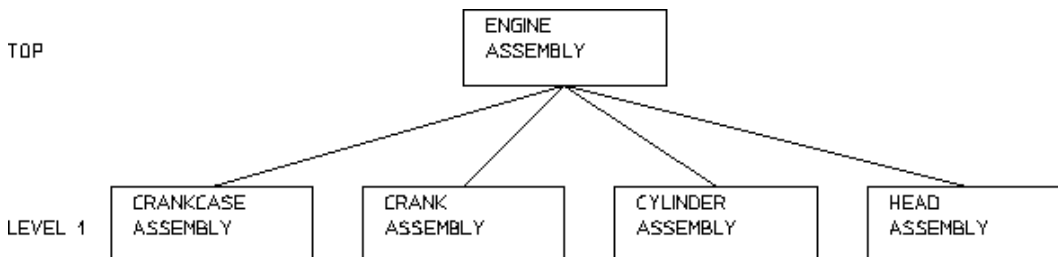
Figure 181. Multi-Level Parts Structure
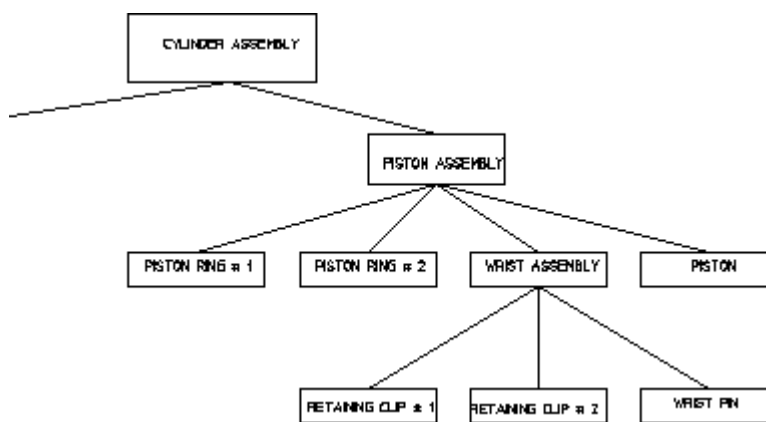
Hierarchical Parts Structure

The hierarchical structure of parts means that one part can own other parts. Refer to the CYLINDER ASSEMBLY hierarchy shown in Figure 181. Multi-Level Parts Structure on page 338. 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 Line.

📋 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** menu (for example, **Delete** and **Change**) only operate on the active part. The **Edit** command in the **Parts** menu allows you to set the active part. Thus by using the **Edit** command to make different parts active, you can place elements in different parts.

Note

The name of the active part is displayed in the System Status Line.

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 will help recognize the part.

> **📝 Note**
>
> ME10 can assign a default name to a part. The ME10 default name will be in the series **.1, .2 , .3** and so on. See .

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** menu 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** commands (such as **Symbol**) allow you to create custom symbols.

Creating an Assembly Drawing

To make full use of the ME10 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 ME10 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

shows the WRIST ASSEMBLY structure.

Figure 182. WRIST ASSEMBLY Structure
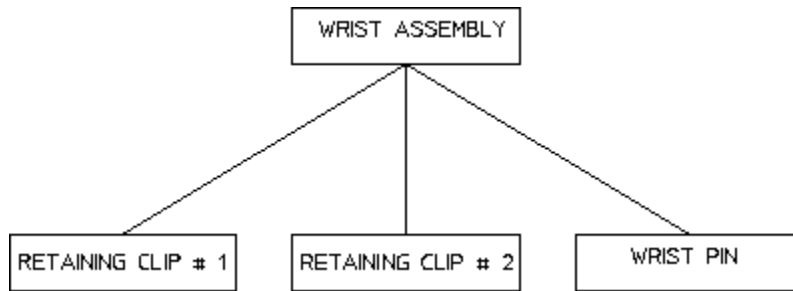
Initial Stages

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.

Load the drawing file `retain_clip1_d.mi`.

1. Click **Open** in **File**.
2. Select `retain_clip1_d.mi`.

    ME10 displays the drawing of `RETAINING CLIP #1`.

    You must now set a reference point on the drawing to enable accurate positioning in the final assembly drawing.

3. Click **Set Ref Point** in **Parts**.
4. Specify a reference point.

    Delete all unnecessary construction lines.

5. Click **Delete**.
6. Click **C-Geo** and **All** in **Select**.

    Give the drawing the part name you wish to see in the Part Browser.

7. Click **Part Browser** in **Parts**.

8. In the **Part Browser**, click on the part you want to rename.

9. Press the right mouse button to display a popup menu.

10. Select **Rename** from the popup menu.

11. Replace the current part name by `RETAINING CLIP # 1`.

    Store the new drawing (without construction lines).

12. Click **Save** in **File**.

13. Enter the new storefile name (for example `retainclip1`). Note that the `mi` suffix is appended automatically.

14. Click **Delete**, **All** and **Confirm**.

15. 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. Switch on (tick) the **Spotlight** check box option in **Parts**. This command causes the active part to have a normal appearance and all non-active parts to be shown in magenta dotted lines.

    Make sure that the ME10 top part is empty.

2. Click **Edit** in **Parts**.

3. Click **Top** in **Edit**.

4. Click **Delete**, **All** and **Confirm**.

    Rename the top part with the assembly drawing name you wish to see in the parts list.

5. Click **Part Browser** in **Parts**.

6. In the **Part Browser**, click on the Top part.

7. Press the right mouse button to display a popup menu.

8. Select **Rename** from the popup menu.

9. Replace the current name of the Top part to WRIST ASSEMBLY.

    The name of the active part in the status line changes to WRIST ASSEMBLY. Note that this part contains only the assembly name WRIST ASSEMBLY but no drawing elements.

    Now load the first part of the assembly, for example the wrist pin.

10. Click **Open** in **File**.

11. Select or specify the filename of the drawing to be loaded: `wristpin`.

    ME10 asks you to indicate a reference point to position the drawing.

12. Specify a reference point.

13. Repeat steps 3 to 6 for the drawings:

    • retainclip1.mi

    • retainclip2.mi

14. 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 ME10.

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 183. Completed Cylinder Assembly Drawing on page 344 shows how the completed cylinder assembly drawing might appear:

Figure 183. Completed Cylinder Assembly Drawing
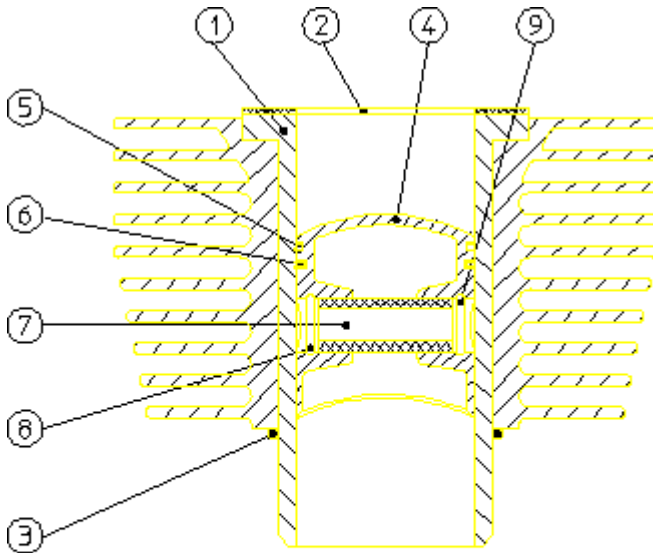
1	CYLINDER	6	PISTON RING #2
2	GASKET	7	WRIST PIN
3	BOTTOM 'O' RING	8	RETAIN CLIP #1
4	PISTON	9	RETAIN CLIP #2
5	PISTON RING #1		

Deleting Parts

Consider the CYLINDER ASSEMBLY hierarchy with PISTON ASSEMBLY active. Figure 184. Deleting Parts on page 344 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 184. Deleting Parts
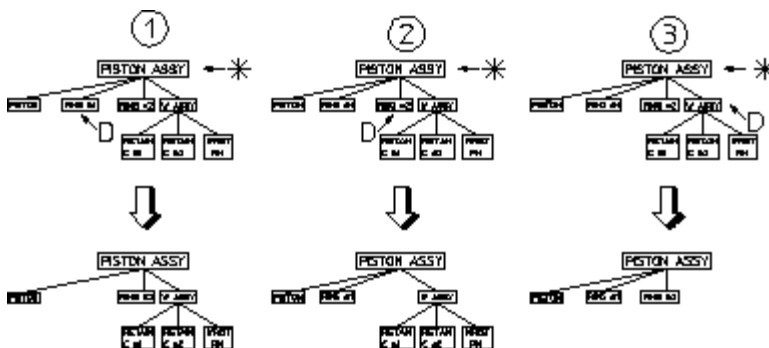
Note that in the third case:

- 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 **Edit** in **Parts**.
2. Click **Part** in **Edit**.
3. In the parts hierarchy, select the appropriate part you want to make active. Note that selection of the active part will control what is actually deleted.
4. Click **Delete**.
5. Click the part(s) you wish to delete.

Modifying Parts

Use the **Modify** menu to modify your parts.

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 **Edit** in **Parts**.
2. Click **Part** in **Edit**.
3. Select the CYLINDER ASSEMBLY.

   With CYLINDER ASSEMBLY active, you only have to identify an element in PISTON ASSEMBLY. ME10 will accept the assembly and all its subparts.
4. Complete the required move commands in **Modify**.

Note

To modify parts or elements above the active part you must click Global in Select.

Editing Parts

This section shows 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** and **Delete** commands. If you have the **Spotlight** turned on (refer to Spotlighting Parts on page 366), 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 System Status Line.

The three **Edit** options are:

- **Part**
- **Top**
- **Parent**

Editing a Part

This is the default option.

To edit a part (make a part active):

1. Click **Edit** in **Parts**.
2. Click **Part** in **Edit**.

   ME10 will prompt for the name of a part.

3. Enter the name of a part, or select an element of the part in any viewport.

   ME10 then 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 **Edit** in **Parts**.
2. Click **Top** in **Edit**.

   ME10 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 **Edit** in **Parts**.
2. Click **Parent** in **Edit**.

ME10 sets the parent of the currently active part as the active part.

Initiating a New Empty Part

This section shows 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. The **New** command options are:

* **Part**
* **Subpart**

Initiating a Part

This is the default option. 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 option. Every new part you create is automatically positioned immediately below the Top part.

To create a new empty part immediately below Top:

1. Click **New** in **Parts**.
2. Click **Part** in **New**.
3. Enter a name for the new part or allow ME10 to assign a default name.

   If you wish ME10 to assign a default name to the new part, click **End**. ME10 will generate a default name based on incremental numbers (.1, .2, .3 and so forth).

   ME10 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.
4. Place at least one element (or subpart) in the new part otherwise it will be 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**, make sure 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 **New** in **Parts**.

2. Click **Subpart** in **New**.

3. Enter a name for the new subpart.

    ME10 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 will be deleted when you change the active part status.

Creating Parts

This section shows how to make a new part from existing elements in the active part. The active part status will be unaffected after the command has been executed.

The two **Create** options are:

- **Subpart**
- **Detail**

Creating a Subpart

This is the default option. You can create a new part by transferring elements from the active part into the newly created one. The new part will be placed below the active part.

Before selecting the command make sure 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 **Create** in **Parts**.

2. Click **Subpart** in **Create**.

3. Enter the name for the new part.

    Having entered the part name, select the elements in the active part you wish to transfer to the new one.

4. When you have selected all of the elements you want to include in the subpart, click **End** (or any other command except **Cancel**) to complete the command.

   ME10 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.

Figure 185. Creating a Detail Part on page 349 shows how to create a detail drawing within your main drawing:

Figure 185. Creating a Detail Part

ENLARGED PART-SECTION A-A
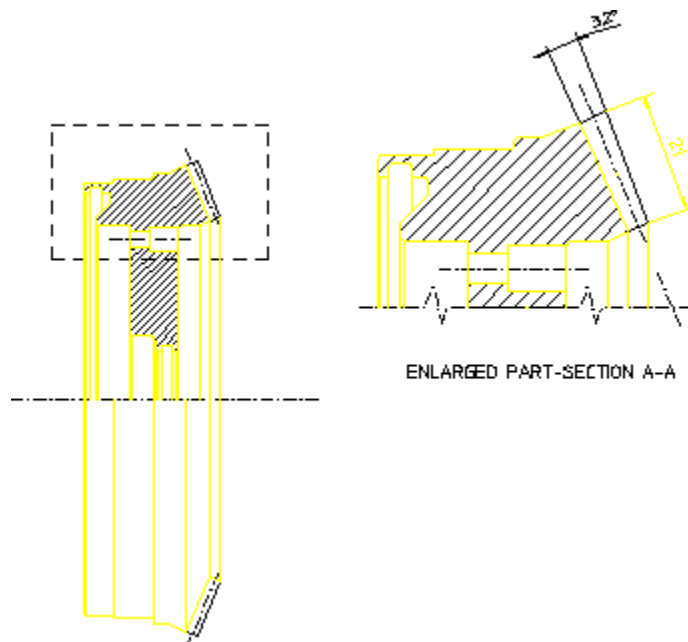
Finish drawing the part to be detailed and make it the active part.

This technique of finishing the part before creating a detail differs from traditional drafting methods. However, using **View** functions, you can just as easily draw intricate features on the main drawing as on the detail drawing.

The **Detail** option is used to produce a magnified copy for dimensioning purposes.

To produce a detail drawing:

1. Click **Create** in **Parts**.
2. Click **Detail** in **Create**.
3. Enter the required magnifying factor for the detail.
4. Box the area of the active part from which you wish to form the detail. The edges of the box will 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 copy operation, then specify the desired location for the reference point.

   ME10 now creates the detail part.

Notes on Detail Parts

- 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 ME10 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 335). 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.

📋 Note

To dimension a detail part you must make it the active part.

Gathering Elements Into Other Parts

During the drawing process, it may prove useful to transfer the ownership of existing elements in a part to the active part. Click **Element** in **Gather** to transfer elements.

If you click the **Keep Scale** check box in **Element**, the element is moved or copied to the active part, and it is drawn to the scale of 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
- Take care to identify the correct elements as mistakes can only be rectified by changing the active part and gathering the element back.
- Switch on the Spotlight check box option to see the changes made in your drawing.

To transfer elements from one part to the active part:

1. Click **Gather** in **Parts**.
2. Click **Element** in **Gather**.
3. Click **Delete Old** in **Element**.
4. Select the elements (in non-active parts) that you wish to transfer to the active part.

   As you select each element, it will be deleted from its original part and it appears in the active part.

Copy

The elements indicated will be duplicated in the active part.

To copy elements from any part to the active part:

1. Switch on the **Spotlight** check box option in **Parts**.
2. Click **Gather** in **Parts**.

3. Click **Element** in **Gather**.

4. Click **Copy** in **Element**.

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 186. Including Elements on page 352. You have incorrectly added drawing elements to PISTON RING # 1. Use **Element** in **Gather** to transfer these elements to RETAINING CLIP # 1.

Figure 186. Including Elements
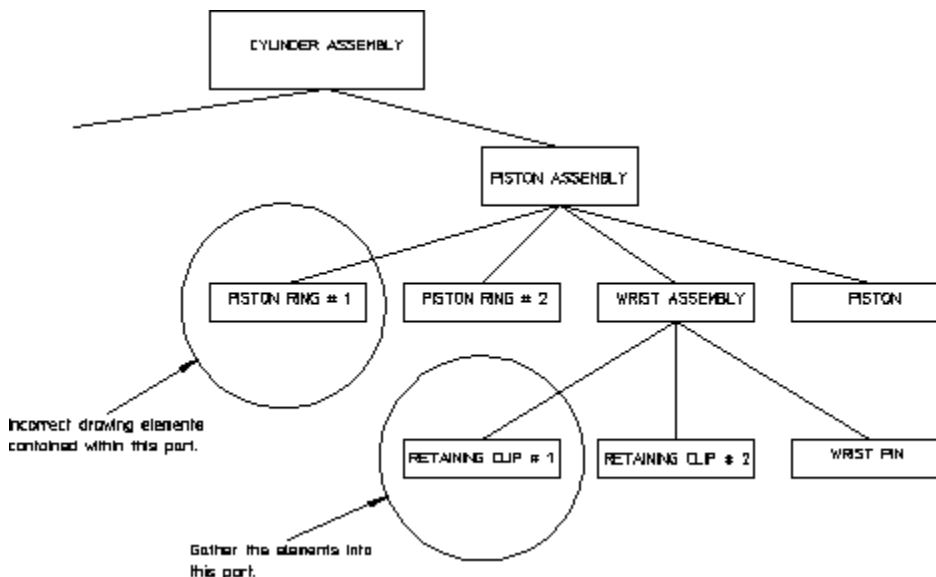
To transfer elements from PISTON RING # 1 to RETAINING CLIP # 1:

1. Click **Edit** in **Parts**.

2. Click **Part** in **Edit**.

3. Click RETAINING CLIP # 1.

   ME10 makes RETAINING CLIP # 1 the active part.

4. Click **Element** in **Gather**.

5. Click **Delete Old** in **Element**.

6. Select elements in PISTON RING # 1 you wish to transfer to the active part. As you select each element, it will be deleted from its original part and appear in the active part.

Including Dimensions in Other Parts

This section shows how to transfer the ownership of existing dimensions.

The **Gather ▸ Dimension** options are:

- **Move**
- **Move All Below**
- **Move All Above**

Move

If the reference geometry of the selected dimension is in or below the active part then the selected dimension is moved to the active part.

Move All Below

All dimensions in parts hierarchically below the active part are moved to the active part.

Move All Above

All dimensions in parts hierarchically above the active part with reference geometry in or below the active part are moved to the active part.

Including Parts into Other Parts

This section shows 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.

The **Gather ▸ Part** options are:

- **Move**
- **Copy**

Move

This is the default option.

To transfer the ownership of a part to the active part:

1. Click **Gather** in **Parts**.
2. Click **Part** in **Gather**.
3. Click **Move** in **Part**.
4. Select the part to be gathered or enter its name. You cannot gather a part from above the level of the active part.

    ME10 then 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 **Gather** in **Parts**.
2. Click **Part** in **Gather**.
3. Click **Copy** in **Part**.
4. Select the part to be included or enter its name.
5. Specify a reference point for the copy operation.
6. Specify the location for the reference point.

    ME10 copies the part such that the new part is owned by the active part.

Example

Refer to Figure 187. Including a Part on page 355. You have incorrectly created the part RETAINING CLIP # 1 as a subpart of PISTON ASSEMBLY. Use **Part** in Include to transfer RETAINING CLIP # 1 to its correct position as a subpart of WRIST ASSEMBLY.

Figure 187. Including a Part
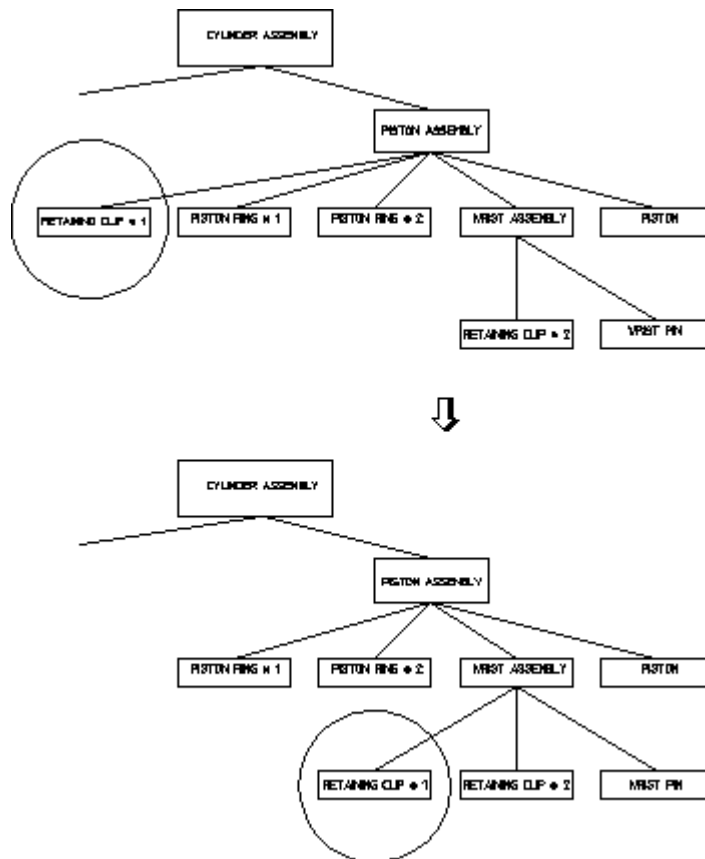
To transfer RETAINING CLIP # 1 to its correct position as a subpart of WRIST ASSEMBLY:

1. Click **Edit** in **Parts**.

2. Click **Part** in **Edit**.

3. Click WRIST ASSEMBLY.

   ME10 makes WRIST ASSEMBLY the active part.

4. Click **Move** in **Part**.

5. Click RETAINING CLIP # 1.

   ME10 transfers the part so that it becomes a subpart of WRIST ASSEMBLY.

Showing Parts

This section shows 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.

The three **Show** options are:

- **Part**
- **Part Boxed**
- **All**

Part

This is the default option. 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 **Show** in **Part**.

2. Click **Part** in **Show**.

3. Click the part.

   ME10 will redraw it normally.

However, if you wish to look down to a certain parts level and no further:

1. Enter the level number.

2. Press **[Enter]**.

   ME10 redraws the chosen viewport with all parts in the entered level and below shown as boxes.

   For example, if you enter **3** , the chosen viewport will be 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.

Part Boxed

You can change a part and all its subparts to a box representation in the chosen viewport. The box will enclose all elements of the part.

1. Click **Show** in **Part**.

2. Click **Part Boxed** in **Show**.

3. Identify the part to be boxed or enter its name.

   ME10 will redraw 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 **Show** in **Part**.
2. Click **All** in **Show**.

   ME10 immediately 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 **Parts ▸ Hide ▸ Part**.
2. Click the part.

All

To hide all parts in your drawing:

1. Click **Parts ▸ Hide ▸ All**.

You can use the **Show** command 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. Make sure that the correct part is active.
2. Click **Smash** in **Parts**.
3. Click the required **Smash** option:

Move	The subpart is merged with the active part.
Move KeepData	The subpart is merged with the active part, and it is drawn to the scale of the top part (scaling is maintained – drawing A).
Move 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 enter the part name. The part to be smashed must belong to the active part.

ME10 will now transfer the elements of the identified part to the active part and delete the subpart.

Example

Refer to Figure 188. Smashing Parts on page 358. You want to make a single part drawing (WRIST UNIT) from the assembly drawing WRIST ASSEMBLY.

Figure 188. Smashing Parts
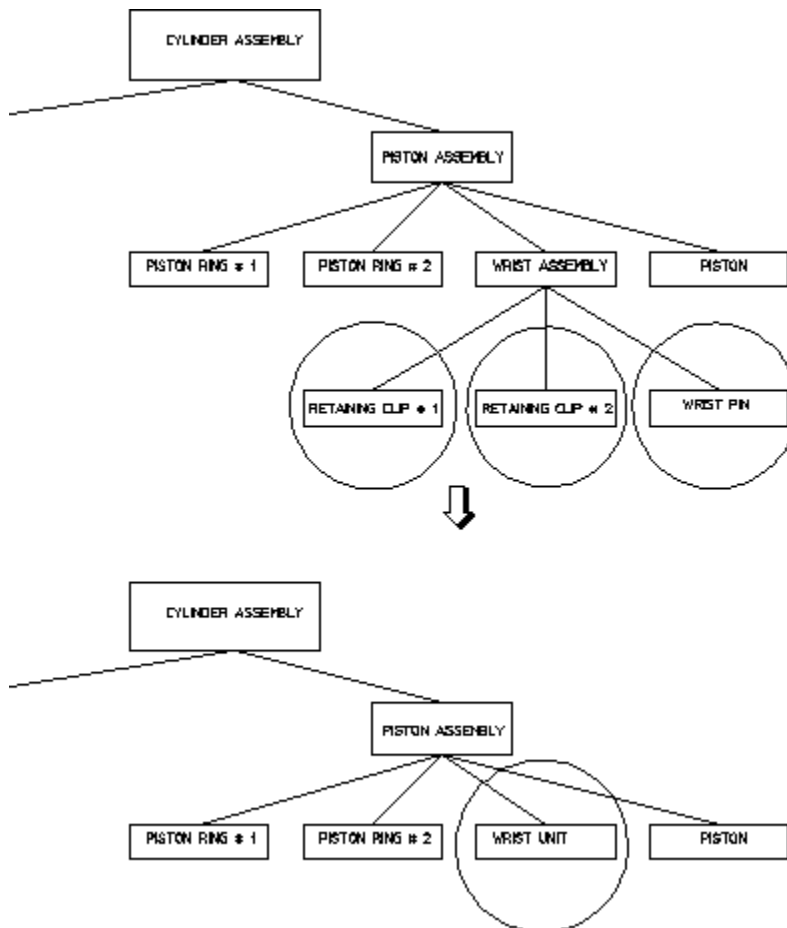
To create WRIST UNIT from WRIST ASSEMBLY:

1. Click **Edit** in **Parts**.

2. Click **Part** in **Edit**.

3. Click WRIST ASSEMBLY.

   ME10 makes WRIST ASSEMBLY the active part.

4. Click **Smash** in **Parts**.

5. Click RETAINING CLIP # 1, RETAINING CLIP # 2 and WRIST PIN.

ME10 transfers all drawing elements to the active part and then deletes the subparts.

6. In the Part Browser, press the right mouse button, select **Rename** and rename the part to WRIST UNIT.

Figure 189. Effects of Smashing Scaled Parts
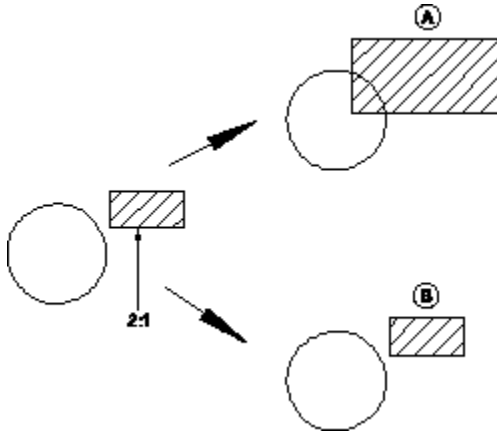
📝 **Note**

- 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** command.

You can also determine the plane in which the shared part lies by using the **Isometric** command from the **Modify** menu. If you place a shared part in a different plane, make it active and then turn on 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 **Share** in **Parts**.

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**.

2. Click **Move** in **Modify**.

3. Click **Multiple** in **Move**.

4. Select the shared part to be copied.

5. Specify a reference point for the move.

6. 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.

	Uniq	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:

	Uniq	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 for further information.

📝 Note

Any changes made to a shared part will 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 will be given the new name.

Unsharing Parts

This section shows 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 will be altered simultaneously.

📋 Note

You cannot unshare a part above the active part or at the same level.

To unshare a part:

1. Click **Unshare** in **Parts**.

2. Select the part to be unshared. The active status does not affect the command except that the part you wish to unshare cannot be active.

   ME10 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.

Part Browser

To display the Part Browser, click **Part Browser** in **Parts**. To perform an action on a given part, click on that part in the **Part Browser**, then press the right mouse button to display a popup menu, and select an option from this menu. Note that the Part Browser can be docked anywhere in the ME10 window.

What is Shown in the Part Browser?

The Part Browser provides a graphical overview of the current parts structure in your ME10 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 on the **+** (plus) sign icon to expand the tree.

- Click on the **-** (minus) sign icon to collapse the tree.

- Tick the (checkbox-like) state icon to the left of any part icon to view the associated part (and any parts hierarchically below this part).

- Untick the (checkbox-like) state icon 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 colums 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 Popup Menu

To display the Part Browser tree popup menu, press the right mouse button on any part icon in the Part Browser. This popup menu provides the following commands (some of which are also available via the **Parts** menu):

View	Shows only the selected part. ME10 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 357 for further information.
Share	Creates a shared part. See Sharing Parts on page 359 for further information.
UnShare	Unshares the selected (shared) part. See Unsharing Parts on page 362 for further information.
Symbol	Changes the part to a symbol. See Changing Parts into Symbols on page 366 for further information.
Show	In addition to the options explained in Showing Parts on page

	355, 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 357, 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 an ME10 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**, click on the part you want to rename.
2. Press the right mouse button to display a popup menu.
3. Select **Rename** from the popup menu.
4. Replace the current part name by the new name.

Note

When you rename a shared part, all occurrences of that shared part will be given the new name.

Viewing Parts

This section shows how to view individual parts of your drawing.

You can only view drawings in the current viewport.

Use viewing in the following cases:

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- 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(s) you are currently working on.

The three **View** options are:

- **Part**

- **Parent**

- **Top**

Part

To look at an individual part:

1. Click **View** in **Parts**.

2. Click **Part** in **View**.

3. Enter the name of the part to be viewed, or click an element of the part in any viewport.

   ME10 then redraws the chosen viewport containing only the desired part and any subparts belonging to it. The name of the part being viewed will be 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 **View** in **Parts**.

2. Click **Parent** in **View**.

   The current viewport will be redrawn displaying the parent of the active part and all parts belonging to the active part.

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:

1. Click **View** in **Parts**.

2. Click **Top** in **View**.

The chosen viewport will be redrawn to show all the parts of your drawing. The name of the top part will be displayed in the top left corner of the viewport.

Spotlighting Parts

Spotlighting allows you to distinguish between the active part and all other parts. The function works as an on/off switch. We recommend you switch on **Spotlight** (by ticking its check box) box when you are creating and modifying parts.

1. Switch off the **Spotlight** check box option in **Parts**.

   All parts appear in their normal linetype and color, this makes it difficult to distinguish between the active part and all other parts.

2. Alternatively, switch on the **Spotlight** check box option in **Parts**.

   Only the active part has its normal appearance. All non-active parts are shown in magenta dotted lines.

Changing Parts into Symbols

This section shows 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 draw scale (refer to Preparing the Drawing Environment on page 29).

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. This is best done with the draw scale set to 1.

2. When the part is finished, click **Symbol** in **Parts**, then click **On** in **Symbol**.

3. Select the part or enter its name.

   ME10 now defines the part as a symbol part. Check this by viewing the Part Browser. clicking **Part List**. You will see a small "x" in the **Symbol** column.

You can also change a symbol part back into a normal part:

1. Click **Symbol** in **Parts**.

2. Click **Off** in **Symbol**.

3. Select the symbol part or enter its name.

   ME10 will now change the symbol part back into 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 Include commands.

The ME10 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 **Set Ref Point** in **Parts**.

    The present reference point for the active part will be shown by a line joined to the cursor.

2. Specify the new reference point.

Multiple Drawing Scales

ME10 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.
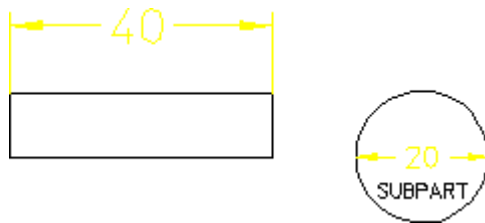
2. Look at the Part Browser. It should look similar to the following:
    ```
    Top          ~1    1:1
      SUBPART    ~2    1:1
    ```

3. Click **Scale** in **Parts**.

4. Click **Prt + Subprt** in **Scale**.

5. Select the top part (both parts are highlighted).

6. Enter **2**.

    ME10 scales both the top part and subpart by the scaling factor 2.
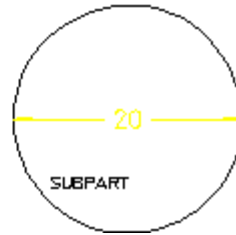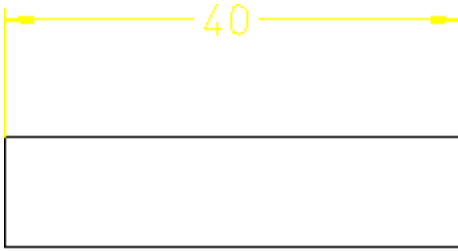
Now look at the parts tree again to see how the scaling factors change:

```
Top        ~1    2:1
  SUBPART  ~2    2:1
```

1. Click **Scale** in **Parts**.

2. Click **Prt Only** in **Scale**.

3. Select the subpart (it does not need to be active).

4. Enter **3**.

    ME10 scales the subpart only.
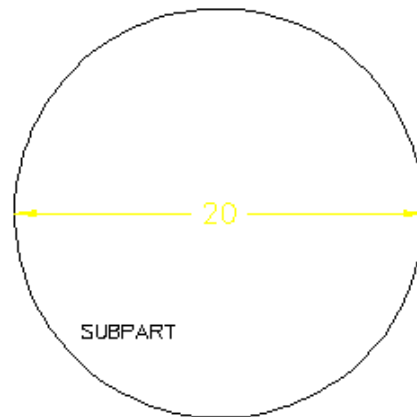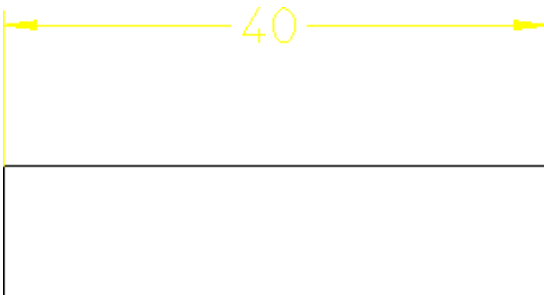
Now look at the parts tree again to see how the scaling factors change:

```
Top        ~1    2:1
  SUBPART  ~2    3:1
```

Notice that the subpart is scaled according to its original value. In others words, it is scaled by a factor 3 — not 2 and then 3.

Text, dimensioning text, and symbols are not changed.

Scaling Parts about a Reference Point

The **Set Ref Point** option of the **Scale** command in **Parts** scales parts and subparts about one of the following:

- **Ref Pnt**
- **Origin**
- **Center**

Unique Partnames

ME10 automatically generates names for parts when you create them. These names are shown in the parts tree and are represented by numbers: ~1,~2, ...

This feature is useful when handling parts by macros. For example, a typical parts tree may look like this:
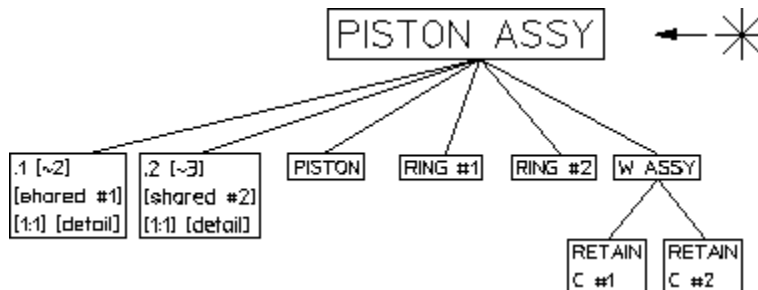
Unique parts naming has the following functionality:

- 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, ME10 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 entering the command:
  RESET_PART_NUMBER

Further Information

For further technical information about parts, the parts structure and so on, refer to the ME10 help system:

Enter `help parts` at the ME10 command line.

The screen will clear and ME10 will display further information about parts.

13

Generating Hidden Lines

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 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 Mode** and **On** — this mode effectively redraws your assembly drawing, hiding any hidden lines and faces.

• Click **Hidden Mode** and **Off** — 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**

Click **Backup** in Change to Hidden to store your assembly drawing (in `workfile.hid`) before generating hidden lines. The Change to Hidden command smashes the parts structure and creates 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). You can delete any covering face by clicking **Delete Face** in **Edit** and then clicking the face. Note that 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 194. Faces — With Z-Values 0 and 1 on page 373 shows the faces of two components (one with z-value 0 and the other with z-value 1).

Figure 194. Faces — With Z-Values 0 and 1
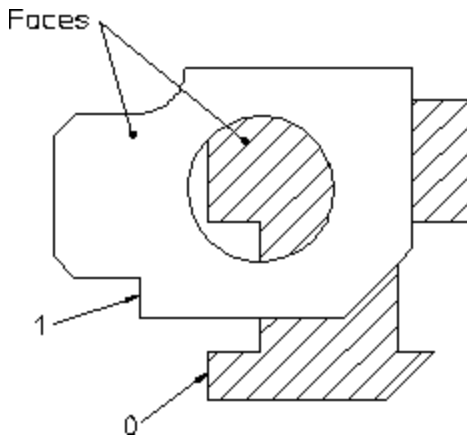
> 📝 **Note**
>
> A face does not necessarily need to have the same z-value as its boundary lines. In this case the boundary lines of the face may be partially covered by the face itself (solid lines may appear dashed in some cases). Note, the linetype of the boundary lines is not changed.

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 System Status Line.

Figure 195. Visualizing Z-Values
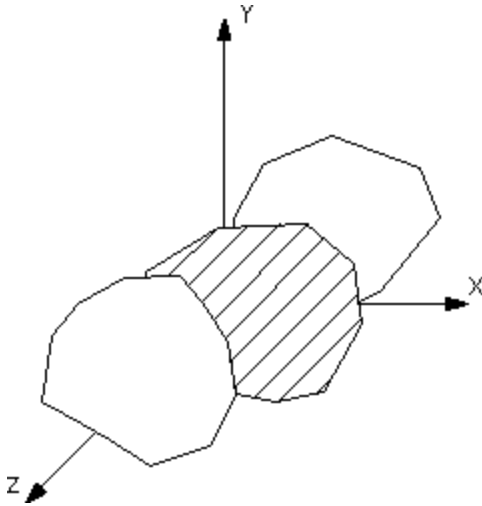
For example, three faces could be given the z-values 0, 1, 2. The face with z-value 1 would be positioned above that with z-value 0. The face with z-value 2 would be positioned above those with z-values 1 and 0. Depending on where these faces were positioned with respect to each other, certain areas would be obscured.

shows 3 faces assigned the z-values 0, 1 and 2.

Figure 196. Z-Values 0, 1 and 2
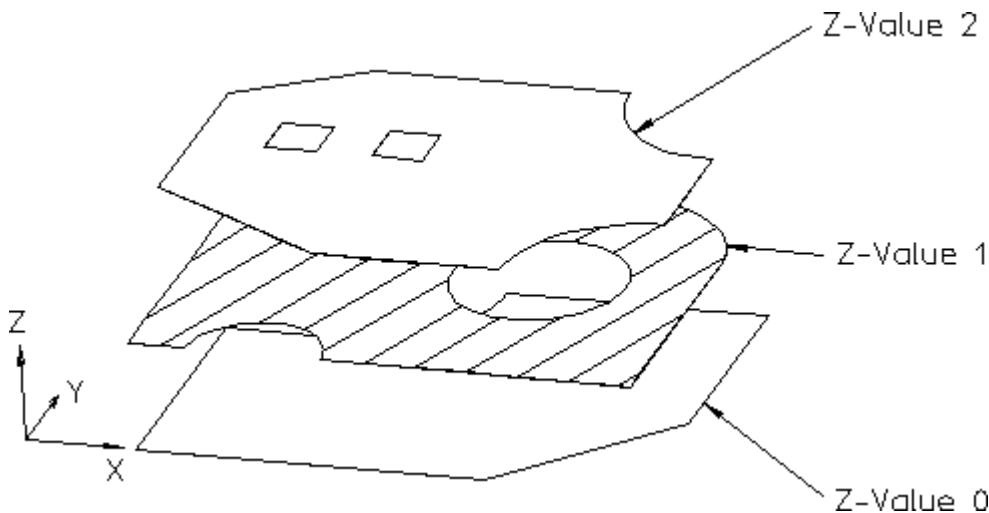
Hatches are treated as faces. This means hatched areas can cover geometry and faces. Hatches that are partially covered are displayed partially obscured.

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 375).

You can specify z-values in three ways:

- Automatically — if you have clicked **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 197. Z-Value Structure within a Drawing on page 375 shows how the z-value structure relates to your drawing:

Figure 197. Z-Value Structure within a Drawing
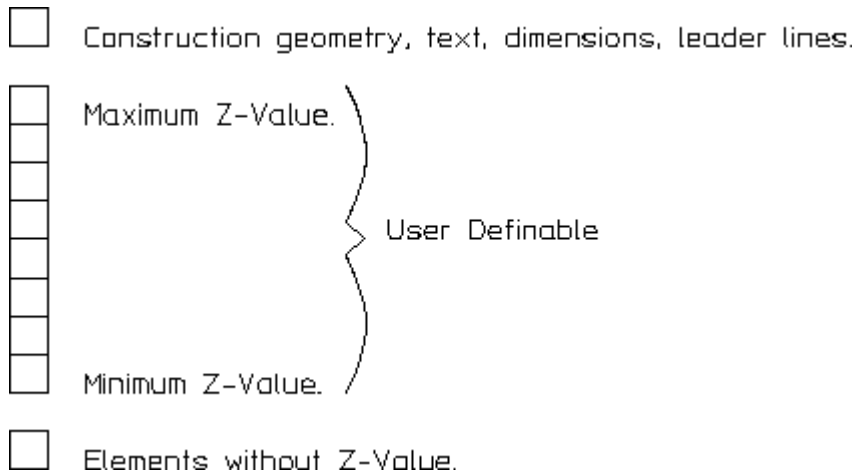
Z-Values and Shared Parts

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.

The hidden lines can be switched on and off by clicking **Hidden Mode ▶ On** or **Hidden Mode ▶ Off** in **Hidden**.

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 **Delete ▶ All ▶ Confirm** to clear the screen.
2. In the Part Browser, click on the Top part, press the right mouse button, select **Rename** and rename the top part to:

```
Frame Assembly
```

Drawing Frame 1

1. Click **New** in **Part**.

2. Click **Part** in **New**.

3. Enter `` `Frame 1' `` as the part name.

4. Click **Generate Element** in **Hidden**.

5. Click **Current Z Value** in **Generate Element** and enter **1**

6. Click **Line** in **Geometry**.

7. Click **Rectangle** in **Line**.

8. Draw two rectangles as shown in

Figure 198. Frame 1
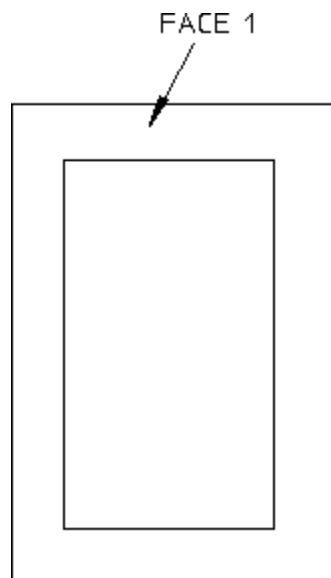
Generating Face 1

1. Click **Generate Face** in **Hidden**.

2. Click **Single Automatic** in **Generate Face**.

3. Click Face 1 (see).

   Creo Elements/Direct Drafting creates Face 1.

4. Click **End**.

Drawing Frame 2

1. Click **New** in **Part**.

2. Click **Part** in **New**.

3. Enter `Frame 2` as the part name.

4. Click **Generate Element** in **Hidden**.

5. Click **Current Z Value** in **Generate Element** and enter **2**

6. Click **Line** in **Geometry**.

7. Click **Rectangle** in **Line**.

8. Draw two more rectangles as shown in .

Figure 199. Frame 1 and Frame 2
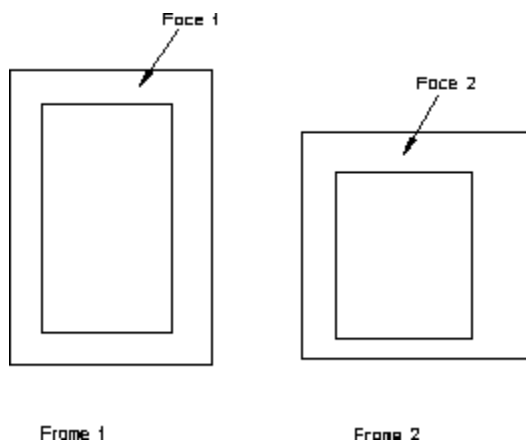
Generating Face 2

1. Click **Generate Face** in **Hidden**.

2. Click **Single Automatic** in **Generate Face**.

3. Click `Face 2`.

4. Click **End**.

Positioning Frame 2 Above Frame 1

1. Switch off **Keep** in **Modify**.

2. Select Frame 2 by enclosing it in a box.

3. Click **2 Points** in **Move**.

4. Specify a reference point for the move.
5. Reposition Frame 2 above Frame 1 as shown in Figure 200. Frame 2 Above Frame 1 on page 379.

Figure 200. Frame 2 Above Frame 1

Frame 1   Frame 2
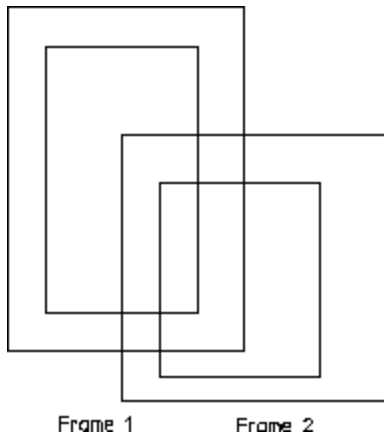
Assembly — Hidden Lines Invisible

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 Mode On** in **Hidden**.
2. Click the current viewport.

    To confirm hidden mode, Creo Elements/Direct Drafting displays -H- in the top left hand 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 201. Assembly View with Hidden Lines Invisible on page 380.

Figure 201. Assembly View with Hidden Lines Invisible

Frame 1   Frame 2
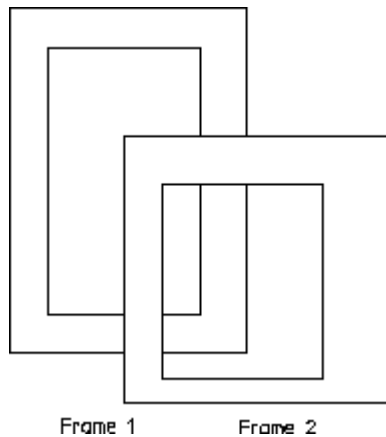
Assembly — Hidden Lines Shown Dashed

At this point you will generate the actual hidden lines.

⚠ **Caution**

Click **Backup** in Change to Hidden to save your assembly drawing before generating hidden lines. The Change to Hidden command smashes the parts structure and creates the hidden line geometry.

---

1. Click **Edit** in **Parts**.
2. Click **Top** in **Edit**.
3. Click **Backup** in **Change to Hidden**.
4. 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 202. Assembly with Hidden Lines Shown Dashed on page 381.

Figure 202. Assembly with Hidden Lines Shown Dashed
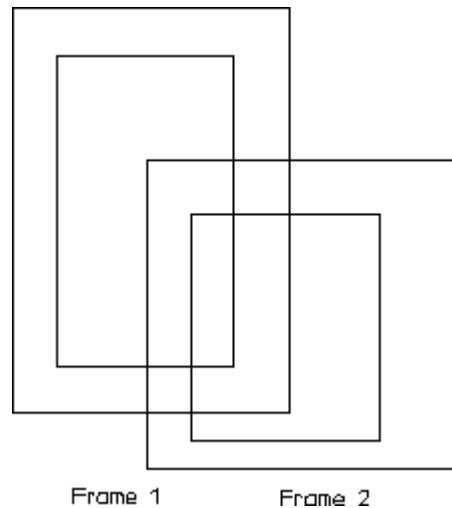
Setting the Current Offset

The current offset in the z-direction is used to calculate the z-value used in the above/below/between relationship with another element. The current offset z-value is added to/subtracted from the z-value of a reference element.

To set the current offset:

1. Click **Generate Element** in **Hidden**.

2. Click **Current Offset** in **Generate Element**.

3. Enter the current offset z-value.

Setting Z-Values

This section shows how to assign and change element z-values.

The **Generate Element** options for assigning z-values are:

• **Element**

• **Shared Part**

Use the normal Creo Elements/Direct Drafting selection techniques to select the elements to which you want to assign z-values.

If you select a part and you have clicked **Element** in **Generate Element**, all elements in that part are assigned the specified z-value. This works recursively down to all subparts.

Additional options for the above command can be selected from **Options** in **Hidden**:

- **Above**
- **Below**
- **Same**
- **Between**
- **None**

You can set a z-value by clicking **Current Z Value** in **Generate Element** and entering a numerical value; this 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 **Element** in **Generate Element**.
2. Select the elements to which you wish to assign z-values.
3. Enter KEEP via the keyboard.
4. Enter a numerical value.

You can also click one of the options **Above**, **Below**, **Same**, or **Between**. This will result in a computed z-value.

📝 Note

These options 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 the options **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**) 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** in **Options**, **Current Offset** = 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** in **Options**, **Current Offset** = 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** in **Options**, **Current Offset** = 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 **Generate Element** in **Hidden**.
2. Click **Element** in **Generate Element**.
3. Select the element to which you want to assign a z-value.
4. Enter the z-value.

Assigning a Z-Value to a Shared Part

To assign a z-value to a shared part:

1. Click **Generate Element** in **Hidden**.
2. Click **Shared Part** in **Generate Element**.
3. Select the shared part to which you want to assign a z-value.
4. Enter the z-value.

Deleting a Z-Value

To delete a z-value of an element or a shared part:

1. Click **Generate Element** in **Hidden**.

2. Click **Element** or **Shared Part** in **Generate Element**.

3. Select the element(s) or the shared part.

4. Click **None** in **Options**.

Positioning an Element at a Z-Value Above Another Element

To position an element at a z-value above another element:

1. Click **Generate Element** in **Hidden**.

2. Click **Element** in **Generate Element**.

3. Select the elements to be set above the reference element.

4. Click **Above** in **Options**.

5. 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 **Generate Element** in **Hidden**.

2. Click **Element** in **Generate Element**.

3. Select the elements to be set below the reference element.

4. Click **Below** in **Options**.

5. Select the reference element.

Assigning an Element's Z-Value to Another Element

To set the same z-value:

1. Click **Generate Element** in **Hidden**.

2. Click **Element** in **Generate Element**.

3. Select the elements to be set to the same level as the reference element.

4. Click **Same** in **Options**.

5. 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 **Generate Element** in **Hidden**.

2. Click **Element** in **Generate Element**.

3. Select the elements to be set to a level between two existing elements.

4. Click **Between** in **Options**.

5. Select the lower reference element.

6. Select the upper reference element.

Generating Faces and Setting Z-Values in a Complete Part

This section shows you how to easily generate faces and set z-values in a complete part. Using the **With Holes** option, you can detect the presence of holes to prevent them being generated as faces.

The **Generate Part** options are:

• **Elements & Faces**

• **With Holes**

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 **Generate Part** in **Hidden**.

2. Click **Elements & Faces** in **Generate Part**.

3. Select the part.

4. Enter the z-value or click **Above**, **Same**, or **Below** in **Options**.

5. Click **End**.

   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 **Generate Part** in **Hidden**.

2. Click **With Holes** in **Generate Part**.

3. Select the part.

4. Enter the z-value or click **Above**, **Same**, or **Below** in **Options**.

5. Click **End**.

   Creo Elements/Direct Drafting generates all faces ignoring any holes and assigns z-values to all elements.

Checking Z-Values

This section shows how to check z-values.

An Element's Z-Value

To check the z-value of an individual element:

1. Click **Generate Element** in **Hidden**.

2. Click **Inquire Z Value** in **Generate Element**.

3. Click **Absolute** in **Inquire Z Value**.

4. 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 into 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

Refer to Figure 203. Z-Values for Shared Parts on page 387.

To check the computed z-value of an individual element with z-value 2 in Shared_part1:

1. Click **Generate Element** in **Hidden**.
2. Click **Inquire Z Value** in **Generate Element**.
3. Click **Absolute** in **Inquire Z Value**.
4. 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 203. Z-Values for Shared Parts
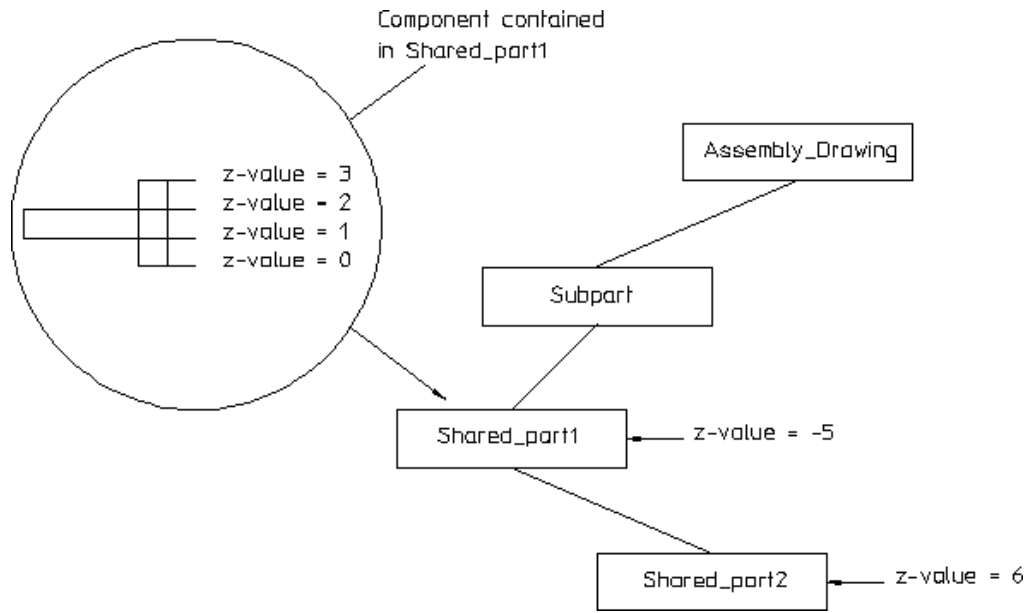
The Minimum Z-Value in an Assembly Drawing

To check the minimum z-value:

1. Click **Generate Element** in **Hidden**.
2. Click **Inquire Z Value** in **Generate Element**.
3. Click **Minimum** in **Inquire Z Value**.

   Creo Elements/Direct Drafting displays the minimum z-value in the assembly drawing on the User Input Line (if the drawing has no z-values assigned, an error beep is given). The value is also written into the System Inquiry Array.

The Maximum Z-Value in an Assembly Drawing

To find the maximum z-value:

1. Click **Generate Element** in **Hidden**.
2. Click **Inquire Z Value** in **Generate Element**.
3. Click **Maximum** in **Inquire Z Value**.

   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 into the System Inquiry Array.

The Z-Value of a Shared Part

To find the computed z-value of a shared part:

1. Click **Generate Element** in **Hidden**.
2. Click **Inquire Z Value** in **Generate Element**.
3. Click **ShPrtOffset** in **Inquire Z Value**.
4. 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 into the System Inquiry Array.

Example

Refer to Figure 203. Z-Values for Shared Parts on page 387.

To check the z-value of Shared_part2:

1. Click **Generate Element** in **Hidden**.
2. Click **Inquire Z Value** in **Generate Element**.
3. Click **ShPrtOffset** in **Inquire Z Value**.
4. 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 **Generate Element** in **Hidden**.
2. Click **Inquire Z Value** in **Generate Element**.
3. Click **ShPrt Elem** in **Inquire Z Value**.
4. 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 into the System Inquiry Array.

The Current Z-Value

To check the default current z-value assigned to each newly created drawing element:

1. Click **Generate Element** in **Hidden**.

2. Click **Inquire Z Value** in **Generate Element**.

3. Click **Curr Z-Val** in **Inquire Z Value**.

   Creo Elements/Direct Drafting displays the current z-value in the User Input Line. If required, it can be used as input without retyping. The current z-value is also written into the System Inquiry Array.

The Load Value

To check the current load z-value:

1. Click **Generate Element** in **Hidden**.

2. Click **Inquire Z Value** in **Generate Element**.

3. Click **Load Z-Val** in **Inquire Z Value**.

   Creo Elements/Direct Drafting displays the current load mode and load z-value on the User Input Line. If required, it can be used as input without retyping. Both these values are written into the System Inquiry Array.

Setting the Default Z-Value

This section shows how to define 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:

1. Click **Generate Element** in **Hidden**.

2. Click **Current Z Value** in **Generate Element**.

3. Enter the new z-value into the User Input Line or click **Above**, **Same**, or **Below** in **Options**.

   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 shows how to define 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:

1. Click **Generate Element** in **Hidden**.
2. Click **Load Z Value** in **Generate Element**.
3. Enter the required load offset z-value into the User Input Line. Alternatively, click a reference element set to a given z-value and then click **Above**, **Same**, or **Below** in **Options**.

   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:

1. Click **Generate Element** in **Hidden**.
2. Click **Load Z Value** in **Generate Element**.
3. Enter 2.
4. Load an existing drawing file.

   Creo Elements/Direct Drafting loads the drawing elements as shown in Figure 204. Loading Drawing Elements with a Load Z-Value of >=2 on page 390.

Figure 204. Loading Drawing Elements with a Load Z-Value of >=2
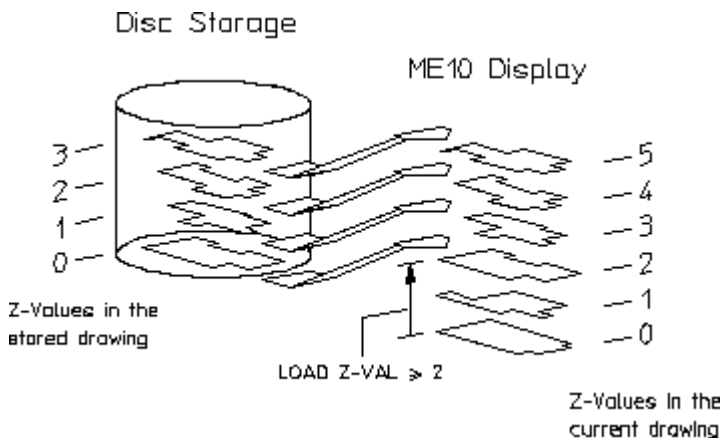
Generating Faces

This section shows 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).

The **Generate Face** options are:

*   **Single Automatic**
*   **Single Manual**
*   **All in Part**
*   **With Holes**

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 **Generate Face** in **Hidden**.
2. Click **Single Automatic** in **Generate Face**.
3. Specify a point inside the face.
4. Enter a z-value or click **Above**, **Same** or **Below** in **Options**.

    Creo Elements/Direct Drafting generates the face automatically and assigns the required z-value.

Manually

Note that 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 **Generate Face** in **Hidden**.
2. Click **Single Manual** in **Generate Face**.

3. First specify points along the outer contour (the points selected do not have to lie on any geometric element).

4. Specify points along the inner contour.

5. Click **End**.

6. Enter a z-value or click **Above**, **Same** or **Below** in **Options**.

All Faces in a Part Excluding Holes

To automatically generate all faces in a part and exclude any holes:

1. Click **Generate Face** in **Hidden**.

2. Click **With Holes** in **Generate Face**.

3. Enter a z-value or click **Above**, **Same** or **Below** in **Options**.

   Creo Elements/Direct Drafting generates the faces automatically and assigns the required z-values. Note that 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:

1. Click **Generate Face** in **Hidden**.

2. Click **All in Part** in **Generate Face**.

3. Enter a z-value or click **Above**, **Same** or **Below** in **Options**.

Example

Refer to Figure 205. Automatically Generating Faces In a Part on page 393. In this example Creo Elements/Direct Drafting would generate the four faces shown.

Figure 205. Automatically Generating Faces In a Part
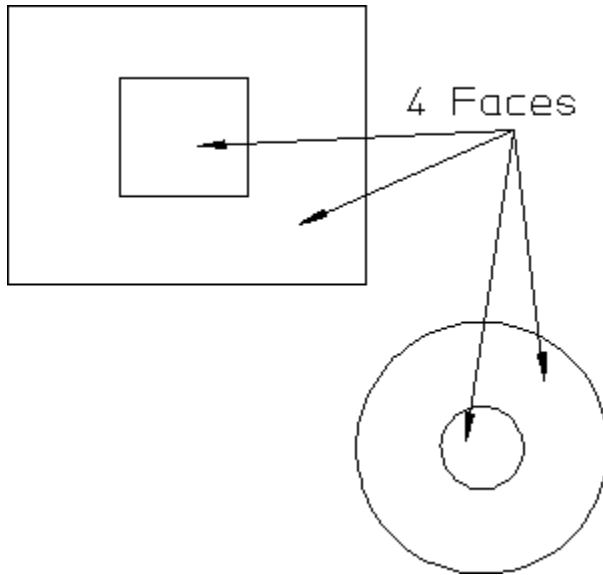
Undoing a Face

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 **Generate Face** options is:

Single Automatic	The last face specified will be removed.
Single Manual	During creation of a contour, points will be removed. If you have finished defining a face and have clicked **End**, pressing **Undo** results in the removal of the entire face.
All in Part	All faces generated automatically will be removed.
With Holes	All faces generated automatically will be 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 **Edit** in **Hidden**.

2. Click **Delete Face** in **Edit**.

3. 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 **Edit** in **Hidden**.

2. Click **Delete Face** in **Edit**.

3. Click **All**, **Confirm** and **Apply** in **Options**.

   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 Mode** and **On** in **Hidden**.

2. Specify the appropriate 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 that Creo Elements/Direct Drafting displays "-H-" in the upper left-hand corner to indicate hidden mode.

Normal Mode

This is the default mode. To redraw normally:

1. Click **Hidden Mode** and **Off** in **Hidden**.

2. Specify the appropriate viewport.

Hidden Settings

Proceed as follows to define the required hidden line type, line color and face color settings:

1. Click **Settings** in **Hidden**.

2. In the **Hidden Settings** dialog box, pull down the selection list for each of the following settings:

Line Type	Select the line type for hidden lines in your drawing.
Line Color	Select the color for hidden lines in your drawing. When the **Default** check box is ticked, the default hidden line color will be applied. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.
Color	Select the color for faces in hidden mode. When the **Default** check box is ticked, the default face color will be applied. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

3. Click **OK**.

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 371 for instructions on generating hidden lines). To change the linetype and/or color of existing hidden lines:

1. Click **Edit** in **Hidden**.

2. Click **Change Line** in **Edit**.

3. In the **Change Hidden Line** dialog box, select a linetype from the **Type** selection list.

4. Select a line color from the **Color** selection list. Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

5. Click **OK**.

   All existing hidden lines in the drawing will be 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.

1. Click **Edit** in **Hidden**.

2. Click **Change Face** in **Edit**.

3. In the **Change Hidden Face** dialog box, select a color from the **Color** selection list. If you tick the **None** check box, the face color is set to none (i.e. faces will be transparent). Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

4. Click **OK**.

5. Select the face(s) to be changed.

   The selected face(s) will be redrawn in the color you selected.

Viewing Z-Values

The **ShowRange** 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.

The **ShowRange** options are:

*   **Elements in Drawing**
*   **Faces in Drawing**
*   **Elements in Part**
*   **Faces in Part**

To display the **Show Range** dialog box:

1. Click **Show** in **Hidden**.

2. Click **Range** in **Show**.

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. Click the **Limit** radio button in the **Range** section of the **ShowRange** dialog box.

2. Specify the "from" z-value in the **Lower** field.

3. Specify the "to" z-value in the **Upper** field. For example, to limit the visible range to z-levels 3 through 8, you would enter **3** into the **Lower** field and **8** in the **Upper** field.

This range is used in the following way:

- In connection with the options **Elements in Drawing** and **Elements in Part**, this range specifies the range of z-values belonging to elements that are visible.

- In connection with the options **Faces in Drawing** and **Faces in Part**, this range specifies the range of z-values belonging to faces that are visible and colored.

Elements in All Parts

Click the **Elements in Drawing** radio button to select all elements:

- In a selected range of z-values.
- In all parts (default).

Elements in Current Part

Click the **Elements in Part** radio button to select elements:

- In a selected range of z-values.
- In the current part.

Faces in all Parts

Click **Faces in Drawing** to select faces:

- In a selected range of z-values.
- In all parts.

Faces in Current Part

Click **Faces in Part** to select faces:

- In a selected range of z-values.
- In the current part.

Applying a Color to Facilitate Visualization

To color faces in a particular range of z-values as an aid to visualization:

1. In the **ShowRange** dialog box, click the appropriate radio button for the items to be displayed.

2. In the **Range** section, specify a the range of z-values to be displayed in the same color (see Setting a Range of Z-Values on page 396).

3. In the **Show** section of the **Show Range** dialog box, click the **Color** radio button and select a color from the **Color** list. Refer to Defining Custom Colors on page 59 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 206. Viewing Selected Elements on page 398.

   Figure 206. Viewing Selected Elements
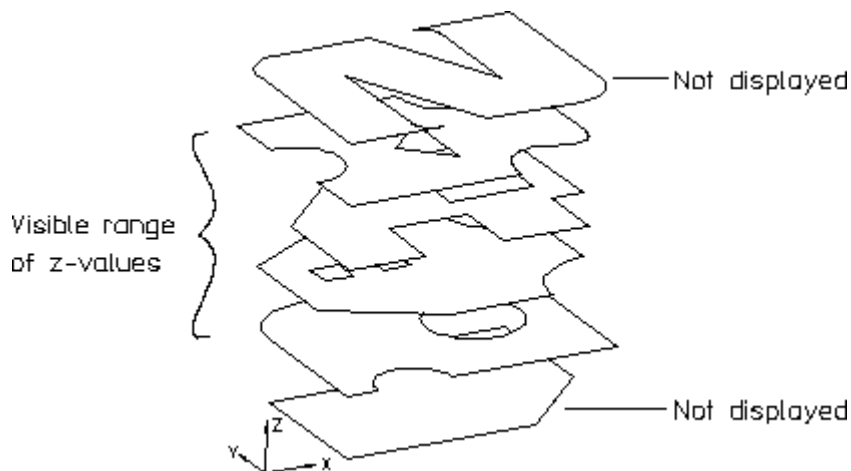
Resetting the Viewport after Viewing

Having used the **Show Range** dialog box to display selected elements/geometry, you will need to reset the viewport so that you can see all elements/geometry again. To reset the viewport:

1. Click **Show** in **Hidden**.

2. Click **Reset Show** in **Show**.

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

You are recommended to click Backup in Change to Hidden to store your assembly (in `workfile.hid`) before generating hidden lines as this command smashes the parts structure and creates the hidden line geometry.

To generate hidden lines:

1. Click **Change to Hidden** in **Hidden**.

2. Click **Backup** or **No Backup** in **Change to Hidden**.

Loading the Backup Drawing

If you wish to recover the backup drawing (`workfile.hid`) produced by clicking **Backup** in **Change to Hidden**:

1. Store your hidden line view.

2. Click **Change to Hidden** in **Hidden**.

3. Click **Load Backup** in **Change to Hidden**.

   Creo Elements/Direct Drafting will load the original drawing (`workfile.hid`) produced 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 469.

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 207. CRANK_ ASSEMBLY on page 400.

Figure 207. CRANK_ASSEMBLY
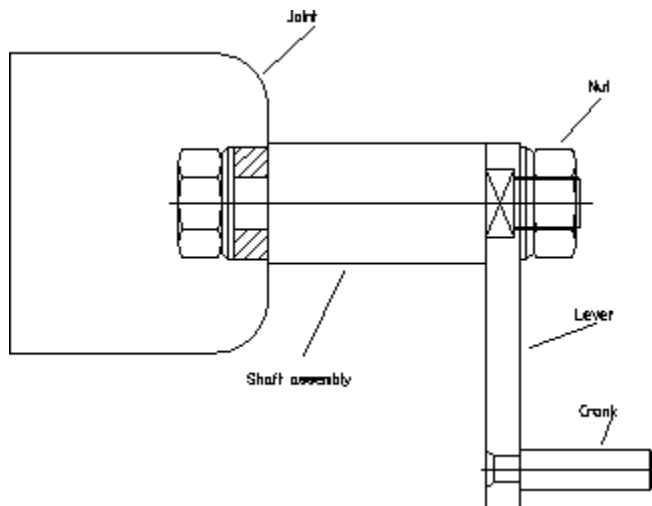
This crank assembly consists of the top part named CRANK_ASSEMBLY and five subparts. The parts tree is indicated in the following list.

```
CRANK_ASSEMBLY
      Nut
      Lever
      Crank
      Joint
      Shaft assembly
```

Draw a Cutaway Hole in the Joint

1. Click **Parts ▸ Edit ▸ Part**.

2. Click the `Joint` to make it the active part.

3. Click **Interpolation Spline** in **Geometry**.

4. Switch on the **Closed** check box option in **Interpolation Spline**.

5. Click points for the spline to create the cutaway hole shown in .

   Figure 208. CRANK_ASSEMBLY With Cutaway Hole
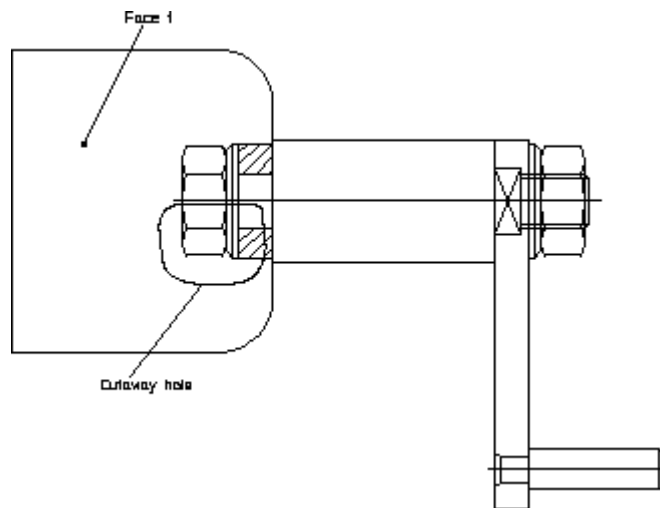
6. Click **End**.

Create Face 1

1. Click **Hidden Mode** and **On** in **Hidden**.

2. Click the current viewport to get visual feedback while creating faces (the face is highlighted).

3. Click **Generate Element** in **Hidden**.

4. Click **Element** in **Generate Element**.

5. Box the Joint.

6. When prompted for the z-value, enter **1**

   Creo Elements/Direct Drafting assigns a z-value of 1 to all elements in the box.

7. Click **End**.

Generate Face 1

1. Click **Generate Face** in **Hidden**.

2. Click **Single Automatic** in **Generate Face**.

3. Click a point on Face 1.

4. When prompted for the z-value, enter **2**

5. Click **End**.

6. Pull down the **View** menu and click **Redraw**.

Generate Faces in the Nut

1. Click **Parts ▶ Edit ▶ Part**.

2. Click the `Nut` to make it the active part.

3. Click **Generate Element** in **Hidden**.

4. Click **Element** in **Generate Element**.

5. Box the Nut.

6. When prompted for the z-value, enter **1**

   Creo Elements/Direct Drafting assigns a z-value of 1 to all elements in the box.

7. Click **End**.

8. Click **Generate Face** in **Hidden**.

9. Click **All in Part** in **Generate Face**.

10. Pull down the **View** menu and click **Redraw**.

Generate a Face in the Lever

1. Click **Parts ▶ Edit ▶ Part**.

2. Click the `Lever` to make it the active part.

3. Click **Generate Element** in **Hidden**.

4. Click **Element** in **Generate Element**.

5. Box the Lever.

6. When prompted for the z-value, enter **1**

   Creo Elements/Direct Drafting assigns a z-value of 1 to all elements in the box.

7. Click **End**.

8. Click **Generate Face** in **Hidden**.

9. Click **All in Part** in **Generate Face**.

10. Pull down the **View** menu and click **Redraw**.

At this point you would check for any errors you may have made when assigning z-values and generating faces. See Figure 209. CRANK_ASSEMBLY with Hidden Mode On on page 403.

Figure 209. CRANK_ASSEMBLY with Hidden Mode On
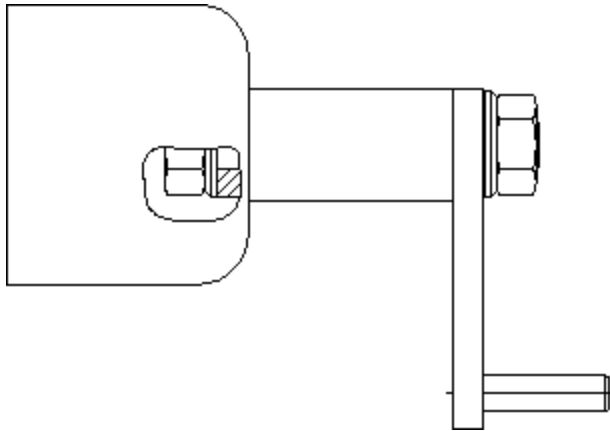
Generate Hidden Lines

1. Click **Change to Hidden** in **Hidden**.

2. Click **Backup** in **Change to Hidden**.

> ⚠ **Caution**
>
> We recommend that you use Backup in Change to Hidden to store your assembly (in `workfile.hid`) before generating hidden lines, as this command smashes the parts structure and creates the hidden line geometry.

Creo Elements/Direct Drafting would generate hidden lines as shown in Figure 210. Cutaway Hole and Hidden Lines on page 404, Figure 211. Nut — Showing Hidden Lines on page 404 and Figure 212. Lever — Showing Hidden Lines on page 404.

Figure 210. Cutaway Hole and Hidden Lines
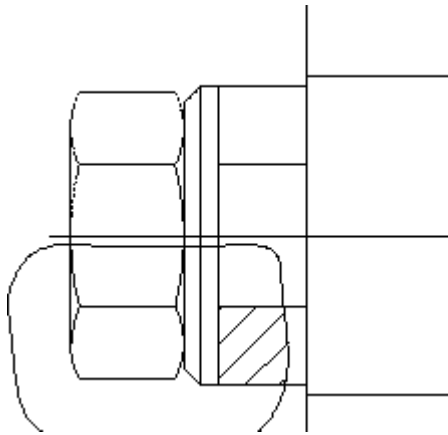
Figure 211. Nut — Showing Hidden Lines
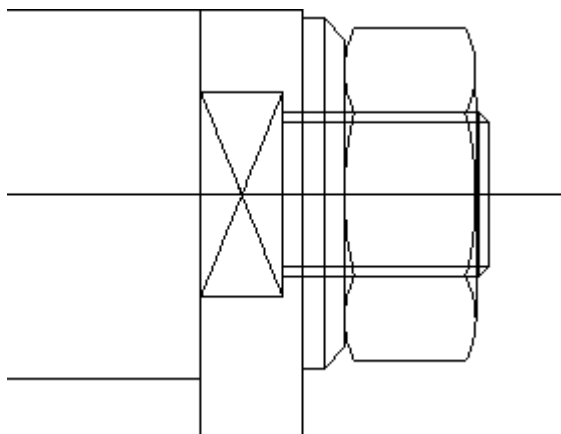
Figure 212. Lever — Showing Hidden Lines
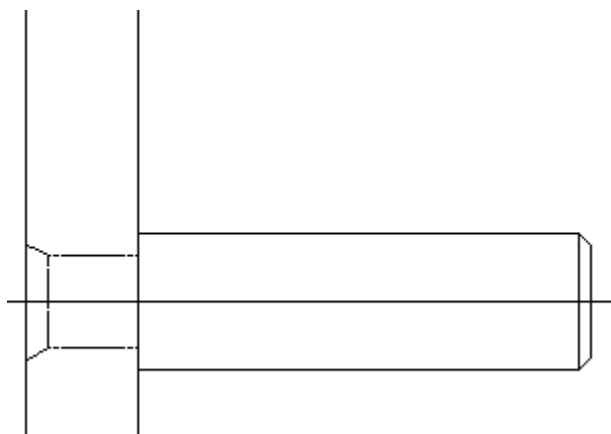
3. Click **Hidden Mode** and **On** or **Off** in **Hidden** to toggle the hidden lines on or off.

14

Using Layers and Infos

Drawing Using Layers

The layers commands in the **Info** menu allow you to split up your drawing into a number of layers. Thus you can place individual elements in a particular layer, or in a number of layers.

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.

For 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.

shows the parts structure.

Figure 213. 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 214. Layer 1 — Housing Elements
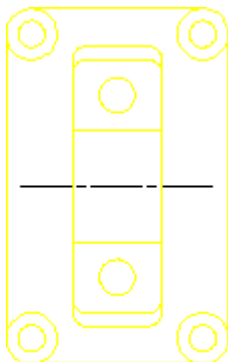
Layer 2 contains all the BEARING elements.

Figure 215. Layer 2 — Bearing Elements

Layer 3 contains all the SHAFT elements.

Figure 216. Layer 3 — Shaft Elements
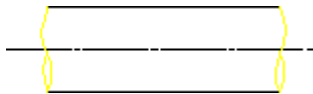
Layer 4 contains all the CAP elements.

Figure 217. Layer 4 — CAP Elements
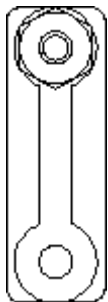
Layer 5 contains most of the HOUSING elements, all the BEARING elements and some of the SHAFT elements.

Figure 218. Layer 5 — Housing, Bearing and Shaft Elements
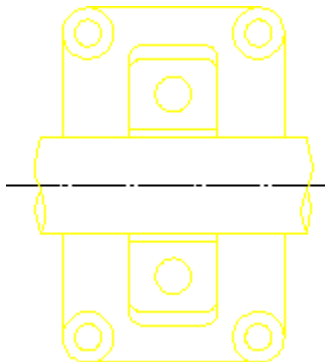
Layer 6 contains some of the HOUSING and SHAFT elements, plus all the CAP ones.

Figure 219. Layer 6 — Housing, Shaft and CAP Elements
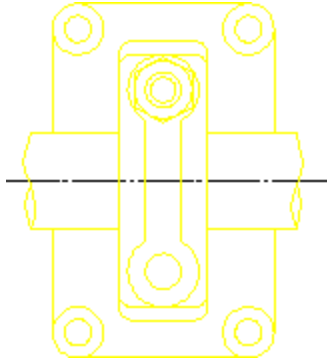
Adding a New Layer

To add a new layer to your drawing:

1. Click **Add Layer** in **Info**.

2. Specify a unique number for this new layer.

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. In the status toolbar, use the layer field pull-down menu and select the active layer.

2. Click the desired number and you are ready to start drawing in the layer.
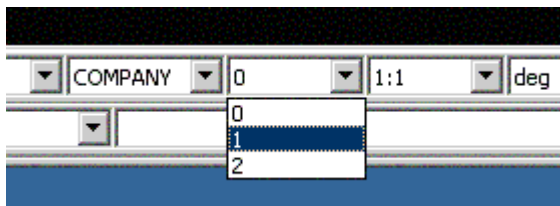
Duplicating Elements in Other Layers

This section shows 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 **Add Layer** in **Info**.

2. Creo Elements/Direct Drafting will prompt for the number of the layer to which elements are to be added.

3. Select the elements to be added. This can be done in one of two ways:

   • Click or box the elements to be added. This method is most suitable for simple selections of one or two elements.

   • Alternatively, click **Select** and **Advanced** to use the various **Select** options. When your selection is complete, click **Confirm** and **OK**. Creo Elements/Direct Drafting will add 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 shows 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 **Change Layer** in **Info**.

2. Enter the number of the layer to which elements are to be transferred.

3. Identify elements to transfer. This can be done in one of two ways:

   a. Click or box the elements to be transferred. This method is most suitable for simple selections of one or two elements.

   b. Alternatively, click **Select** and **Advanced** to use the various **Select** options. When your selection is complete, click **Confirm** and **OK**. Creo Elements/Direct Drafting will transfer the elements to the specified layer. This method is most suitable for large selections as you can see which elements have been identified so far.

Using the Layer Dialog in Show

Click on the **Layer** tab in **Show** to display the **Layer** dialog box.

In this dialog box, 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 **Show** in **Setup**.
2. Click the **Layer** tab.
3. In the **Layer** dialog box, select the layer(s) you want to switch on or off.
4. Click **On** or **Off**.
5. Enter the number of the layer.

   Creo Elements/Direct Drafting will now switch the specified layer on or off.

Showing Layer Color

You can color a layer to aid visualization. This will not affect plotter pen numbering.

To color a layer:

1. In the **Layer** dialog box in **Show**, select a layer number.
2. Select a color from the **Color** selection list.
3. Click **OK** to apply the layer color change.

   Creo Elements/Direct Drafting will redraw the elements in the specified layer in the new color.

To restore the original color of the layer, switch the layer on again in **Layer**.

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** menu.

To modify or delete complete layers:

1. Click **Delete** or click a **Modify** option;.
2. Click **Layer** in **Select**.
3. When prompted, specify the layer you want to delete or modify.
4. Click **Confirm** and **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 during the Layers commands. Thus if you place an element in layer 1, Creo Elements/Direct Drafting will attach the info `LAYER: 1` to it.

The **Info** menu contains the following info commands:

- **Add**
- **Edit**
- **Change**
- **List**
- **Delete**

Adding Infos

This section shows how to add infos to drawing elements. The **Add** options are:

- **Current Element**
- **Select Element**
- **Screen**

To a Selected Element

To add infos to existing drawing elements:

1. Click **Add** in **Info**.

2. Click **Select Element** in **Add**.

3. Enter the info text.

   This can be done in one of two ways:

   • Enter the required text enclosed in quotes (' ' or " ") into the command line. This is the best method to use for infos of up to one line in length.

   • Alternatively, click **Screen** to display a text box (based on a standard text editor) in which you can enter the info text. Don't enclose 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. Identify the required elements and the system will add the info to them. Alternatively, you can use the **Select** menu in the normal way to identify elements.

Adding the Current Info

You can create infos as current such that they are attached to all subsequently drawn elements:

1. Click **Add** in **Info**.

2. Click **Current Element** in **Add**.

3. Enter the info text.

   This can be done in one of two ways:

   • Enter the required text enclosed in quotes (' ' or " ") into the command line. This is the best method to use for infos of up to one line in length.

   • Alternatively, click **Screen** to display a text box (based on a standard text editor) in which you can enter the info text. Don't enclose 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.

Editing an Info

This section shows how to edit any existing info.

The command options are:

- **Current Element**
- **Select Element**

An Existing Element

To edit the info for any existing element:

1. Click **Edit** in **Infos**.

2. Click **Select Element** in **Edit**.

3. Select the drawing element to be edited.

   The display will clear and be replaced by the infos belonging to the element.

4. Edit the text using the edit keys and keyboard.

5. When you have finished editing, close the text box to return to the normal Creo Elements/Direct Drafting display.

The Current Info

To edit the current info:

1. Click **Edit** in **Infos**.

2. Click **Current Element** in **Edit**.

   The normal display will clear and be replaced by the current infos.

3. Edit the text using the edit keys and keyboard.

4. When you have finished editing, close the text box to return to the normal Creo Elements/Direct Drafting display. All newly drawn elements will contain the revised current infos.

Deleting an Info

This section shows how to delete existing infos or the current info.

The command options are:

- **Current Element**
- **Select Element**

An Existing Info

To delete the infos belonging to any existing drawing elements:

1. Click **Delete** in **Infos**.

2. Click **Select Element** in **Delete**.

3. Select elements. This can be done in one of two ways:

    a. Click individual elements or use a box to select several. The system will now delete the infos.

    b. Alternatively, use the **Select** menu options. When selection is complete, click **Confirm** and **OK** to delete the infos.

The Current Info

Click **Current Element** in **Delete** and Creo Elements/Direct Drafting will delete the current info. If you now continue drawing, all new elements will be without infos.

Changing Infos

This section shows how to change infos.

The command options are:

* **Current Element**

* **Select Element**

* **Global**

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 421.

An Existing Info

To change any existing info:

1. Click **Change** in **Infos**.

2. Click **Select Element** in **Change**.

3. Enter the line of info text to be changed.

   For example, "DRAWN ON 28.02.93".

4. Enter the replacement text.

   For example, "DRAWN ON 03.03.93".

5. Select the elements to be changed. This can be done in one of two ways:

    * Click individual elements, or box several elements.

Creo Elements/Direct Drafting changes the info text for the identified elements.

- Alternatively, use the **Select** menu options. When selection is complete, click **Confirm** and **OK** to change the elements.

The Current Info

To change the current info:

1. Click **Change** in **Infos**.
2. Click **Current Element** in **Change**.
3. Enter the line of the current info to be changed.
4. Enter the replacement line and the system will change the current info.

   Once the change is made, you are ready to continue drawing. The revised current info will be added to all new drawing elements.

All Occurrences

To perform changes to all occurrences of an info:

1. Click **Change** in **Infos**.
2. Click **Global** in **Change**.
3. Enter the line of info text to be changed.
4. Enter the replacement line and the system will change every such occurrence without you having to identify elements.

Listing Infos

This section shows how to produce a list of all infos used in your drawing.

1. Click **List** in **Infos**. All infos used in your drawing are displayed in a text box.
2. Close this text box 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 **Namespaces** in **Infos**. All namespaces used in your drawing (for example, SYS:HL_SCOPE includes Z-LEVELS) are displayed in an Info Browser.

2. Close the Info Browser to return to the normal Creo Elements/Direct Drafting display.

Using the Info Dialog in Show

Click on the **Info** tab in **Show** to display the **Info** dialog box.

In this dialog box, you can:

- Switch infos on a 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 **Show** in **Setup**.

2. Click the **Info** tab.

3. In the **Info** dialog box, select the info you want to switch on or off.

4. Click **On** or **Off**.

   All elements containing this info text will be switched on (displayed) or off (hidden).

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 via the **URL** dialog box accessible from the **Infos** menu.

Adding URLs

To add a URL to the current Creo Elements/Direct Drafting drawing or to a part or to an element in the drawing:

1. Click **URL** in **Infos**.

2. In the **URL** dialog box, click **Add**.

3. In the **Add** dialog box, select the area to which you want to add a URL:

Current	to add a URL to the current part
Element	to add a URL to an element or a group of elements.
Part	to add a URL to a part other than the current part

4. Specify the URL in the **Info** field of the **Add** dialog box.

5. Click **OK**.

6. Unless you selected **Current** (the default area) in the **Add** dialog box, select the part or element to which you want to attach the URL.

7. Click **End**.

Changing URLs

To change the URL attached to a part or element:

1. Click **URL** in **Infos**.

2. In the **URL** dialog box, click **Change**.

3. In the **Change** dialog box, select the area in which you want to change a URL:

Current	to change a URL of the current part
Element	to change a URL of an element or a group of elements.
Part	to change a URL of a part other than the current part

4. Unless you selected **Current** (the default area) in the **Change** dialog box, select the part or element whose URL you want to change.

5. Select the URL to be replaced from the old selection list.

6. Specify the URL you want to use instead.

7. Click **OK**.

8. Unless you selected **Current** (the default area) in the **Change** dialog box, select the part or element whose URL you want to change.

Deleting URLs

To remove a URL attachment from a part or element:

1. Click **URL** in **Infos**.

2. In the **URL** dialog box, click **Delete**.

3. In the **Delete** dialog box, select the area from which you want to delete a URL attachment:

Current	to remove a URL attachment from the current part
Element	to remove a URL attachment from an element or a group of elements.
Part	to remove a URL attachment from a part other than the current part

4. Select the URL attachment to be removed from the **Info** selection list.

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:

1. Click **URL** in **Infos**.

2. Display the **List** selection list at the top of the **URL** dialog box.

3. Select the area whose URLs you want to list:

Global	to list all URLs attached to any items in the current drawing
Current	to list all URLs attached to the current part
Element	to list all URLs attached to an element or group of elements.
Part	to list all URLs attached to a part other than the current part

4. Unless you selected **Global** or **Current**, click **Select** in **URL** and select the part or element(s) whose URLs you want to list.

   The associated URL list is displayed in the **URL** dialog box.

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 **URL** in **Infos**.

2. List the URLs attached to your drawing. See Listing URLs on page 418 for instructions.

3. Select a URL from the list.

4. Click **Show** to highlight the element or part to which this URL is attached.

Accessing a URL

To access a URL attached to a part or element in a drawing:

1. Click **URL** in **Infos**.

2. List the URLs attached to your drawing. See for instructions.

3. Select the URL you want to access.

4. Click **Browse** to access this URL by means of your default Web browser or navigator.

Using the URL Dialog in Show

Click on the **URL** tab in **Show** to display the **URL** dialog box.

In this dialog box, 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 **Show** in **Setup**.

2. Click the **URL** tab.

3. In the **URL** dialog box, select the URL you want to switch on or off.

4. Click **On** or **Off**.

   Creo Elements/Direct Drafting will now switch the specified URL on or off.

Assigning a Color to a URL

You can color a URL to aid visualization.

To color a URL:

1. In the **URL** dialog box in **Show**, select a URL.

2. Select a color from the **Color** selection list.

   Creo Elements/Direct Drafting will indicate the URL in the newly selected color.

Further Information

For further technical information about layers and infos, refer to the Creo Elements/Direct Drafting help system:

Enter `help layer` or `help info` or `help url` at the Creo Elements/Direct Drafting command line. The screen will clear and Creo Elements/Direct Drafting will display further information about using layers or infos or URLs with Creo Elements/Direct Drafting.

15

Storing, Loading and Filing

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. shows file system hierarchy.

Figure 220. File System Hierarchy
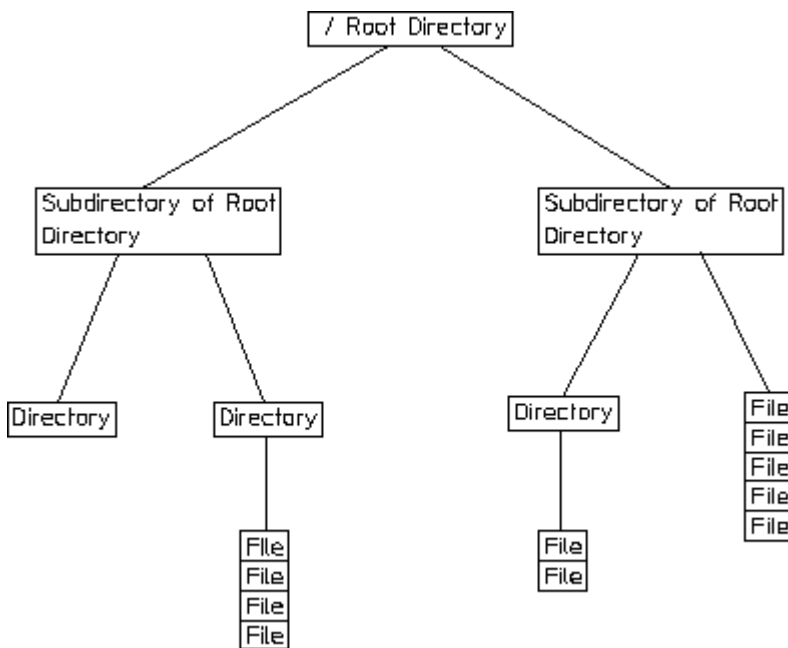
Filenames

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.

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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 shows how to identify 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 later in this chapter).
- By using the Navigator.

Using the File Path Name

The file path consists of the volume/drive identifier, a directory path and a filename, all separated by a backslash ("\") or a colon (":"). Suppose you wish to perform a file command on the following:

Filename:	nvb_drw_101.mi
In directory:	larry
In volume/drive:	D:

To identify the file you would enter the following string:
```
'D:\larry\nvb_drw_101.mi'
```

> **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'
> ```

Backslashes are generally used in this manual because this most closely matches the MS-DOS file system convention.

Using the Search Command

The SEARCH command allows you to specify further directories, other than the current directory, to be searched during the LOAD and INPUT commands. Use this command to quickly load macros, standard parts and so on contained in other directories.

For example, suppose that the volume and directory `D:\users\eng_constr` is to be searched for the file `geom_1.mi`. Then if the following had been entered into the search command:
```
SEARCH
    'd:\users\eng_constr'
END
```

all you would have to enter is:
```
LOAD 'geom_1.mi'
```

Opening and Saving Drawings by Means of the 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 enter the filename on the command 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 columns of files and directories.
	Displays files and directories in report mode. In this mode, you can sort the listed files and directories by type, size, last modification date/time and attributes.

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 via the Creo Elements/Direct Drafting File Browser

When you use **Open**, the drawing file you select or specify will replace 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 means of the Creo Elements/Direct Drafting File Browser:

1. In Creo Elements/Direct Drafting, pull down the **File** menu. Note that the last four drawing files you used are listed by default. If the file you want to open is included in this list, simply select the associated filename.

2. Click **Open** to display the File Browser.

3. Unless the file you want to open is listed in the file pane of the File Browser, pull down the **Directory** selection list and change to the directory where the file is stored. If this directory is specified in the file pane, double-click on the directory name to list the files contained in it.

4. 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.

5. Click on the filename. If the file you want has a format different from the most recent opened, pull down the **Filter** selection list and select the applicable format:

    • **Model Interface Standard (*.mi)**

    • **Compressed MI (*.bi)** (compressed by means of the Z-lib mechanism)

    • **Bundle (*.bdl)** (Note that 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)**

    • **All files (*.*)**

    Refer to Handling External File Formats on page 439 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.

    Optionally, tick the following checkboxes:

Check 2D	Creo Elements/Direct Drafting will check this file for invalid elements (such as extremely short lines) and will remove such elements.
Check 2D Retain Dependency	Creo Elements/Direct Drafting will re-allocate 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.

6. Click **Open** to open this file in Creo Elements/Direct Drafting. Alternatively, enter the filename and press **[Enter]**.

7. 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 428 for instructions.

8. 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 means of the Creo Elements/Direct Drafting Navigator:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Open** to display the File Browser. Alternatively, click the following button in the Creo Elements/Direct Drafting toolbar:

2. Click **Navigator** to display 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.

3. In the left pane of the Navigator, switch to the directory whose drawing files you want to view.

4. In the right pane of the Navigator, scroll down until you see the preview image of the drawing file you want to open. Note that the Navigator displays a preview image for each file provided in the formats listed above.

5. Double-click on the preview image to open its associated drawing file in Creo Elements/Direct Drafting.

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 428 for instructions.

7. Once the drawing is displayed in Creo Elements/Direct Drafting, you can edit it as required.

Note

To return to the File Browser, simply close the Navigator.

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:

1. Pull down the Creo Elements/Direct Drafting **File** menu.

2. Click on the name of the drawing file you want to load.

Saving a Drawing File via the Creo Elements/Direct Drafting File Browser

To save a drawing by means of the Creo Elements/Direct Drafting File Browser:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Save** to display the File Browser. Alternatively, click the following button in the Creo Elements/Direct Drafting toolbar:

2. Unless the filename you want to use is listed in the file pane of the File Browser, pull down the **Directory** selection list and change to the directory

where you want to store the drawing file. If this directory is specified in the file pane, double-click on the directory name to list the files contained in it.

3. Click on an existing filename (to overwrite an existing file) or enter a new filename (to create a new drawing file).

4. 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.

5. If you want to save the drawing in a format different from the most recent saved, pull down the **Filter** selection list and select 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)

   • **AutoCAD [version] DXF (*.dxf)** (version 2013, 2010, 2007, and so on)

   • **Initial Graphics Exchange Format (*.igs)**

   • **AutoCAD [version] Drawing (*.dwg)**

   • **Virtual Reality Modeling Language (*.wrl)**

   • **Drawing Web Format (*.dwf)**

   • **Scalable Vector Graphics (*.svg)**

   • **Text file (*.txt)**

   • **ME10 [version] - MI [version] (*.mi)**

   • **AutoCAD [version] Drawing (*.dwg)** (version 2013, 2010, 2007, and so on)

   Refer to Handling External File Formats on page 439 for further information on external file format translators used by Creo Elements/Direct Drafting.

6. Click **Add fonts** if you have want to ensure that the fonts you used with this drawing can be seen by others. This adds to the size of the file, but will ensure that others who do not have the same fonts on their systems can read your drawing text. Available for MI format only.

7. 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, pull down the **File** menu.

2. Click **Save As**.

3. Unless the filename you want to use is listed in the file pane of the File Browser, pull down the **Directory** selection list and change to the directory where you want to store the drawing file.

4. Enter the name of the file in which you want to save the current drawing.

5. If you want to save the drawing in a format other than the most recent saved, pull down the **Filter** selection list and select the applicable 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 428 for a list of supported formats.

6. 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 an HPGL format:

1. Click **File**.

2. Click **Print**.

3. Click the drop-down menu in the **Name** field to see options:

   • HPGL_GENERIC - Hewelett-Packard plotter language

   • HPGL_GENERIC - improved version with text defined

   • HPGL2_RTL_GENERIC - improved version with text and images

4. Select a generic option or click a boxed plus sign to select a model number suboption.

5. Note that the **Print to file** box is automatically selected.

6. Click **OK** in the **Print** dialog box.

7. The **Print to File** save box appears. Browse to select an output location.

8. Click **SAVE** in the **Print to File** save box.

Note that 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 470.

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**.

2. Click **Print**.

3. Click the drop-down menu in the **Name** field.

4. Note that the **Print to file** box is automatically checked.

5. Click **OK** in the **Print** dialog box.

6. The **Print to File** save box appears. Browse to select an output location.

7. Click **SAVE** in the **Print to File** save box.

For further information on setting up a print configuration, refer to Creating a Print Configuration on page 470.

Saving Selected Elements

To save a selection of elements in your drawing:

1. Enter the following command into the user input line:
   STORE

2. Select the element(s) you want to store.

3. Specify the name of the file (in apostrophes) in which you want to store this selection.

Saving a Part

To save the active (current) part in the current Creo Elements/Direct Drafting drawing:

1. In Creo Elements/Direct Drafting, pull down the **File** menu.

2. Click **Store Part** to display the **Store Part** dialog box.

3. Unless the filename you want to use is listed in the file pane of the File Browser, pull down the **Directory** selection list and change to the directory where you want to store the drawing file.

4. Enter the name of the file in which you want to save the current part.

5. If you want to save the part in a format other than MI (the default), pull down the **Filter** selection list and select the applicable format. Refer to Saving a Drawing File via the Creo Elements/Direct Drafting File Browser on page 428 for a list of supported formats.

6. Click **Save** to save the part in the specified drawing file.

Loading a Part

When you use **Load File**, the drawing file you select or specify will be 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. In Creo Elements/Direct Drafting, pull down the **File** menu.

2. Click **Load File**.

3. Unless the file you want to add as a part is listed in the file pane of the **Load File** dialog box, pull down the **Directory** selection list and change to the directory where the file is stored. If this directory is specified in the file pane, double-click on the directory name to list the files contained in it.

4. Switch on the **Part** check box option in the **Load File** dialog box.

5. Click or specify the filename.

6. If a file is provided in a format other than MI (the default), pull down the **Filter** selection list and select the applicable format. Refer to Opening a Drawing File via the Creo Elements/Direct Drafting File Browser on page 425 for a list of supported formats.

   Optionally, you can tick any or all of the following checkboxes:

As Subpart	Loads a drawing as a subpart of the active part.
Absolute	Loads the drawing back into 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 will check this file for invalid elements (such as extremely short lines) and will remove such elements.
Check 2D Retain Dependency	Creo Elements/Direct Drafting will re-allocate dependent elements (such as dimensions or hatches) of removed invalid elements to plausible valid elements.

7. 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.

Loading a Subpart

You can load a part file as a subpart under the active part.

To load a subpart:

1. In Creo Elements/Direct Drafting, pull down the **File** menu.
2. Click **Load File**.
3. Unless the file you want to add as a part is listed in the file pane of the **Load File** dialog box, pull down the **Directory** selection list and change to the directory where the file is stored. If this directory is specified in the file pane, double-click on the directory name to list the files contained in it.
4. Switch on the **As Subpart** check box option in the **Load File** dialog box.
5. Click or specify the filename.
6. If a file is provided in a format other than MI (the default), pull down the **Filter** selection list and select the applicable format. Refer to Opening a Drawing File via the Creo Elements/Direct Drafting File Browser on page 425 for a list of supported formats.
7. Click **Open** to add the file as a part 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.

Creating a New Drawing

To create a new drawing:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **New ▸ Drawing**. Alternatively, click the following button in the Creo Elements/Direct Drafting toolbar:

2. If Creo Elements/Direct Drafting already contains a current drawing, you will be 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 429.

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:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **New ▸ Window**. Alternatively, click the following button in the Creo Elements/Direct Drafting toolbar:

> **Note**
> - Each instance will run in a separate window.
> - All Creo Elements/Direct Drafting instances are independent of each other; there is no main or secondary instance.
> - 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.
> - Newly created Creo Elements/Direct Drafting instances are initiated without the `input_customize` command. Consequently, any additional applications associated with a given Creo Elements/Direct Drafting instance are available in that instance only. For example, if you run a licensed third-party standard parts library in a Creo Elements/Direct Drafting instance, this library is not automatically available in another Creo Elements/Direct Drafting instance.

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. In Creo Elements/Direct Drafting, pull down the **File** menu and click **New ▸ Dwg in New Window**.

2. In the file browser, switch to the appropriate directory.

3. Select the drawing file you want to load.

4. Click **Open** to load this drawing file in a new Creo Elements/Direct Drafting instance.

Importing Macros, Fonts, Pictures, and HPGL Files

You can import the following files into your current Creo Elements/Direct Drafting drawing:

- Creo Elements/Direct Drafting Macros
- Fonts
- Pictures
- HPGL Files

Importing a Creo Elements/Direct Drafting Macro

To import a Creo Elements/Direct Drafting macro:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Import**.

2. Click **Macro**.

3. Click one of the following options:

Input	to import a macro provided in ASCII format.
Load	to import a macro provided in binary format.

4. Select the macro from the dialog box.

5. Click **Open** to open the macro.

Importing a Font

To import a font:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Import**.
2. Click **Font**.
3. Click one of the following options:

Convert From TT/T1	to import a TrueType font provided in ASCII format. Select a font from the dialog box and click **Convert** to convert and load the font for use in Creo Elements/Direct Drafting.
Load	to import a font provided in binary format. Select a font from the dialog box and click **Open** to load the font for use in Creo Elements/Direct Drafting.

Specifying the Resolution of an Imported TrueType Font

To specify the resolution of a TrueType font you want to convert to a Creo Elements/Direct Drafting font:

1. Enter the following command into the user input line:
   CONVERT_TO_ME10_FONT
2. Enter the desired resolution. Note that the maximum value is 255.
3. Enter the font name.

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 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. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Import**.
2. Click **Picture**.
3. Select a bitmap file from the **Import Raster Image** dialog box.
4. Click **Open** to open the bitmap file in Creo Elements/Direct Drafting.
5. Give the picture a name.
6. In Creo Elements/Direct Drafting, specify the position for the imported bitmap file.

Refer to Editing Imported Pictures on page 445 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 an HP-GL (Hewlett-Packard Graphics Language) or HP-GL2 file:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Import**.
2. Click **Load HPGL**.
3. Specify the name (complete with path) of the HP-GL file you want to import.
4. In Creo Elements/Direct Drafting, specify the position for the imported HP-GL file.

Exporting Macros, Fonts, and Pictures

You can export the following from your current Creo Elements/Direct Drafting session:

- Creo Elements/Direct Drafting Macros
- Fonts
- Pictures

Exporting a Creo Elements/Direct Drafting Macro

To export a Creo Elements/Direct Drafting macro:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Export**.
2. Click **Macro**.
3. Click one of the following options:

Save	to export and save a macro in ASCII format.
Store	to export and save a macro in binary format.

4. Select a macro from the **Select Macro** dialog box. Optionally, click **All** to select all macros.
5. Click **OK** to display the **Save Macro** dialog box.
6. Select or specify a macro file name (*.mac).
7. Click **Save** to save the macro file.

Exporting a Font

When you have edited a font (or any character(s) included in a given font) by means of the Font Editor (see Font Editor on page 305), you need to export this font for further use.

To export a font:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Export**.
2. Click **Font**.
3. Click one of the following options:

Save	to export and save a font in ASCII format.
Store	to export and save a font in binary format.

4. Select a font from the **Select Font** dialog box. Optionally, click **All** to select all fonts.
5. Click **OK** to display the **Save Font** dialog box.
6. Select or specify a font file name (*.fnt).
7. Click **Save** to save the font file.

Exporting a Picture

You can output your drawings in high-resolution pixel format:

1. Click **File**
2. Click **Export**
3. Select a picture for export.

Prompts are:

`Enter PORT, BOX, DEL_OLD or "destination"` - to select input

- `Enter point, CURRENT or viewport number` - to further select after **PORT**
- `Enter lower left and upper right corner of box which is to be dumped` - to select after **BOX**

`Enter RESOLUTION or "destination"`

- `Enter PAPER_TO_PIXELS or output image width, height` - if you select `RESOLUTION`

  ○ `PAPER_TO_PIXELS` - recalculates paper format and dpi to your X,Y resolution. You will be prompted for a paper format (A-E, A4-A0) or size, and then you will be prompted for dpi.

- ○ `Enter output image width, height` - enter width and height in pixels. Enter -1 for either width or height to retain the ratio of the current viewport and specify the other dimension in pixels.

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.

Handling External File Formats

Creo Elements/Direct Drafting provides the following translators for external file formats:

- DWF Translator

- DXF Translator

- IGES Translator

- VRML Translator

To run one of these translators, double-click on the associated icon in the Creo Elements/Direct Drafting program group. Each of these translators is controlled via a DOS window.

Exiting Creo Elements/Direct Drafting

To exit Creo Elements/Direct Drafting:

1. In Creo Elements/Direct Drafting, pull down the **File** menu and click **Close**.

2. If there were any changes to your current drawing, you are prompted to save these changes.

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Revision Checking

This chapter tells you how to use the commands 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 221. Original Drawing and Revised Drawing on page 442 shows the original drawing (A) and the revised drawing (B). Figure 222. Both Files Loaded on page 442 shows A and B superimposed so that you can easily identify the modified elements.

Figure 221. Original Drawing and Revised Drawing
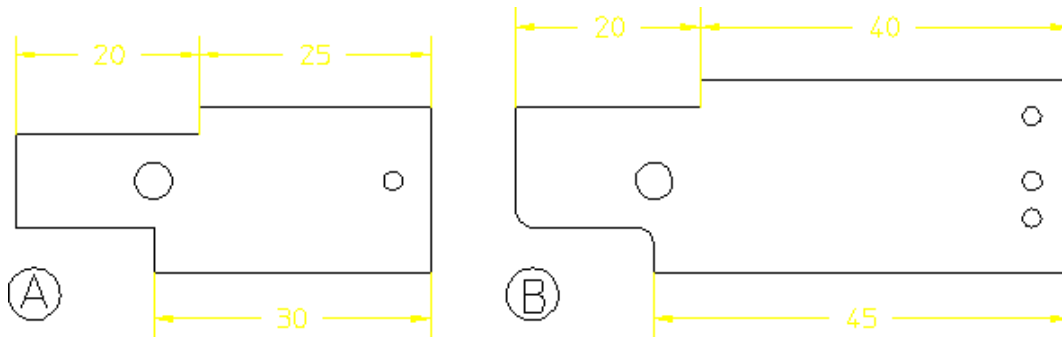
Figure 222. Both Files Loaded
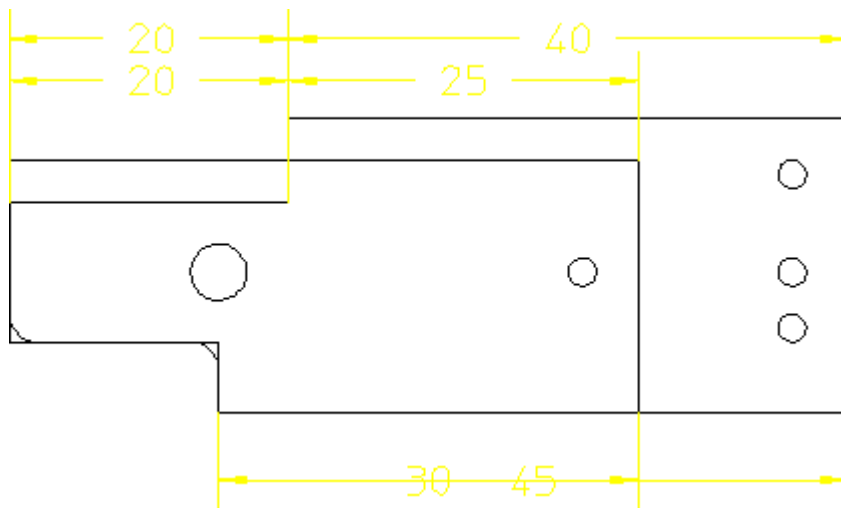
Using Revision Checking

To use the revision checking facility, click **Rev Check**.

1. Click **Load Files** in **Rev Check**.

2. In the **Load** dialog box, specify the name of the original drawing file (for example: dt_cs_rev06) and the name of the revised drawing file (for example: dt_cs_rev10).

3. Enter `dt_cs_rev10'

    Creo Elements/Direct Drafting displays the two drawings overlapping.

4. Click **Geometry** in **Compare**.

    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

The other options in the **Compare** menu do the following:

Annotation	In addition to **Geometry**, this option also compares the properties of the geometry and texts — linetype, color, text attributes, info texts — and the dimensions and hatches in both drawings.
Parts	In addition to **Annotation**, this option compares the parts structure in both drawings.
	If you modify a drawing solely by a scaling factor, clicking **Parts** displays all elements in red (indicating a difference). However, the options **Geometry** and **Annotation** displays 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 (**Parts ▸ Scale**).

Viewing Single Drawings

When both drawings are loaded, you can toggle between them using **View** in **Rev Check**. To display the original drawing only, click **Original** in **View**. To display the revised drawing only, click **Revised** in **View**. To display both drawings, click **Both** in **View**.

Continuing with the Drawing

When you are ready to continue with your drawing, click **Original** or **Revised** in **Continue**. 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 **View** command, the **Continue** command does 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 an option in the **Compare** menu can be reset to their original colors.

Reset Show	Use this command to show the drawings in their original colors.
Show Difference	Use this command to show the drawing elements in different colors, corresponding to the result of the last comparison.

Changing the Color Scheme

To change the color scheme:

1. Click **Set Color** in **Rev Check**.
2. Select the color for changes in the original drawing.
3. Select the color for changes in the revised drawing.
4. Select the color for identical elements.

Refer to Defining Custom Colors on page 59 for instructions on defining custom colors.

The color changes take effect with the next redraw.

17

Editing Imported Pictures

Show - Image Options

Show commands allow you to control the tradeoff between image quality and processing speed for redraw.

Show/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 the SHOW PICTURES function to show or hide pictures.

Tip: When zooming with the dynamic mouse ([Ctrl] + right mouse button), hide the images with the **Box** option.

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 **Picture**.

2. Click **Anti-alias Factor** in **Show**.

   - `Specify picture quality number.` Enter a number between -2 and +2.

   - `Select pictures for which to apply the change.`

Positive input between **0** and **2** tells 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 436, you can edit it as an image in Creo Elements/Direct Drafting.

Moving an Image

To move an image, click **Modify ▶ Move**, and select a method.

Rotating an Image

To rotate an image, click **Modify ▶ Rotate**, and select a method.

Scaling an Image by X and Y Factors

To scale an image, click **Modify ▶ Scale**, and select a method.

Arranging Images

To arrange images:

1. Click **Change Level** in **Picture**.

2. Choose among these options:

   • **To Front** to move an image in front of all other objects.

   • **Forward** to move one level forward.

   • **In Front of** [selection] to move in front of a reference object.

   • **To Back** to move an image in back of all other objects.

   • **Backward** to move one level backward.

   • **Behind** [selection] to 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**
>
> The command CHANGE_LEVEL changes the level of a single part or subpart as well as a picture.

Restriction: 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 **External Edit** in **Picture**.
2. Click the picture to be edited. The external editor will open a new window.
3. 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 **Update** in **Picture**.
2. Click the picture to be updated.
3. Enter:

    • the file name of the new picture, or
    • an empty string (two single quote marks - ") to use the remembered, or default, file.

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 223. Cleaning an Imported Image
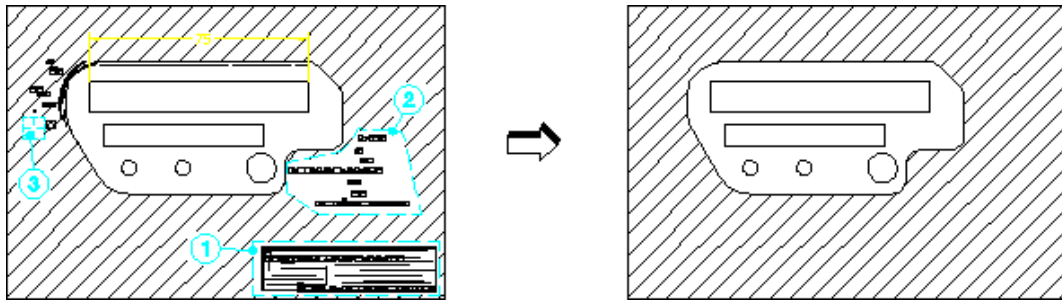
You can clean up or edit an image with the following **Change Color** options:

Item	Description
1	**Box** option
2	**Poly** option
3	**Brush** option

To change the color of pixels:

1. Click **Picture** ▸ **Change Color** ▸ **Change**.

2. Select the image.

3. Select one of the following options:

   a. **Box** - and enclose the required pixels in a box

   b. **Poly** - and enclose the required pixels in a polygon

   c. **Brush** - press and hold down the **[Shift]** key while dragging the cursor over the pixels to be colored (the cursor changes shape)

4. Click **End** to end the command.

Additional Information:

- If the **Brush** color is different from the background color, you can add or redline information on the image.

- The image is always described by a rectangular frame. If you rotate the image, the frame increases/decreases according to the horizontal and vertical lines passing through the corners; see . The swept area is only temporary — it disappears along with any pixel editing you may have done.

Figure 224. Rotating an Imported Image
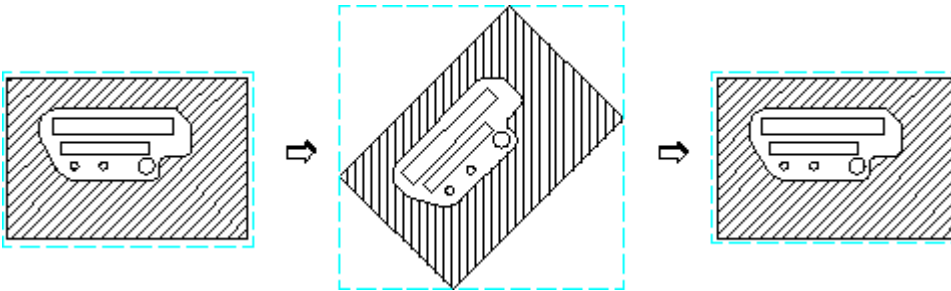
Modify

Relocate allows you to move a picture:

1. Click **Picture ▸ Modify ▸ ReLocate**.
2. Select an image to move.
3. Select a reference point.
4. Select a destination point.

BW Invert allows you to change between white-on-black and black-on-white displays:

1. Click **Picture**.
2. Click **Modify**.
3. Click **BW Invert** - prompts you to select a picture to invert.
4. Click **End**.

> **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 **Picture**.
2. Click **Transparency**.

- **On** - Prompts you to select a picture. If that picture has a transparency color, the color will become invisible during overlay.

- **Off** - Prompts you to select a picture. If that picture has a transparency color, the color will not become invisible (default setting).

- **Color** - To select a picture and apply the transparency color you have chosen from the color menu 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. will not be supported. All imported files will be opened as not transparent.

Using the Picture Browser

When you click **Picture Browser** in **Picture**, Creo Elements/Direct Drafting displays the Picture Browser. This browser displays information about the currently loaded images:

Name	Filename and path 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

Deleting an Image

To delete an image, use the Delete command in the main menu.

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, the options in **Picture ▸ Change Level** can be used to move the image forwards or backwards. Click in a non-overlapping area to select the image. Alternatively, enter the name on the user input line.
- If drawing geometry overlaps an image, it is placed at the front of the image (s).

💡 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.

18

Linking and Embedding OLE Objects

Files (such as spreadsheets, graphics or text pages) created in external applications can be included in Creo Elements/Direct Drafting by inserting associated OLE objects. Drawings containing OLE objects can be printed as PDF and PS files.

Please note the following:

• OLE functionality is only supported in the Windows user interface.

• An 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 will not be 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 into an Creo Elements/Direct Drafting drawing:

1. In Creo Elements/Direct Drafting, pull down the **Insert** menu and click **Object**.

2. The **Insert Object** dialog box lists all applications on your system that provide OLE capability.

3. Scroll down the list and select an application.

4. Select one of the following options by clicking the associated radio button:

   - **Create New**
   - **Create from File**

Create New

This option inserts a new OLE object of the selected application into your Creo Elements/Direct Drafting drawing.

To create a new OLE object:

1. Select an object type from the **Object Type** list and click **OK**.

2. If you use the **Display As Icon** option, the OLE object will be represented by the associated application icon in Creo Elements/Direct Drafting. Optionally, click **Change Icon** to select a different icon for the OLE object.

3. Retain the default OLE object name or specify a name for the OLE object in the User Input Line.

4. 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

This option inserts the contents of a file as an object into your document so that you can activate the object by means of the application in which it was created.

To create an OLE object from an existing file:

1. Click **Browse**. Select the file you want to link from the File Browser.

2. Click **Open** to open this file.

3. Optionally, switch on **Link** 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. If you use the **Display As Icon** option, the OLE object will be represented by the associated application icon in Creo Elements/Direct Drafting. Optionally, click **Change Icon** to select a different icon for the OLE object.

5. Click **OK**.

6. Retain the default OLE object name or specify a name for the OLE object in the User Input Line.

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.

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. Pull down the **Insert** menu.

2. Click **OLE**, **Change Level** and **To Front** or **To Back**, **Forward** or **Backward**, **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. OLE objects you created by means of the **Create from File** option will reflect changes in the underlying file.

To activate an OLE object:

1. Pull down the **Insert** menu.

2. Click **OLE** and **Activate**.

3. Select the OLE object to be activated.

Renaming an OLE Object

You can rename an existing OLE object in a drawing.

To rename an OLE object:

1. Pull down the **Insert** menu.

2. Click **OLE** and **Rename**.

3. Select the OLE object to be renamed.

4. Type the new name of OLE object in the User Input Line.

5. Press **Enter**.

Using the OLE Browser

To display the OLE Browser:

1. Pull down the **Insert** menu.

2. Click **OLE** and then **OLE Browser**.

The OLE Browser displays 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

The OLE objects can be activated, deleted, renamed, saved, converted to icons and hidden or shown using the OLE Browser.

19

Annotation

This chapter discusses the Creo Elements/Direct Drafting Annotation module.

Introduction

The **Annotation** module included 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 general Creo Elements/Direct Drafting Annotation procedure comprises the following steps:

- Prepare the Creo Elements/Direct Drafting drawing environment for Annotation.
- Load an MI file generated in Creo Elements/Direct Annotation.
- 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 that a warning message is displayed for any "structurally dangerous" operations that would impair 3D/2D associativity.
- After making changes to the underlying 3D model in Creo Elements/Direct Drafting, update the relevant view(s) to visualize and review the resulting changes in 2D.
- Edit the 2D views as appropriate.
- 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/2D associativity for further use in Creo Elements/Direct Annotation, you must observe the following rules when using the Creo Elements/Direct Drafting **Annotation** module:

- Do not copy one drawing into another drawing.
- Do not manually insert one or more sheets into another drawing.
- Do not manually insert views into a sheet (using the view definition commands of Creo Elements/Direct Annotation). Only use the **Sheet** commands provided by the Creo Elements/Direct Drafting **Annotation** module.
- Do not change or delete info texts beginning with "SD_" or "DOCU" or "AM_ ".

- Do not modify view parts or 3D parts below view parts with the commands in the **Modify** menu of Creo Elements/Direct Drafting. Only use the **View** commands provided by the Creo Elements/Direct Drafting **Annotation** module.

- Do not scale view parts or 3D parts below view parts with the command in the **Parts** menu of Creo Elements/Direct Drafting. Only use the **Scale** command provided by the Creo Elements/Direct Drafting **Annotation** module.

- Do not change the scale of the TOP part or the parts below the TOP part with the Creo Elements/Direct Drafting **Parts ▸ Scale** commands.

- Do not change the owner of a part or subassembly (which belong 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** command in Creo Elements/Direct Annotation to create detail views.

Preservation of Modifications

The following modifications in the Creo Elements/Direct Drafting **Annotation** module are preserved when an update is performed in Creo Elements/Direct Annotation:

- Attribute changes such as line type or line color, info text added to geometry elements derived from the 3D model.

- Centerlines created in Creo Elements/Direct Drafting.

- Dimensions, texts, symbols and additional 2D geometry created in the appropriate view parts. (Note that symbols, text and additional 2D geometry can be added to each part except the TOP part).

- Geometry 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 in the Creo Elements/Direct Drafting **Annotation** module 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 (and which are not owned by the TOP part) will not be 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 will not behave associatively in Creo Elements/Direct Annotation.

- Hatch holes automatically created in Creo Elements/Direct Drafting (forced by dimensions or texts when Creo Elements/Direct Drafting option TEXT_HOLE_INSERTION is set to ON by default) do not update associatively in Creo Elements/Direct Annotation.

- Hatch holes automatically created in Creo Elements/Direct Annotation for text do not update associatively in Creo Elements/Direct Drafting when the text is changed in Creo Elements/Direct Drafting.

- Hatches that were automatically created in Creo Elements/Direct Annotation but deleted in Creo Elements/Direct Drafting will re-appear 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 update in Creo Elements/Direct Annotation.

- If you split or merge 2D geometry in Creo Elements/Direct Drafting which was derived from the 3D geometry during an update in Creo Elements/Direct Annotation, the geometry will look like the original one after the next update in Creo Elements/Direct Annotation.

Preparing the Creo Elements/Direct Drafting Drawing Environment

Before you load an MI file generated in Creo Elements/Direct Annotation, be sure to complete the following steps:

1. To make sure that the top part is empty, enter the following into the user input line:
```
RESET_SYSTEM
CONFIRM
```

2. Activate the Creo Elements/Direct Drafting **Annotation** module by clicking **Modules ▸ Annotation**.

Loading an Annotation Drawing

Load an MI file generated in Creo Elements/Direct Annotation as follows:

1. Go to **File ▸ Open**.
2. Select the MI file you want to load 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. In **Annotation**, click **Drawing ▸ Sheet ▸ Sheet Browser**.
2. Drag the Sheet Browser to where you want to display it.

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. Click on the associated sheet number in the right-hand column of the Sheet Browser.

To select a view from the Sheet Browser:

1. In the Sheet Browser, select the sheet in which this view is included.
2. Click **Curr View**.
3. Click on the associated view number in the right-hand column of the Sheet Browser.

Views

The Creo Elements/Direct Drafting **Annotation** module provides a number of commands for selecting, changing and manipulating views.

Selecting the Current View

In addition to the method described in , you can proceed as follows to select the current view in the current sheet:

1. In **Annotation**, click **Drawing ▸ View ▸ Current View**.

2. Click on the view you want to make current.

Modifying Views

You can change the position of views on a sheet. Also, you can move views between sheets. Note that a warning message is displayed for any "structurally dangerous" operations that would impair 3D/2D associativity.

To change the position of one or multiple views:

1. In **Annotation**, click **Drawing ▸ View ▸ Modify View**.

2. Select the option for the view change you want to perform:

Move 2Pts	Moves the selected view(s) from the first point to the second point.
Move Horizontal	Moves the selected view(s) by the specified horizontal distance.
Move Vertical	Moves the selected view(s) by the specified vertical distance.
Move Sht to Sht	Moves the selected view(s) to another sheet.
Rotate	Rotates the selected view(s) by the specified angle about the specified center.
Scale	Scales the selected view(s).

Hiding Hidden Lines

To hide hidden lines in the Annotation MI file:

1. In **Annotation**, click **Drawing ▸ View ▸ Hide ▸ Hidden Line**.

2. Select the view(s) in which you want to hide hidden lines.

Hiding Tangent Lines

To hide tangent lines in the Annotation MI file:

1. In **Annotation**, click **Drawing ▸ View ▸ Hide ▸ Tangent Line**.

2. Select the view(s) in which you want to hide tangent lines.

Showing Hidden Lines

To show (previously hidden) hidden lines in the Annotation MI file:

1. In **Annotation**, click **Drawing ▸ View ▸ Show ▸ Hidden Line**.

2. Select the view(s) in which you want to show hidden lines.

Showing Tangent Lines

To show (previously hidden) tangent lines in the Annotation MI file:

1. In **Drawing ▸ View ▸ Show ▸ Tangent Line**.

2. Select the view(s) in which you want to show tangent lines.

Showing Differences in a View Following a 3D Change

Any view you see in the Creo Elements/Direct Drafting **Annotation** module is a 2D derivative of a 3D model view in Creo Elements/Direct Drafting. When the underlying 3D model changes in Creo Elements/Direct Drafting, you can update the associated view in Creo Elements/Direct Drafting **Annotation** and check for differences in the view before and after this update.

Showing Differences

In **Annotation**, click **Drawing ▸ View ▸ Show Difference**.

*   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 the update of the underlying 3D model, click **Drawing ▸ View ▸ Reset Show**. Note that **Reset** returns the view to the state prior to **Show Difference**.

Handling Annotation Elements

Optionally, you can change or accept annotation elements.

Changing an Annotation

You can change an annotation, e.g. by selecting a different reference element.

To change an annotation:

1. In **Annotation**, click **Drawing ▸ View ▸ Annotation ▸ Change**.
2. Click on the annotation you want to change.
3. Follow the prompts displayed in the prompt line.

Accepting All Annotation Elements

To accept all (current or changed) annotation elements, click **Drawing ▸ View ▸ Annotation ▸ Accept**.

Dimensions

Moving a Dimension

To move a dimension:

1. In **Annotation**, click **Drawing ▸ View ▸ Dimension ▸ Move**.
2. Click on the dimension you want to change.
3. Drag the dimension text to its new position.

Returning a Moved Dimension to Its Original Position

To move a dimension back to its original position:

1. In **Drawing ▸ View ▸ Dimension ▸ Reset**.
2. Click on the dimension you want to restore.

Sheets

You can create, select, change, and modify sheets.

Creating a Sheet

To create a new sheet:

1. In **Annotation**, click **Drawing ▸ Sheet ▸ Create Sheet**.

2. Enter a number for the new sheet. Check the Sheet Browser for existing sheet numbers.

3. In the **Load Frame** browser, select a frame for the new sheet.

You can now add new views to this sheet, e.g. by moving from an existing sheet or by loading a view from Creo Elements/Direct Drafting.

Selecting the Current Sheet

To make a sheet current:

1. In **Annotation**, click **Drawing ▸ Sheet ▸ Current Sheet**.

2. Click on the sheet you want to make current.

Alternatively, select the current sheet from the Sheet Browser (see).

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 that a warning message is displayed for any "structurally dangerous" operations that would impair 3D/2D associativity.

To modify a sheet:

1. In **Annotation**, click **Drawing ▸ Sheet ▸ Modify Sheet**.

2. Select the option for the change you want to perform:

Frame Type	to select a different frame from the **Load Frame** browser, thus replacing the existing frame by the newly selected one.
Frame Position	to move the frame to the desired position.
Delete	to delete the sheet.

Hiding, Deleting, Unhiding

The Creo Elements/Direct Drafting **Annotation** module 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 that a warning message is displayed for any "structurally dangerous" operations that would impair 3D/2D associativity.

To hide or delete elements:

1. In **Annotation**, click **Hide Geometry**.
2. Select the option for the operation you want to perform:

No Delete	Hides element but does not delete them. Elements hidden in this way can be restored through an Unhide operation.
Prev Delete	Deletes any elements you have previously marked as **No Delete**. Elements deleted in this way cannot be restored through an Unhide operation.
Final Delete	Deletes (removes) elements. Elements deleted in this way **cannot** be restored through an Unhide operation.

3. Click **Hide** (the menu field above these three options).
4. Click or box the elements you want to hide or delete.

Unhiding Elements

Any elements that you have hidden by means of the **No Delete** Option can be unhidden.

Unhiding Selected Elements

Elements that have been hidden (but not deleted) can be selectively unhidden.

1. In **Annotation**, click **Unhide Geometry ▸ Selected**. Any elements previously hidden by means of the **No Delete** option are now shown in a special highlight color for selection.
2. Click or frame the element(s) you want to unhide.

Sequentially Unhiding Elements

Optionally, you can sequentially unhide elements that have been hidden (but not deleted).

1. In **Annotation**, click **Unhide Geometry ▸ Sequential**. Any hidden elements are shown in a special highlight color.

2. Select the direction for your sequential Unhide operation:

Backward	Undoes your **Hide** operations in reverse order; i.e. the last **Hide** operation is undone first. Optionally, enter the number of backward **Unhide Geometry** steps.
Forward	Undoes your **Hide** operations in the order in which they were performed; i.e. the first **Hide** operation is undone first. Optionally, enter the number of forward **Unhide Geometry** steps.
Next	Undoes the next **Hide** operation in the current direction (**Backward** or **Forward**).

3. Any unhidden elements are now shown in a special highlight color.

Changing Colors, Linetypes and Hatches

In the **Annotation** module, you can change the color and linetype of hidden lines and tangent lines. Also, you can change the color, angle, and distance of hatches.

1. In **Annotation**, click **Drawing**, **View**, **Change**.

2. Select the change you want to perform:

Hidden Line	to change the color and/or linetype of hidden lines.
Tangent Line	to change the color and/or linetype of tangent lines.
Hatch	to change the color and/or angle and/or distance of hatches.

3. To change these settings, follow the general instructions provided in Changing the Drawing Environment and Making Measurements on page 51.

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 **Show as Annotation** in **Annotation**. This mode reflects your **Hide** operations.

Showing the Drawing in Creo Elements/Direct Drafting Mode

To show the drawing in Creo Elements/Direct Drafting mode, click **Show as Annotation**Creo Elements/Direct Drafting in **Annotation**. This mode ignores any **Hide** operations but reflects 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. Go to **File ▸ Save**.

2. Enter the name of the file in which you want to store the current Annotation MI drawing.

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Printing and Plotting

This chapter shows you how to print and plot drawings and how to produce screen dumps.

Using the Print Manager

In a Windows environment, 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" into another application.

> 🗐 **Note**
>
> Depending on the output device(s) on your system, the Print Manager may allow you to set several printer/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/plotter to work with Windows.

Creating a Print Configuration

To facilitate the creation of consistent print/plot output, Creo Elements/Direct Drafting enables you to create any number of print configurations.

To create a print configuration:

1. Pull down the **File** menu and click **Print**. Alternatively, click the following button in the Creo Elements/Direct Drafting toolbar:

   🖶

2. In the **Print** menu, select and/or specify the settings you want to save in the new print configuration. These settings are discussed in the following sections of this chapter.

3. In the **Print configuration** section of the **Print** menu, click **Settings**.

4. At the top of the **Print Configurations** dialog box, specify a name for the print configuration you want to create.

5. Click **Add** to add this new print configuration name to the list of print configurations.

6. If you want to save the print configuration for future use, select it and click **Store**.

7. In the **Store Print Configurations** dialog box, select or specify a name (such as `my_print.m`) for your print configuration file. Note that this filename does not have to be the same as the print configuration name. However, be sure to specify a name that is readily identified.

8. Click **Save** to save this file.

Example of a Print Configuration

The following is an example of a Creo Elements/Direct Drafting print configuration.

```
{ 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

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

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.

Re-Using a Saved Print Configuration

When you quit Creo Elements/Direct Drafting, the system automatically saves your current print settings. This will be the setting 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 then these files appear in the **Print** menu as a **Print configuration**. Configurations that you create with the Print dialog are automatically stored in this subdirectory when you store your configuration.

To re-use binary legacy print configuration files (with file name `<print configuration>.cfg`) from a CoCreate Drafting version < 13.00:

1. Click **File ▸ Print** from the main menu. The **Print** dialog box opens.

2. Under **Print Configuration**, click **Settings.** The **Print Configurations** dialog box opens.

3. In the **Print Configurations** dialog box, click **Load**. The **Load Print Configurations** dialog box opens.

4. In the **Load Print Configurations** dialog box, select the print configuration file.

5. Click **Open**. The print configuration appears in the **Name** list.

6. In the **Print Configurations** list, selects the print configuration.

7. Click **Close** to close the **Print Configurations** dialog box.

   The **Print** menu settings change to the settings contained in the newly loaded print configuration.

Defining the Print/Plot Settings

Before printing or plotting a drawing, you should define exactly what will be printed/plotted and how the drawing will be printed/plotted.

Setting the Plot/Print Destination

Creo Elements/Direct Drafting provides a wide range of printer/plotter output options. After you select an option, you can send plots and screendumps either to the Windows Print Manager or to the Clipboard. You usually 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 into another Windows application.

> **Note**
>
> Creo Elements/Direct Drafting automatically derives the extension for the output file according to the selected printer/plotter.

Selecting a Printer or Plotter

To set a printer or plotter:

1. Click **File ▸ Print** from the main menu.
2. Click the tab labeled **Printer**.
3. Pull down the **Name** selection list in the **Printer** section of the **Print** menu.
4. Select the printer or plotter from this list.
5. Note the boxed plus sign beside some generic options. Click the plus sign to see and select model number suboptions.
6. Click **Properties** to 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/plotter to work with Windows.

Stop on Error

Check this box if you want a warning message when drawings do not fit on the selected paper size. The system will check:

- whether the size of the drawing will fit on the selected paper format
- when printing Annotation drawings, whether the largest one matches the chosen plot configuration. If not, you will see the warning, "Plot doesn't fit on paper" and you can stop the process.

Routing the Print/Plot Output to a File

If you want to send the print/plot output to a file, click on the **Print to file** check box.

Be sure to select the printer or plotter in whose format you want to create this 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.

When you click **OK** to start the print/plot job, you are prompted for a filename.

Setting the Paper Size and Number of Copies

Select the applicable paper size from the **Format** selection list in the **Paper** section of the **Print** menu.

If you have a multi-sheet drawing, click **Use sheet format** to automatically set the format and orientation of each sheet based on the sheet size.

To set the number of copies, either click the up or down arrow in the spin box to change the number, or enter the number in the **Copies** field.

Setting Orientation

Select:

- Portrait
- 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** from the main menu.
2. Click the **Drawing** tab.
3. Select a **Position**.

- **Center**: the default option. 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/plotting. Creo Elements/Direct Drafting will place the bounding box in the lower left corner of the paper.

For both **Center** and **Left**, you can specify an offset in the X direction and in the Y direction. The print/plot output will be moved accordingly in the X direction and/or Y direction.

Note that 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

If you want to rotate the drawing output to the printer/plotter,

1. Click **File ▸ Print** from the main menu.

2. Click the **Drawing** tab.

3. Type the angle of rotation in the **Rotate** field.

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/Plotting

1. Click **File ▸ Print** from the main menu.

2. Click the **Drawing** tab.

The **Print** menu **Range** options are:

- **All**: to print/plot the entire drawing - irrespective of the current viewport contents. This is the default option.

  - **Sheet**. When printing an Annotation drawing, you can select **Sheet** to print each Annotation sheet on a separate page. See Stop on Error on page 473 for information on a possible warning.

- **Viewport**: to print/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/plotted.

- **Box**: to print/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/plotted.

For example, suppose you are currently viewing a part, and the other parts/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 not displayed, will be 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 reach this setting

1. Click **File ▸ Print** from the main menu.
2. Click the **Drawing** tab.

Entering a Scale Factor

This is the default option. The entire print/plot output will be scaled, but dimension values are not affected.

Figure 225. Operation of Print/Plot Scale on page 476 shows the operation of print/plot scale.

Figure 225. Operation of Print/Plot Scale
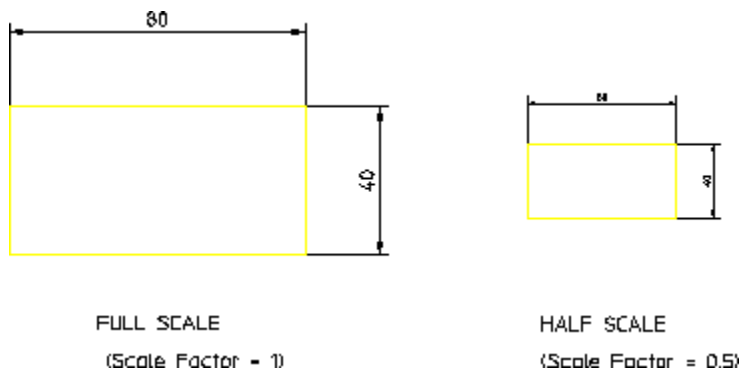
To set the print/plot scale factor:

1. Click the **Factor** radio button in the **Scale** section of the **Print** menu.
2. Enter the required scaling factor.

If you set the plotting scale such that the drawing is too large for the paper, Creo Elements/Direct Drafting will warn you at the start of a plot by displaying the maximum scale possible. Pressing ESC will abort the plot and allow you to reset the plot scale. Otherwise, press the space bar and the plot will continue, cutting off elements outside the paper format.

Fitting a Drawing

Click the **Fit** radio button in the **Scale** section of the **Print** menu. The print/plot output of your drawing will be scaled as large as possible within the limits of the paper size you specified.

Setting the Content of the Print/Plot Output

Creo Elements/Direct Drafting enables you to specify what will be included in the print/plot output. To reach this setting:

1. Click **File ▸ Print** from the main menu.
2. Click the **Drawing** tab.

- **Drawing**: all entities will be included in the print/plot output expect Construction lines. Spotlights for the current part are omitted. The drawing will appear as a normal drawing irrespective of the screen status (that is, the plot is unaffected by the **Show** functions).

- **Current Show**: to print/plot the drawing as shown in the viewport.

   1. Select a viewport in which you want to set up your own drawing conventions.
   2. Use the **Show** functions to set up the desired attributes.
   3. Click **Current Show** in the **Content** section of the **Print** menu to print/plot what is currently shown in the drawing.

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 will appear. The plotting area is unaffected.

Colors

This section of the **Print** menu enables you to set the **True Color** option, a linetype, and a **Plot Transformation** map for your printer/plotter environment. To reach this setting:

1. Click **File ▸ Print** from the main menu.
2. Click the **Drawing** tab.
3. In the **Colors** box, click **Pens**. The **Plot Transformations** dialog appears.

True Color

From the Plot Transformations pull-down menu, click **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

---

Setting the Linetype Length

To reach this setting:

1. Click **File ▸ Print** from the main menu.
2. Click the **Drawing** tab.
3. In the **Colors** box, click **Pattern Length**. The **Plot Linetype Length** dialog appears.

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. Via the **Plot Linetype Length** map, you can specify the length of each pattern and thus alter the fineness of the line. Figure 226. Different Linetype Lengths on page 479 shows two DASH_CENTER lines drawn with different linetype lengths. The linetype length is shown arrowed.

Figure 226. Different Linetype Lengths
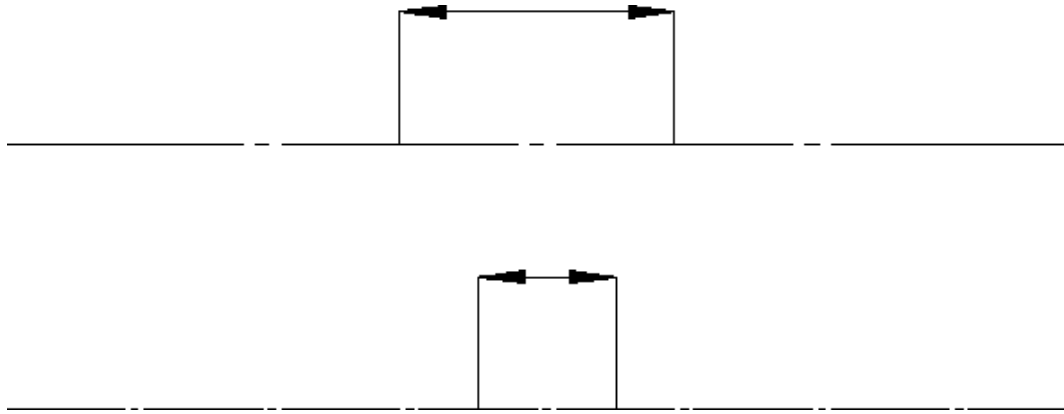
The seven broken linetypes that you can set are:

- DASHED
- LONG_DASHED
- DOT_CENTER
- DASH_CENTER
- PHANTOM
- CENTER_DASH_DASH
- DOTTED

To set the linetype length and minimum space for individual broken linetypes:

1. Click **Pattern Length** at the bottom of the **Print** menu to display the **Plot Linetype Length** map.

2. In the **Length** field, specify a segment length for each broken linetype.

3. In the **Min. Dist.** field, specify the minimum distance (the gap) between the broken line segments.

Alternatively, if you want to set all broken linetypes to the same segment length and the same minimum distance, click **All Linetypes**. Then specify a single length and a single minimum distance.

Plot Transformation

To reach this setting:

1. Click **File ▸ Print** from the main menu.

2. Click the **Drawing** tab.

3. In the **Colors** box, click **Pens**. The **Plot Transformations** dialog appears.

4. From the **Plot Transformations** pull-down menu, click **Plot Transformations**.

- The **Edit** button displays a separate **Pen Format** dialog in which you can set up a custom plot transformation map.
- The **Reset** button restores the default settings that you see the first time you display the **Plot Transformation** dialog box.
- The **Default Color** button applies a default color-to-pen assignment map.
- The **Default B/W** button applies a monochrome color-to-pen assignment map.
- The **Default GDI** button applies a 125-color plot pen color to pen assignment map.

Proceed as follows to set up the **Plot Transformation** map:

1. In the **Plot Transformation** map, click the row you want to change. This row will be highlighted.

2. Click **Edit** to display the **Pen Format** dialog box.

3. In the **Pen Format** dialog box:

   - Select a "from" linetype and then 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 **All Linetypes**.

   - Select a "from" color and then a "to" color to which you want to map the "from" color. For example, if you want all cyan lines in the drawing to be plotted in a custom pen color, select **Cyan** from the **Color** selection list and then select **Custom** from the **To** selection list to define that custom color.

     If you want to map all colors in the drawing to the associated colors on the plotter, click **All Colors**.

   - Select a "from" pensize and then 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, enter **0.3** in the **Pen Size** field and **0.45** in the **To** field.

     If you want to map all pensizes in the drawing to the associated pensizes on the plotter, click **All Pen Sizes**.

Assigning a Specific Pen to a Given Linetype/Color Combination

In the **Plot Transformation** map, proceed as follows to assign a specific plotter pen to a given linetype/color combination:

1. Select the color from the **Color** selection list.

2. Select the linetype from the **Linetype** selection list.

3. In the **Plotted As** section, select the linetype to be plotted. If you want to plot the same linetype that is used in the drawing, select **Same Linetype**.

4. Use the spin box to select the associated pen number.

5. Specify a pen width.

6. Click **OK** to add the changed row to the **Plot Transformation** map.

For example, let us 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. Select the dashed linetype and the color blue in the **Drawing** section of the **Plot Transformation** map.

2. In the **Plotted As** section, click **Same Linetype**.

3. Specify a pen width of 0.3.

4. In the **Pen** spin box, set the pen number to 5.

> 📝 **Note**
>
> The difference between line width (pen size) and line size is explained in Line Width vs. Line Size on page 56.

Repeat this procedure until the **Plot Transformation** map contains the required map of colors, linetypes, and associated plotter pens.

To restore the **Plot Transformation** default settings, click **Reset**. To restore the default color settings, click **Default Color**.

Setting the Picture Color

When printing or plotting a drawing that includes embedded pictures, you can set the output of the image to be either **Black & White** or **Color**.

To reach this setting,

1. Click **File ▸ Print** from the main menu.

2. Click the tab labeled **Pixmap**.

Retaining or Inverting Black and White Pictures

The **Reverse** radio button in the **B&W Pixmap** section of the **Print** menu enables you to reverse black and white pictures in the print/plot output.

To reach this setting,

1. Click **File ▸ Print** from the main menu.

2. Click the tab labeled **Pixmap**.

Picture Quality

To reach this setting,

1. Click **File ▸ Print** from the main menu.

2. Click the tab labeled **Pixmap**.

Select:

- **Best**: high quality
- **Low**: draft quality

Previewing a Print/Plot

Open the **Plot Preview** system viewport to see print/plot output on your screen. You can open the viewport after you select all the necessary print settings (as a last step before printing), or open it to watch while working on your drawing.

1. Click **File**.

2. Click **Print**.

3. Click **Show Preview**.

4. Select a color setting for the **Plot Preview** viewport only.

   - **Original**: to preview color printer/plotter output.
   - **Grayscale**: to preview black and white printer/plotter with dithering capability output.
   - **BW**: to preview black and white printer/plotter output.

5. The **Plot Preview** viewport appears or is updated.

The **Plot Preview** viewport remains open until you uncheck it in the **Print** dialog box.

Starting a Print or Plot Job

When you have selected or specified all of the required print/plot output settings, click **OK** at the bottom of the **Print** menu 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 into 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(s) to be printed or plotted.

2. Press the right mouse button on this selection.

3. Click **Print**.

Dumping the Creo Elements/Direct Drafting Screen

To copy the entire Creo Elements/Direct Drafting screen to the Clipboard, press **[Alt]** & **[Print]**.

Then paste the screendump into a graphics application of your choice for further processing.

For example, in Microsoft® Paint, pull down the **Edit** menu and click **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 438 for instructions.

Further Information

For further technical information about plotting or dumping, refer to the Creo Elements/Direct Drafting help system:

Enter `help plot` or `help bitmap_to_file` at the Creo Elements/Direct Drafting command line.

The screen will clear and Creo Elements/Direct Drafting will display further information about plotting or dumping.

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Customizing the Creo Elements/ Direct Drafting User Interface

Creo Elements/Direct Drafting provides a rich set of tools and options for user interface customization. Any customization changes in Creo Elements/Direct Drafting are written to macros.

Displaying and Hiding Menus

To display or hide menus in the Creo Elements/Direct Drafting user interface:

1. Press the right mouse button on any menu item.

2. Check the applicable checkbox to display a menu or set of menus. Alternatively, uncheck any items you want to hide.

Selecting Menu Look and Feel

Menus can be displayed as icons, text, or both. To select a menu style:

1. Click **Setup ▸ Customize ▸ Customize Layout**. The **Customize** window appears.

2. From the **Customize** window, select your preferred view from the **Text Options** field.

Editing Toolbar Items

To change the name, type, action or other property of a menu item in the Creo Elements/Direct Drafting user interface:

1. Display the menu item you want to change.

2. Press the right mouse button on that menu item. This brings up the **Edit Toolbar Item** dialog box.

Caption Key	The string in this field is the label displayed for the current menu item.
Enabled	Tick this box to enable the menu item or untick this box to gray out the menu item.
Unique Name	The Creo Elements/Direct Drafting internal name for this menu item. Note that this name must be unique.
Item Type	This combo box enables you to pull down a selection list of standard Microsoft Windows item types. The item types in this list are context sensitive. For example, a popup button enables you to select a submenu (discussed below) which will be displayable via this popup button.
Submenu	Enables you to branch to a submenu.
Tooltip Key	A brief description of the current menu item.
On Action	A command or function or macro name (e.g.: change_color) to be triggered via this menu item. Note that any macro specified here must be created separately.

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On Right Click	Command, function or macro name to be triggered by pressing the right mouse key on this menu item.
Category	This string allocates the menu item to a category for selection in the **Toolbar Items** tab.
Domain	Name of the file where Creo Elements/Direct Drafting will look for your localized version (if any) of this menu item. Note that multiple toolbars may share the same domain file. Optionally, a separate domain file may be created for each toolbar.
Image	Brings up an icon editor. Refer to Creating Icons on page 491 for details.
List Keys	Click **New (Insert)** to add an entry to the list of items. Click **Delete** to remove an item from this list. Click the up or down arrow to scroll within the list of keys. Note that any action string you define here requires an associated action definition in a separate macro.

Note

Combo boxes, radio buttons and checkboxes require the creation of a separate macro. For more detailed background information on the user interface customization capabilities of Creo Elements/Direct Drafting, refer to the TOOLBAR_ITEM section in the Creo Elements/Direct Drafting Help. Destroy not only removes items from the Toolbar Items tab but actually deletes the associated files. Note that Destroy cannot be undone.

Creating Custom Menus

To create a custom menu:

1. Pull down the **Tools** menu, click **Customize** and **Customize Layout**. Alternatively, press the right mouse button on any existing menu item and click **Customize Layout**.

2. In the **Customize** dialog box, click **Add**.

3. In the **Add Toolbar** dialog box, specify the following:

Caption Key	The string in this field is the label displayed for the current menu item.
Locked	Locks the menu so it cannot be destroyed.
Unique Name	The Creo Elements/Direct Drafting internal name for

	this menu item. Note that this name must be unique.
On Right Click	Command, function or macro name to be triggered by pressing the right mouse key on this menu item.
Domain	Name of the message file (if any) where Creo Elements/Direct Drafting will look for your localized version (if any) of this menu item. Each toolbar and toolbar item in the Creo Elements/Direct Drafting user interface is associated with a specific "domain" file. This domain file contains all of the strings associated with your toolbar or toolbar items. These strings include: • Caption • Category • Tooltip • List Items In the domain file, you can edit (e.g. localize) these strings.

4. Click **OK**.

5. Drag and drop the menu to any location in the Creo Elements/Direct Drafting user interface

Example: Creating a Background Menu

This example illustrates the procedure for creating a cascaded menu that enables you to change the background color of the current viewport in Creo Elements/ Direct Drafting. The newly created **Background** menu will contain a submenu with two buttons:

• Black

• White

The first task is to create two empty toolbars. The first of these will receive a popup button that displays the second toolbar. The second toolbar will then receive two "action" buttons that change the background color.

Stage 1: Defining Two Empty Toolbars

To define the first (initially empty) toolbar:

1. Click **Tools ▸ Customize ▸ Customize Layout ▸ Add**.

2. Specify the following in the **Add Toolbar** dialog box:

Caption Key	Background
Unique Name	tlb_backg
On Right Click	Retain the default.
Domain	cust_menus

3. Click **OK** to return to the **Customize** dialog box.

To define the second (initially empty) toolbar:

1. Click **Tools ▸ Customize ▸ Customize Layout ▸ Add**.

2. Specify the following in the **Add Toolbar** dialog box:

Caption Key	Background_2
Unique Name	tlb_backg_2
On Right Click	Retain the default.
Domain	cust_menus

3. Click **OK** to return to the **Customize** dialog box.

Result of this exercise:

You have just created two empty toolbars. Both of these are named Background.

Stage 2: Creating a Cascaded Submenu

Next, you will make one of these toolbars a "cascaded submenu" of the other:

1. In **Customize**, click on the **Toolbar Items** tab.

2. Click **Add** to display **Add Toolbar Item**.

3. In **Add Toolbar Item**, select or specify the following:

Caption Key	Black or White
Item Type	**Popup Button**
Unique Name	pob_black_white
Submenu	tlb_backg_2

4. Retain the defaults for the remaining fields.

5. Click **OK**.

6. Drag the newly created pob_black_white item to the Background menu you created in Stage 1.

Result of this exercise:

You have just created a first menu label in the toolbar named Background. The black arrow pointing to the right suggests cascading.

You can now click on the `Black or White` label to display the (still empty) toolbar you just defined as its submenu.

Stage 3: Adding Two Action Buttons to the Cascaded Submenu

Next, you need to define the two menu items for this newly created submenu.

To create the **Black** menu button:

1. In **Toolbar Items**, click **Add**.

2. Select or specify the following in the **Add Toolbar Item** dialog box:

Caption Key	Black
Unique Name	btn_background_black
Item Type	Push Button
ToolTip Key	Changes the background color to black
On Action	change_viewport_color black end

3. Retain the defaults for the remaining fields.

4. Click **OK**.

To create the **White** menu button:

1. In **Toolbar Items**, click **Add**.

2. To define the **White** menu button, select or specify the following in the **Add Toolbar Item** dialog box:

Caption Key	White
Unique Name	btn_background_white
Item Type	Push Button
ToolTip Key	Changes the background color to white
On Action	change_viewport_color white end

3. Retain the defaults for the remaining fields.

4. Click **OK**.

Next, you need to drag these two buttons to the cascaded submenu:

1. Drag the **Black** pushbutton definition from **Toolbar Items** to the cascaded submenu.

2. Drag the **White** pushbutton definition from **Toolbar Items** to the cascaded submenu.

Stage 4: Test

To test your newly created cascaded menu:

1. Drag the **Background** menu to any position in the Creo Elements/Direct Drafting user interface.

2. Click on the **Black or White** menu label.

3. Click **White** to change the viewport background color to white.

4. Click **Black** to change the viewport background color to black.

Creating Icons

To create an icon:

1. In the **Edit Toolbar Item** dialog box, click **Edit** under **Image** to bring up the **Image Editor**.

2. Use the Creo Elements/Direct Drafting Image Editor drawing tools to draw the geometry of your icon. Select colors as appropriate.

3. When you have completed the geometry, pull down the Image Editor **File** menu and click **Save**.

4. Specify a filename for the new image.

5. Click **Save** to save the file.

Adding Icons to the Toolbar

To add an icon (as a button) to the Creo Elements/Direct Drafting toolbar:

1. Pull down the **Tools** menu, click **Customize** and **Customize Layout**.

2. In the **Toolbar Items** tab, select the item for which you want to add or change an icon.

3. Click **Edit** to bring up the Icon Editor.

4. Create the icon.

5. Click **OK** to return to the **Edit Toolbar** dialog box. Check whether or not the icon meets your requirements.

6. Click **Edit** to edit the icon or click **OK** to accept the icon.

7. The menu item now has the icon associated with it. Drag and drop the menu item (complete with its icon) to the menu where you want to use it.

Defining Key Commands

To define a custom key command:

1. Pull down the **Setup** menu and click **Keys**.

2. In the **Key Definitions** dialog box, click a key (such as **[1]**).

3. Specify any or all of the following commands:

Plain	Enter the command to be executed when the key is pressed.
Control	Enter the command to be executed when the key is pressed in combination with the CTRL key.
Alt	Enter the command to be executed when the key is pressed in combination with the ALT key.
Alt-Control	Enter the command to be executed when the key is pressed in combination with the ALT + CTRL keys.

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

If you leave this box unchecked, the command will simply be 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.

Displaying Menus and Dialog Boxes via Mouse Buttons

You can allocate any Creo Elements/Direct Drafting menu or dialog box to a mouse button so the menu or dialog box pops up when that mouse button is pressed. The allocation procedure uses the following syntax:

```
define_mouse_key <key number> <mode> "show_popup_menu '<action text>'"#M
```

where:

define_mouse_key	is the macro command that allocates a command to a mouse button.
key number	is either 1 (left mouse button), 2 (middle mouse button), or 3 (right mouse button).
mode	is one of the following: **Plain**: The command is to be executed when the mouse button is pressed. Note that you should not use plain combined with the left mouse button because this button is used for menu selection and drawing work in Creo Elements/Direct Drafting.

	Shift: The command is to be executed when the mouse button is pressed in combination with the [Shift] key.
	Alt: The command is to be executed when the mouse button is pressed in combination with the [Alt] key.
	Shift Alt: The command is to be executed when the mouse button is pressed in combination with the [Shift]+[Alt] keys.
show_popup_ menu	is the macro command that brings up the menu or dialog box specified through the command.
action text	is the name of a Creo Elements/Direct Drafting menu or dialog box. Note that this string is case sensitive.
#M	causes the command to be executed immediately when the mouse button is pressed. If you decide to leave out #m, the command will be written to the command line.

For example, to display the **Arc** menu by pressing the right mouse button combined with the [Shift] key, enter the following command string into the User Input Line:

```
define_mouse_key 3 shift "TOOLBAR 'tlb_Arc' SHOW_AS_POPUP END"#M
```

📝 Note

The Creo Elements/Direct Drafting menu names are case sensitive.

Displaying Dialog Boxes Directly

The Creo Elements/Direct Drafting dialog boxes can be displayed directly. Simply enter the associated WUI command string in the User Input Line. To find out these command strings, enter:

```
LIST_KEYWORDS SCREEN
```

into the User Input Line. The resulting Creo Elements/Direct Drafting keyword list is displayed in a separate text editor window. Scroll to the section of keywords beginning with WUI_ to see the keywords that will display dialog boxes.

For example, to display the **Construction** menu, enter:

```
WUI_CONSTRUCTION
```

Managing Layouts

Via **Tools**, **Customize**, Creo Elements/Direct Drafting provides the following layout management options:

Save Layout	Saves the entire layout to a file.
Save Toolbars	Saves one or multiple user-selected toolbars to a file.
Load Layout	Loads an entire layout.
Load Toolbars	Loads one or multiple user-selected toolbars.

Converting Existing Multi-Column Menus

The `mcm2lcassic.m` macro supplied with this release of Creo Elements/Direct Drafting enables you to convert multi-column menus created with earlier Creo Elements/Direct Drafting releases.

Resetting the Layout

To return your Creo Elements/Direct Drafting layout to its original (uncustomized) state, simply click **Tools ▸ Customize ▸ Reset Layout**.

If you have inadvertently removed the **Reset Layout** button or if you have lost the **Customize** submenu or the entire **Tools** menu, enter the following command in the user input line to restore the original Creo Elements/Direct Drafting layout:
`toolbar_layout reset confirm`

Windows User Interface Customization History

As of CoCreate Drafting rev. 11.0, all user interface customization is handled through the Creo Elements/Direct Drafting macro language. Any existing user interface customization files from previous releases of Creo Elements/Direct Drafting could be converted.

The following sections provide a brief overview of the Creo Elements/Direct Drafting GUI customization technological history.

CoCreate Drafting Rev. 8.7 or Earlier

All "classical" UI menus were handled through the Creo Elements/Direct Drafting macro language. These were loaded by means of the LOAD_MACRO command and activated by means of the INPUT command.

Figure 228. User Interface Customization in CoCreate Drafting Rev. 8.7 or Earlier
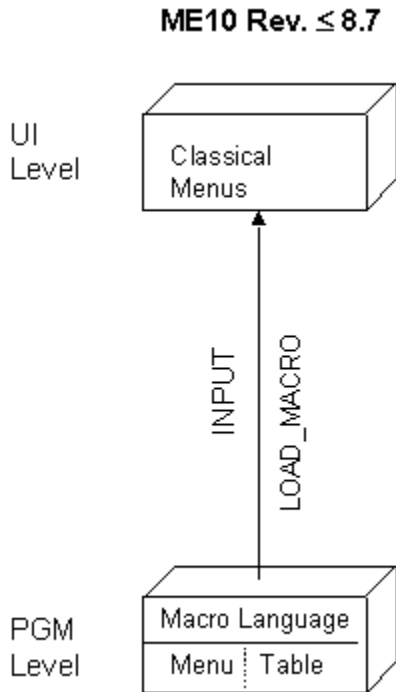
In the "classical" user interface:

- Fonts were defined via the MEFONT or -font command.

- The font color and background color were defined in macros via the MENU, TABLE_TITLE and/or TABLE_COLUMN command, etc.

- Menus could only be moved within the Creo Elements/Direct Drafting window.

CoCreate Drafting Rev. 9.0 and Rev. 10.0

In addition to the classical menus (see previous section), customized Windows style menus were created and managed as `.lt` format files by the Customization Toolkit.

Multi-column and multi-row menus created by the user were managed as ASCII format files. These were loaded by means of the LOAD_MULTICOL_MENU command.

Figure 229. User Interface Customization in CoCreate Drafting Rev. 9.0 and 10.0
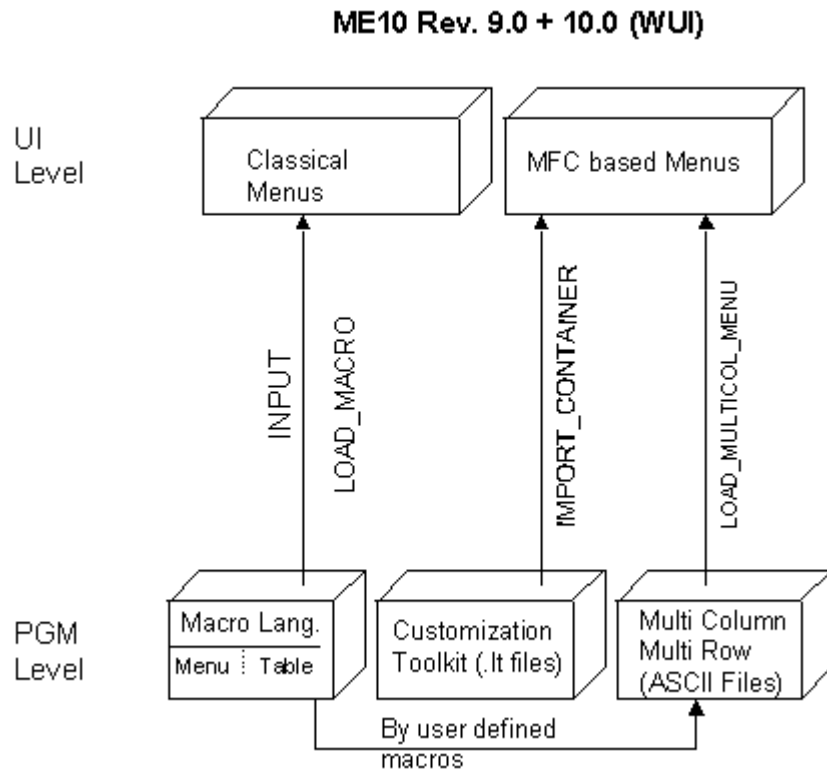
CoCreate Drafting Rev. 10.5 or Higher

The `mcm2lclassic.m` macro enabled users to convert existing multi-column menus for re-use in the current and future Creo Elements/Direct Drafting releases.

Any `*.lt` files were automatically converted by the TOOLBAR_LOAD command.

Figure 230. User Interface Customization in CoCreate Drafting Rev. 10.5 or Higher
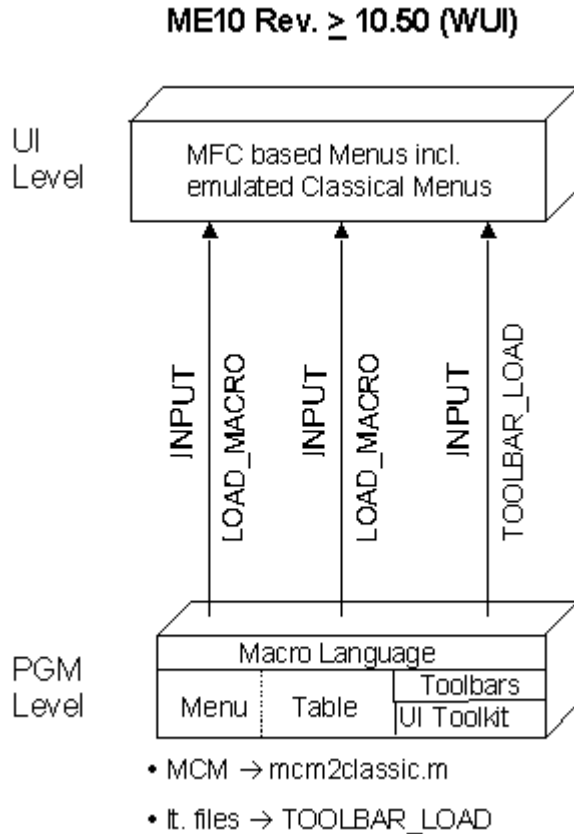
Note the following:

- Fonts are defined (by default) via **Control Panel ▸ Display ▸ Appearance** but can be overridden via the MEFONT and -font commands.

- The `Bcol0` and `Col00` macros (which control the background and text colors in menus) can also be defined via **Control Panel ▸ Display ▸ Appearance**. All other colors will be retained as before.

- All menus can be docked or moved outside of the Creo Elements/Direct Drafting window by clicking on the menu frame and dragging it to its new position.

- Buttons are mouse sensitive. Moving the cursor over a button will raise the button.

- 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 will always remain fixed as defined in the underlying macro. Note that these menus will not be saved for subsequent relaunch when you exit Creo Elements/Direct Drafting.

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Using Fly By Highlighting and Handles

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 on the element) (**B**).
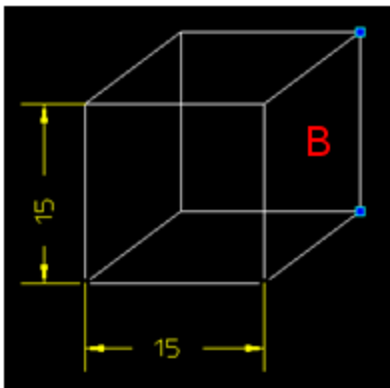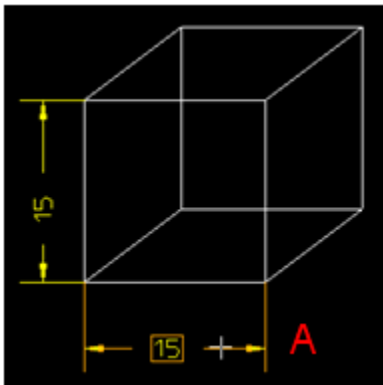
A command is active

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.
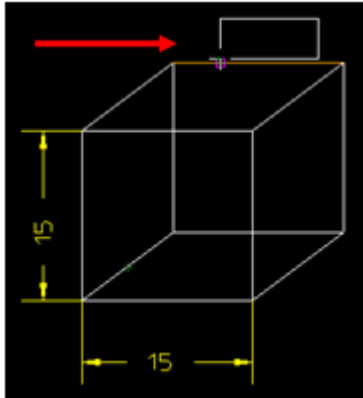
Selection of parts

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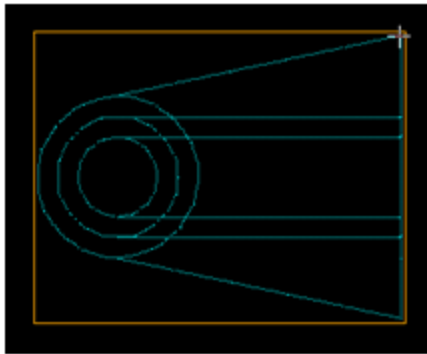
Fly By Highlighting Settings

Enable and disable FlyBy highlighting and change the color of the highlight in the **System** menu: **Setup ▸ Edit Environment ▸ System ▸ FlyBy** tab.

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.

*   a frame around text, symbols, and parts.
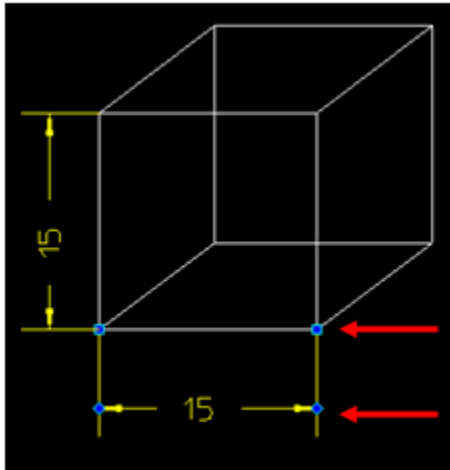
Modify with handles

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 a the new location.

  - Release.

To show the handles on more than one element at a time, hold the Shift key and pick the elements.

To remove the handles, press the ESC key or type MH_CLEAR.

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 on page 226 dimension lines.

- 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: Hold the Shift button and drag the text.

Text/Symbols: Move the text or symbol box.

1. Hover over the text or symbol. Creo Elements/Direct Drafting highlights a box around the text or symbol (if Fly By Highlighting is on).
2. Drag the text or symbol to a new location.

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

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 refline and drag.
- Stretch reference line segments: Click inside the refline handles and drag.

Delete with handles

1. Click on the item to be deleted. Creo Elements/Direct Drafting highlights the item or displays handles.
2. Press **Delete** on your keyboard.

Handles Settings

Enable and disable the handles and change handle settings in **Setup ▸ Edit Environment ▸ System ▸ Handles** tab.

Handles

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

Selection

You must click the element to show the handles.

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

Allowed types

Select which elements will display handles when they are clicked.

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Grouping

You can group annotations or parts together. Groups may consists of a single dimension, multiple text elements, parts, and multiple symbols. Each group has one of its elements as the master element.

Creo Elements/Direct Drafting creates one invisible refline 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 refline 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 ▸ Groups ▸ Group**.
2. Enter GLOBAL if you wish to group elements from different parts.
3. Select the master element for the group.
4. Select additional elements to add them to the group.
5. Click **End**.

Add elements to an existing group

To add elements to an existing group:

1. Click **Modify ▸ Groups ▸ Add Group**.
2. Select an element belonging to the group to which you wish to add.
3. Select elements to add them to the group.
4. Click **End**.

Remove elements from an existing group

To remove elements from an existing group:

1. Click **Modify ▸ Groups ▸ Del Group**.
2. Select an element belonging to the group to which you wish to add.
3. Select elements to remove them from the group.
4. Click **End**.

Ungroup all elements of a group

To ungroup all elements in an existing group:

1. Click **Modify ▸ Groups ▸ Ungroup**.
2. Select an element belonging to the group you wish to ungroup.
3. Click **End**.

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.

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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. To do this, install and start the Windchill Unified Workgroup Manager (UWGM) and set the UWGM server as the primary server.

2. Click **Modules ▸ Windchill WGM** in Creo Elements/Direct Drafting to activate the Windchill Workgroup Manager. The **Windchill** menu is now available in Creo Elements/Direct Drafting.

Windchill PDM commands

The following table explains the PDM commands that are inserted into Creo Elements/Direct Drafting when connected to the Windchill Workgroup Manager.

Action	Menu Location in Windows and Mixed User Interface	Description
Open from Windchill	**Windchill ▸ Open from Windchill** [Also] **File ▸ Open from Windchill**	Presents 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 Workspace	**Windchill ▸ Save to Workspace** [Also] **File ▸ Save to Workspace**	Presents the **Save to Workspace** window. Upon execution of the save, the

Action	Menu Location in Windows and Mixed User Interface	Description
		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**	Cancels the check out of the current document. 📝 **Note** If the document was checked out via the **Revise and Check out** action, the higher revision label is maintained.
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**	Presents 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**	Presents the **Open from Windchill** window, allowing

Action	Menu Location in Windows and Mixed User Interface	Description
		you to navigate to a library containing frames and select a frame to load into the current drawing. The default location shown can be defined by the Windchill server preference, **Default Drawing Frame Location**.
Save Frame to Windchill	**Windchill ▸ Save Frame to Windchill**	Presents 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 Manager**	Opens the **Server Manager** window, where you can view or change Windchill servers and server active workspaces.
Event Management	**Windchill ▸ Managers ▸ Event Manager**	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 in Windows and Mixed User Interface	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

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.

AIl 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 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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