Inspection with Creo Elements/Direct 2D Access Windows User Interface

Creo Elements/Direct 2D Access 20.2.4.0

Contents

Preface

Creo Elements/Direct 2D Access is a screen-only View and Plot facility for integration within an existing CAD system. The Creo Elements/Direct 2D Access software allows you to easily access any Creo Elements/Direct Drafting format 2D drawing.

1

Getting Started with Creo Elements/Direct 2D Access

About Creo Elements/Direct 2D Access

Creo Elements/Direct 2D Access provides a full set of commands for viewing, measuring, and plotting any MI format drawings created in the Creo Elements/ Direct Drafting 2D CAD system. In Creo Elements/Direct 2D Access, you cannot damage or alter such drawings in any way.

To familiarize yourself with Creo Elements/Direct 2D Access, it may be a good idea to complete the tutorial provided in Tutorial on page 113.

The functionality of Creo Elements/Direct 2D Access is a subset of the Creo Elements/Direct Drafting functionality. For further details and illustrated examples of this functionality, refer to the Creo Elements/Direct Drafting User's Guide supplied on the Creo Elements/Direct Electronic Manuals CD.

Starting Creo Elements/Direct 2D Access

To start Creo Elements/Direct 2D Access, click:

Start ▶ All Programs ▶ PTC ▶ Creo Elements Direct 2D Access [version].

📃 Note

Be sure that the License Server is running before you start Creo Elements/ Direct Drafting. See your Installing and Configuring Creo Elements/Direct Drafting manual for instructions.

The Creo Elements/Direct 2D Access User Interface

The Creo Elements/Direct 2D Access user interface is designed to ensure a short learning curve. The following figure illustrates the general (default) layout of the Creo Elements/Direct 2D Access user interface.

Figure 1. The Creo Elements/Direct 2D Access User Interface
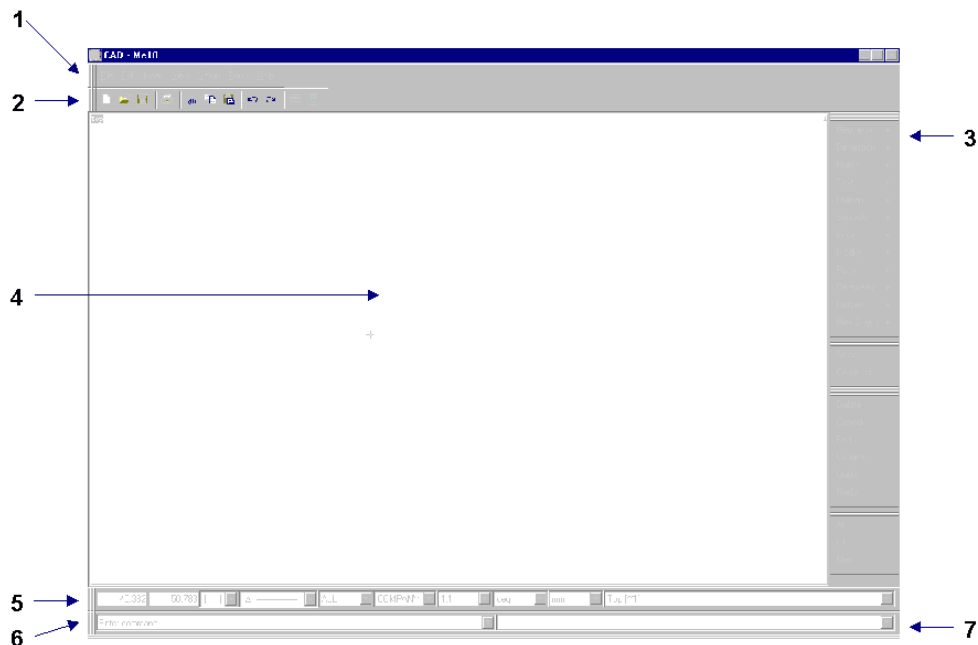
1. The top menu bar. This menu bar provides access to menus such as **File** and **Setup**. Optionally, you can use hotkeys to select menu items.
2. The toolbar. Move the cursor to each of these icons to display the associated tooltips.
3. Viewing-related menus.
4. The drawing area. This is the area where you will view 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.
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 2D Access instances (separate Creo Elements/Direct 2D Access windows) at the same time. Refer to for further information.

Switching Languages

By default, the Creo Elements/Direct 2D Access 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 2D Access installation directory.

📋 **Note**

The Japanese version of Creo Elements/Direct Drafting requires a Japanese operating environment. If you installed Creo Elements/Direct 2D Access in Custom mode, any languages you deselected during installation will not be available at Creo Elements/Direct 2D Access runtime.

Switching to the Classic User Interface

Optionally, you can run Creo Elements/Direct 2D Access using the Classic User Interface. Note that a separate User's Guide is available for this user interface.

To run the Creo Elements/Direct 2D Access 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 2D Access uses the standard Windows style user interface.
MEPELOOK=3	Creo Elements/Direct 2D Access 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 2D Access. Simply enter BEEPER ON or BEEPER OFF into the user input line.

Basic Working Methods

You can quickly and easily delete elements or entire classes of elements from your drawing for visual clarity. The **Select** options help you to select or unselect given classes of elements.

Deleting Elements

Click **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**.

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 2D Access defaults to the current part.

Example

Click **Delete**, **Select**, **Global**, **All**, **Confirm**, **Apply**.

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.

Selecting Drawing Elements

Click **Select** to let Creo Elements/Direct 2D Access know you are about to make a selection. Then press any option or combination of options from the selection block.

All options on the selection block 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.

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

Example

1. Click **Delete**, **Select**; **Color**.
2. Select **Green** from the **Color** menu.
3. Click **All**, **Confirm**, **Apply**.

   This will delete all elements drawn in green in the current part.

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 **Apply**.

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 2D Access prompts you to **Confirm** that you want to delete them. Before you click **Confirm**, you may want to alter your selection by, for example, subtracting or adding some elements.

3. Click **Confirm**.

4. Click **Apply**.

Enclosing Elements with a Polyline

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

Figure 2. Selecting Elements with Polyline
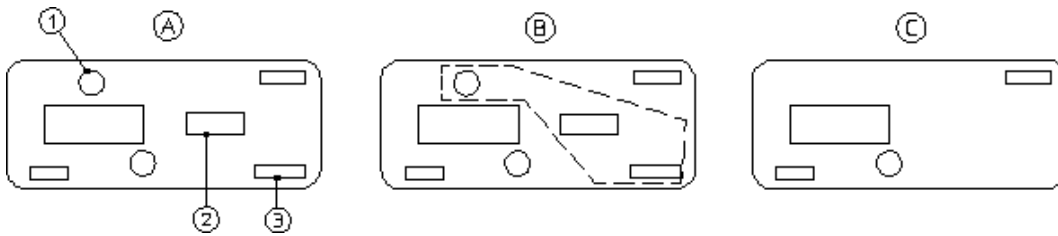
Suppose you want to delete the elements 1, 2, and 3:

1. Click **Delete**.
2. Click **Poly** in **Select**.
3. Click a succession of points to define the enclosing polyline 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**.
3. Click **Confirm**.
4. Click **Apply**.

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 2D Access 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 **C-Geo** 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.

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 **Apply**.

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 **Apply**.

📝 **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 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 2D Access will "remember" your preselection until you replace it by another preselection.

To preselect elements:

1. Click the Preselection icon in the Creo Elements/Direct 2D Access toolbar:
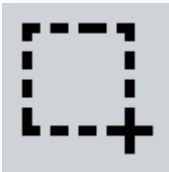
2. Select the elements you want to include in your preselection.

3. Click **Confirm**.

To retrieve your preselection:

1. Click **PreSelected** in the **Select** menu or in the **Advanced** select menu.

2. Click **Confirm**.

Confirming Commands

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

Pasting Elements from Other Applications

	Enables you to paste elements from other applications into Creo Elements/Direct 2D Access.

Refer to Linking and Embedding OLE Objects on page 73 for instructions on linking and embedding OLE objects in Creo Elements/Direct 2D Access.

Undo and Redo

In Creo Elements/Direct 2D Access, 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** or **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** or **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.

Exiting Creo Elements/Direct 2D Access

To exit Creo Elements/Direct 2D Access, pull down the **File** menu and click **Exit**.

Customizing Creo Elements/Direct 2D Access

You can quickly and easily customize Creo Elements/Direct 2D Access for your personal requirements. Detailed instructions are provided in the following HTML format document on your Creo Elements/Direct Electronic Manuals CD:

`/htm/english/viewer/appb.html`

2

Finding, Loading, and Handling Files

This chapter assumes that you are familiar with the directory structure and general file handling methods of your computer.

Using the File Browser

You can open a drawing file quickly and easily by means of the Creo Elements/ Direct 2D Access 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 File Browser, you can move, copy and delete files and directory using standard MS-Windows techniques.

Opening a Drawing File via the File Browser

To open a drawing file by means of the Creo Elements/Direct 2D Access File Browser:

1. In Creo Elements/Direct 2D Access, pull down the **File** menu.

2. Click one of the following options to display the File Browser:

Open	The drawing file you select or specify will replace any drawing already loaded in the Creo Elements/Direct 2D Access workspace.
Load File	The drawing file you select or specify will be added to any drawing already loaded in the Creo Elements/Direct 2D Access workspace.

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. Optionally, select a mode in which to load the drawing file:

As Subpart	Loads a drawing as a subpart to a drawing that is already loaded.
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.

5. Click on the filename. If a file is provided in a format other than MI (the default), 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)

   - **Autodesk Drawing Interchange (*.dxf)**

   - **Initial Graphics Exchange Format (*.igs)**

   - **AutoCAD File Specification (*.dwg)**

   - Creo Elements/Direct Modeling **Bundle (*.bdl)** files. (Note that only the 2D portions of such files are extraced, while any 3D portions are ignored.)

   A preview image of the drawing file is displayed in the preview pane of the File Browser.

6. Click **Open** to open this file in Creo Elements/Direct 2D Access. Alternatively, enter the filename and press **[Enter]**.

7. Once the drawing is displayed in Creo Elements/Direct 2D Access, you can view and measure it.

> 📋 **Note**
> The preview image is pixel-based. It cannot be modified.

Opening a Drawing File by Means of the Navigator

The Creo Elements/Direct 2D Access 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 2D Access Navigator:

1. In Creo Elements/Direct 2D Access, pull down the **File** menu and click **Open** to display the File Browser.

2. Click **Navigator** to display the Creo Elements/Direct 2D Access Navigator.

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 2D Access.

6. Once the drawing is displayed in Creo Elements/Direct 2D Access, you can view and measure it.

📄 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.

Creating a New Creo Elements/Direct Drafting Instance

Optionally, you can run multiple instances of Creo Elements/Direct 2D Access at the same time.

To create a new Creo Elements/Direct 2D Access instance:

1. In Creo Elements/Direct 2D Access, pull down the **File** menu and click **New ▶ Window**. Alternatively, click the following button in the Creo Elements/Direct 2D Access toolbar:

Note

- Each instance will run in a separate window.

- All Creo Elements/Direct 2D Access instances are independent of each other; there is no master or slave instance.

- Only a single Creo Elements/Direct 2D Access license is required for any number of Creo Elements/Direct 2D Access instances run by the same user on the same machine/display.

- Newly created Creo Elements/Direct 2D Access 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 2D Access instance, this library is not automatically available in another Creo Elements/Direct 2D Access instance.

Opening an Existing Drawing File in a New Creo Elements/Direct 2D Access Instance

To open an existing drawing file in a new Creo Elements/Direct 2D Access instance:

1. In Creo Elements/Direct 2D Access, 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 2D Access instance.

Using the CATALOG Command to List Files

The CATALOG command enables you to generate a list of files in a directory:

1. Enter **CATALOG** into the user input line in Creo Elements/Direct 2D Access.

2. Specify the directory to be listed.

3. Specify the applicable output option: SCREEN, APPEND, DEL_OLD, or prt (printer).

Although this catalog only displays five catalog items, each file has eleven catalog items:

- The filename, `"FILE_NAME"`.
- The file physical name, `"PHYS_NAME"`.
- The file size, `"FILE_SIZE"`.
- The file type, `"FILE_TYPE"`.
- The file description, `"FILE_DESC"`.
- The file create date, `"CREATE_DATE"`.
- The last file modify date, `"MODIFY_DATE"`.
- The last file access date, `"ACCESS_DATE"`.
- The number of accesses, `"NUM_ACCESS"`.
- The program name, `"PROGRAM"`.
- The program revision, `"REVISION"`.

Generating Sorted Catalogs

There are many options and methods of producing different types of sorted catalogs. This section describes the main methods. When you have sorted a catalog, you can:

- Display the catalog on the screen.
- Print the catalog using a printer.
- Store the catalog in a file.

The first step in producing a sorted catalog is to enter the **CATALOG** command and specify the directory you want to catalog. Creo Elements/Direct 2D Access then prompts for the catalog option that will carry out the type of sort you want. The catalog options include:

- Listing Files Contained in the Directory
- Sorting a Directory from Smallest Value to Largest
- Sorting a Directory from Largest Value to Smallest
- Selecting Items to Produce a New Catalog

Relational Operators

After you define the catalog item to select on, you need to enter a relational operator. This relational operator specifies the sort operation to be performed.

The following relational operators are available:

Character	Meaning
=	equal to (default operator)
<>	not equal to
>	greater than
<	less than
>=	greater than or equal to
<=	less than or equal to

For example, if you want to find all files whose file size is equal to 3000 bytes, you would enter:

```
"FILE_SIZE" = 3000
```

The catalog item is FILE_SIZE, the relational operator is =, and the item value is 3000.

Item Values

Item values are necessary when you produce sorted catalogs. You will need to define the catalog item, the relational operator, and the item value for the sort.

When you enter an item value that contains letters, you must enter the item value in 'quotation marks'. This is not necessary for item values consisting of digits only.

Wildcards

You can use wildcards when you define the item values for your catalog output. Wildcards are special characters that perform additional exclusive selections. Refer to the Creo Elements/Direct 2D Access Help for further details.

Sorting a Catalog

To sort the file catalog from smallest to largest or vice versa:

1. Enter **CATALOG**.

2. Enter the 'name' of the directory you want to sort. For the current directory, enter **''**.

3. Enter SORT to sort from smallest to largest (sort up) or REVERSE_SORT to sort from largest to smallest (sort down).

4. Enter a catalog item such as "FILE_SIZE", or "FILE_NAME".

5. Enter output option: SCREEN, DEL_OLD, APPEND, or prt (printer).

Creo Elements/Direct 2D Access sends the catalog to the defined output.

Importing Macros, Fonts and HPGL Files

You can import the following files into your current Creo Elements/Direct Drafting drawing:

- Creo Elements/Direct Drafting Macros
- Fonts
- HPGL Files

Importing a Creo Elements/Direct Drafting Macro

To import a Creo Elements/Direct Drafting macro:

1. In Creo Elements/Direct 2D Access, 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 2D Access, 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 2D Access.
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 2D Access.

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 for use in Creo Elements/Direct 2D Access:

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 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 2D Access, 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. Specify the position for the imported HP-GL file.

3

Viewing Drawings in Creo Elements/Direct 2D Access

This chapter describes how to view a drawing in Creo Elements/Direct 2D Access.

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.

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 where you want to create the new view.

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 a view:

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.

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. All viewports are continuously updated.

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 and .

Figure 3. Using Viewport 2 as a Reference
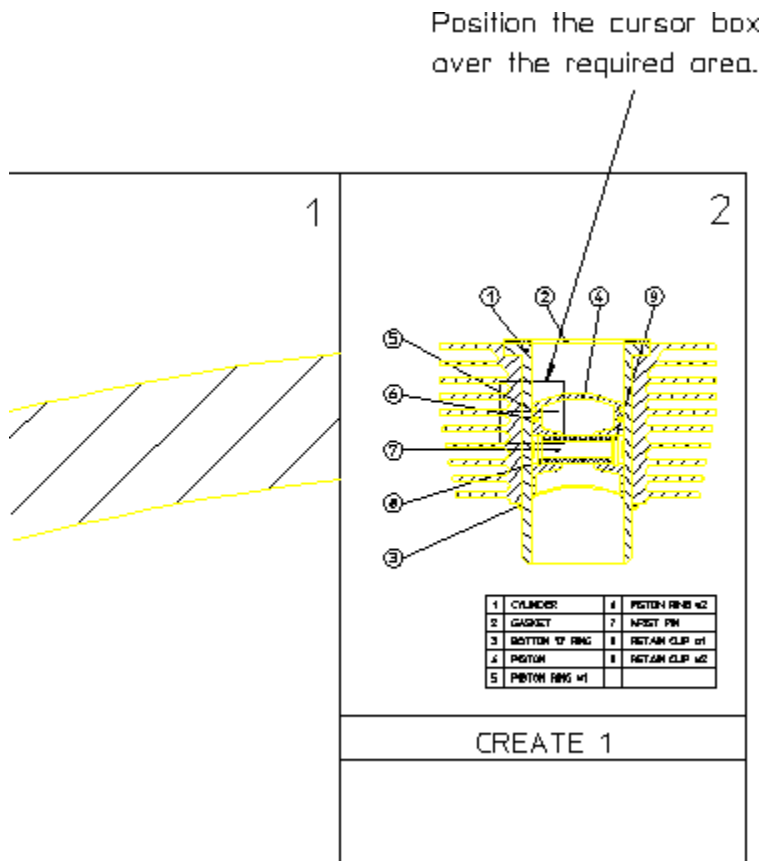
Figure 4. Viewing Piston Assembly Detail in Viewport 1
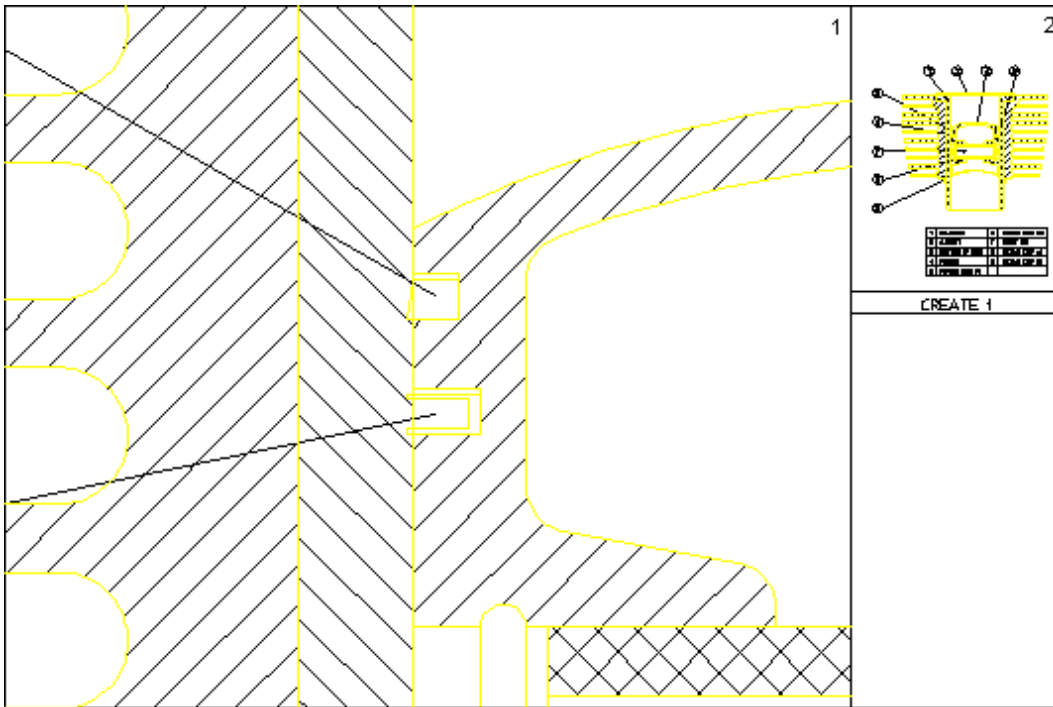
Creating a Viewport

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 2D Access drawing area.

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

2. Click **Viewport** and **Large**.

3. Click the viewport.

Any other viewport are now hidden begind 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 ▸ Apply**.

   To view your current drawing in Creo Elements/Direct 2D Access after you have deleted all viewports, you need to create at least one new viewport.

Viewport Settings

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

To define the viewport settings:

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

2. Click **Viewport** and **Settings**.

3. In **Viewport Settings**, select the viewport border color and specify the viewport border width. Select the background, text and cursor colors.

Selecting What is Displayed on Your Screen

The **Show** dialog box is accessible via the **Setup** menu as well as from the right-hand menu bar.

Inspection with Creo Elements/Direct 2D Access Windows User Interface

You can selectively display or hide:

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

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 all dimensions on or off.

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.

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 hels 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 enance performance.
4. Click **OK** to save these settings.

4

Measuring in Creo Elements/Direct 2D Access

This chapter describes how to measure in Creo Elements/Direct 2D Access. The combination of **Catch** and **Measure** functions provides a powerful tool for fast and accurate measurements.

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 2D Access 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.

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 Prompt 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 Prompt Line and press **[Enter]**. Creo Elements/Direct 2D Access draws the vertical line as specified. The length of this line is the same as that of the line you measured.

📝 Note

- The ruler Follow function 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 Prompt 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 2D Access calculates and displays the following area properties for a single face:

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

Multiple Faces

If you identify multiple faces, Creo Elements/Direct 2D Access 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 2D Access 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 2D Access 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.

Setting the Units of Measurement

You can select linear and angular units from the bottom menu bar. The current units are displayed in the System Status Line. 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 5mm allows you to work in 5mm units, and an input of 5 degrees allows you to work in 5 degree units.

Using the Calculator

Creo Elements/Direct 2D Access 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 the 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** and **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** and **Relative**.

Inspection with Creo Elements/Direct 2D Access Windows User Interface

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** and **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** and **Place**.

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

Figure 5. Ruler Axes on page 46 shows the ruler axes.

Figure 5. Ruler Axes
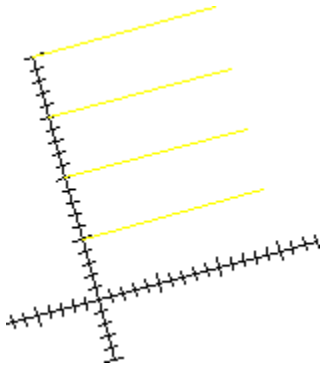
5

Parts

To understand how to use the **Parts** commands, it is necessary for you to have a basic understanding of parts.

A complex drawing is likely to contain several parts. Design Engineers use parts to simplify the drawing into clearly defined groups of elements. Each part has a partname. It is possible to view or highlight only one part of a complex drawing at a time. This provides a clearer visual representation of the part in relation to the entire drawing.

Parts Structure

The use of parts in a drawing results in a parts structure. A parts structure can be thought of as a family-tree showing the relationships between all the parts. The structure is hierarchical and has many levels, each level containing information about the relationship between each part and its subparts. The highest level of the parts structure is called Top. The lowest level contains parts that cannot be subdivided any further.

If a part has subparts then it is called a parent part.

Summary

*   Complex drawings are likely to contain parts.

*   Parts are groups of elements.

*   Each part has a name (partname).

*   Parts containing subparts are called parent parts.

*   You can selectively view, or highlight any part or group of parts, either by means of the , or by means of the following commands.

Editing Parts

Three command options are available to edit parts: editing a part, editing a top part, and editing the parent part. Creo Elements/Direct 2D Access displays the active part in normal linetypes and colors while all the other parts are shown in magenta dotted lines.

Editing a Part

To set a part as active:

1. Enter `EDIT_PART`
2. Enter the part name, or click an element of the part.

Creo Elements/Direct 2D Access sets the clicked part as the active part.

Editing a Top Part

To set the highest part in the parts structure as active, enter `EDIT_PART TOP`.

Editing the Parent Part

As discussed earlier, parts can contain subparts. To set the parent of the active subpart as the new active part, enter `EDIT_PART PARENT`

Producing a Parts List

Use the PARTS LIST command to produce a list of the parts in your drawing. The list shows the relationship between the active part and all the other parts.

Creo Elements/Direct 2D Access can display the parts list on the screen, or it can print the list on a locally-connected printer. The following instructions are for displaying the list on the screen.

Counting Parts

Use the Count option to list all the parts one level below the active part and all the parts owned by the active part. Before using this command, check that you have the correct active part set. To list these parts:

1. Enter `PARTS_LIST` and then **SCREEN**.

   Creo Elements/Direct 2D Access displays the list of parts in a separate text editor window.

Producing a Parts Tree

Use the Tree option to produce a full parts list of the parts in your drawing. The active part has a * next to it. The list displays the hierarchical relationship between each part. This is shown by indentation of the part name. To produce a full parts list:

1. Enter `PART_LIST TREE SCREEN`.

   Creo Elements/Direct 2D Access displays the list of parts in a separate text editor window.

Showing Unselected Parts as Boxes

Use the following commands to look at selected groups of parts. Creo Elements/ Direct 2D Access replaces each part not selected by a box. The command works only in the active viewport. Using boxes is useful when parts overlap.

With Part

To draw a part as a box representation:

1. Enter `SHOW_PART`.
2. Click on the part.

Creo Elements/Direct 2D Access redraws the part as a box. Also, use this command to draw a boxed part as a normal part.

With All

To restore all parts to their normal appearance, enter `SHOW_PART ALL`.

With Part Boxed

To change a part and all its subparts to a box representation:

1. Enter `SHOW_PART PART_BOX`
2. Enter the parent part name, or pick an element of the parent part.

Creo Elements/Direct 2D Access redraws the parent part and all the subparts.

Viewing Parts

Use the **View** commands to look at individual parts of your drawing. The command works only in the active viewport.

Viewing a Part

To look at an isolated part:

1. Enter **VIEW**
2. Enter the part name, or click on an element of the part.

Creo Elements/Direct 2D Access draws only this part. Creo Elements/Direct 2D Access displays the part name in the top left-hand corner of the active viewport.

Viewing All Parts

To view all parts of a drawing, enter `VIEW TOP`.

Creo Elements/Direct 2D Access redraws all the parts making up the drawing. Creo Elements/Direct 2D Access redraws the part name in the top left-hand corner of the active viewport. This is useful if you have been viewing a lower level and want to quickly redraw the drawing.

Viewing Parent Part

To view the parent of the currently viewed part, enter `VIEW PARENT`.

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 Creo Elements/Direct 2D Access window.

What is Shown in the Part Browser?

The Part Browser provides a graphical overview of the current parts structure in your 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	

Column	Description
	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. Creo Elements/Direct Drafting fits this part into the current viewport. All other parts are hidden, and their icons are marked by a yellow X (see list of Part Browser icons above).	
Symbol	Changes the part to a symbol.	
Show	This command provides the following options:	
	Show	Shows the selected part (if it is presently shown as a box or if it is hidden).
	ShowFlat	Shows the selected part but not its subparts (if any).
	ShowAll	Shows the entire selected part complete with all of its subparts.
	ShowBox	Shows the selected part as a box.
Hide	This command provides the following options:	
	Hide	Hides the selected part.
	HideFlat	Hides the selected part but not its subparts (if any).
	HideAll	Hides the entire selected part complete with all of its subparts.
Options	The **Icons** option enables you to display the Part Browser with or without icons. The **State Icons** option enables you to show or hide the (checkbox-like) state icons that indicate whether or not a part is visible.	

Save	You can click **Save** to save an ASCII representation of the current parts tree structure in a Creo Elements/Direct Drafting list file.
Delete	Deletes the selected part.

Spotlighting Parts

You can distinguish between the active part and all other parts by using a spotlight. The command is a simple On/Off switch. The SPOTLIGHT is on when you first load Creo Elements/Direct 2D Access.

• To switch the spotlight off, enter `SPOTLIGHT OFF`.

• To switch the spotlight on again, enter `SPOTLIGHT ON`

6

Viewing Hidden Lines

Introduction to Hidden Lines

This chapter shows how to display hidden lines in assembly drawings.

The 2D hidden line module is used with assembly drawings to automatically display lines that indicate hidden drawing features. In an assembly drawing, these lines are normally shown dashed.

Click **Hidden** to access the hidden line menu.

Before you can display an assembly drawing using the hidden lines mode, you must check to see that the faces of your drawing have been assigned z-values.

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 and geometry lines (depending upon their z-values).

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 6. Faces - With Z-Values 0 and 1. on page 55 shows the faces of two components (one with z-value 0 and the other with z-value 1).

Figure 6. Faces - With Z-Values 0 and 1.
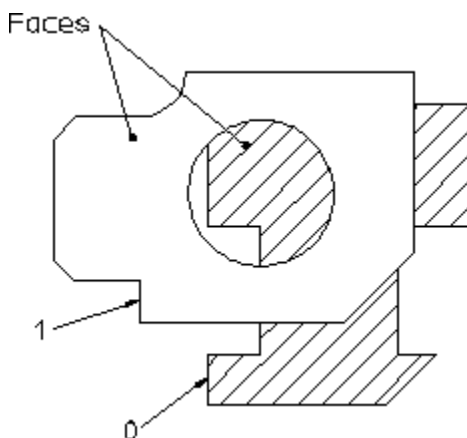
What are Z-Values?

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.

See Figure 7. Visualizing Z-Values. on page 56.

Figure 7. Visualizing Z-Values.
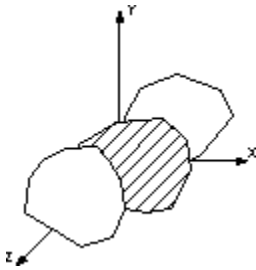
A face or line with a higher z-value covers a face or line with a lower z-value.

Figure 8. Z-Values 0, 1 and 2. on page 56 shows 3 faces assigned the z-values 0, 1 and 2.

Figure 8. Z-Values 0, 1 and 2.
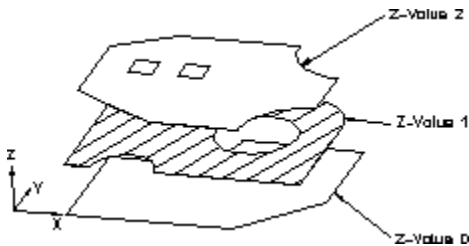
Hatches are treated as faces, this means hatched areas can cover geometry and faces. Hatches that are partially covered are displayed partially obscured.

Construction geometry, texts, leader lines and dimensions are always visible.

> **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.

Checking Z-Values

You can check the z-values of:

* Elements
* Shared parts
* Elements in shared parts

The z-value will be displayed on the User Prompt Line.

Checking the Z-Value of an Individual Element

To check the z-value of an individual element:

1. Click **Inquire Z Value** in **Hidden**.

2. Click **Absolute** in **Inquire Z Value**.

3. Select the element.

Checking the Minimum or Maximum Z-Value in an Assembly Drawing

To check the minimum or maximum z-value:

1. Click **Inquire Z Value** in **Hidden**.

2. Click **Minimum** in **Inquire Z Value**.

Checking the Z-Value of a Shared Part

To find the computed z-value of a shared part:

1. Click **Inquire Z Value** in **Hidden**.

2. Click **ShPrtOffset** in **Inquire Z Value**.

3. Select the shared part.

   Creo Elements/Direct 2D Access computes the z-value as the sum of the shared part's z-value and that of the shared parent part.

Checking the Z-Value of an Element in a Shared Part

To find the z-value of an element in a shared part:

1. Click **Inquire Z Value** in **Hidden**.

2. Click **ShPrt Elem** in **Inquire Z Value**.

3. Select the element in the shared part.

Generating Hidden Lines

When you click **Change to Hidden** to generate hidden lines, Creo Elements/Direct 2D Access does the following:

• Geometry is modified by splitting geometric components if they are covered by faces with a higher z-value.

• Splines are converted.

• Shared parts are unshared.

- The parts tree, starting from the current part, will be smashed.

Showing Hidden Lines On/Off

Once you have generated hidden lines, you can switch the hidden lines on or off by clicking **On** or **Off** in **Hidden Mode**.

Showing Hidden Lines as Dashed Lines

With **Hidden Mode** switched on, you can display hidden lines as dashed lines by ticking **Show Hidden**.

Plotting Hidden Lines

If you are plotting your drawing, note that it is not possible to plot face colors.

In order to suppress plotting of hidden lines you must set the hidden line linetype and color appropriately. For more information, see the PLOT_ TRANSFORMATION command in the online help.

7

Using Construction Geometry

This section shows how to add construction lines and circles to a drawing displayed in Creo Elements/Direct 2D Access. Construction geometry can be used to check the alignment of existing geometry.

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 either click positional points in the Creo Elements/Direct 2D Access drawing area, or you can enter the coordinates (separated by a comma) into the User Input Line.

Creating Construction Lines

The following commands allow you to draw construction lines.

Construction Line by Two Points

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

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.

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.

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.

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 2D Access 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 currenlty defined for your Creo Elements/Direct 2D Access environment.

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 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.

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.

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.

Concentric Construction Circles

	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 Diameter

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

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

	Click the approximate point of tangency of the construction circle on the existing element. Next specify a point on the circumference, then either specify another point or enter the required radius.

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 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). Next specify a peripheral point on the construction circle or enter the desired radius. Creo Elements/Direct 2D Access 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.

8

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.

The differences between the original and revised drawings are shown in color.

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 and the name of the revised drawing file.

   Creo Elements/Direct 2D Access displays the two drawings overlapping.

3. Click **Geometry** in **Compare**.

   Creo Elements/Direct 2D Access 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.

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**.

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 **Color Settings** 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.

The color changes take effect with the next redraw.

9

Using URLs with Creo Elements/ Direct 2D Access

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 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 for instructions.

3. Select a URL from the list.

4. Click **Show Elem** or **Show Parts** 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 2D Access 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 2D Access 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 2D Access help system:

Enter `help layer` or `help info` or `help url` at the Creo Elements/Direct 2D Access command line. The screen will clear and Creo Elements/Direct 2D Access will display further information about using layers or infos or URLs with Creo Elements/Direct Drafting.

10

Linking and Embedding OLE Objects

Files (such as spreadsheets, graphics or text pages) created in external applications can be included in Creo Elements/Direct 2D Access by inserting associated OLE objects.

Inserting an OLE Object

To insert an OLE object into a drawing in Creo Elements/Direct 2D Access

1. In Creo Elements/Direct 2D Access, 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 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 2D Access. Optionally, click **Change Icon** to select a different icon for the OLE object.

3. 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 2D Access.

   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 2D Access.

4. If you use the **Display As Icon** option, the OLE object will be represented by the associated application icon in Creo Elements/Direct 2D Access. Optionally, click **Change Icon** to select a different icon for the OLE object.

5. Click **OK**.

6. 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 2D Access.

   The OLE object is displayed in the Creo Elements/Direct Drafting drawing.

Editing an OLE Object in Creo Elements/Direct 2D Access

Once the OLE object is inserted in the drawing, you can edit it as described in the following sections.

Deleting an OLE Object

To delete an OLE object from your drawing:

1. Pull down the **Insert** menu.

2. Click **OLE** and **Delete**.

3. Select the OLE object to be deleted.

Moving an OLE Object

To move an OLE object:

1. Pull down the **Insert** menu.

2. Click **OLE**, **Edit** and **Move**.

3. Specify a reference point.

4. Specify a destination point.

5. Select the OLE object to be moved.

Scaling an OLE Object

To scale an OLE object:

1. Pull down the **Insert** menu.

2. Click **OLE**, **Edit** and **Scale**.

3. Specify the center of scaling.

4. Enter the scale factor for the X-direction.

5. Enter the scale factor for the Y-direction.

6. Select the OLE object to be scaled.

Arranging OLE Objects

To arrange OLE objects:

1. Pull down the **Insert** menu.

2. Click **OLE**, **Edit** and **To Front** or **To Back**.

3. Select the OLE object you want to move to the foreground or background.

Hiding and Showing OLE Objects

To improve system performance or to reduce screen clutter, OLE objects can be hidden with the **Show** command. If the drawing includes several OLE objects, you can choose to hide individual or all OLE objects. The **Show** options are:

On	Unhides a hidden OLE object
Off	Hides an OLE object
Box	Hides an OLE object but places a reference frame around the hidden image
Icon	Displays an icon that represents the OLE object 💡 **Tip** When zooming with the dynamic mouse (CTRL + right mouse button), hide the OLE objects with the **Box** option.

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 you want to activate.

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 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

11

Creo Elements/Direct Annotation

This chapter discusses the Creo Elements/Direct 2D Access Annotation module for use in Creo Elements/Direct 2D Access.

Introduction

The Creo Elements/Direct **Annotation** module included in Creo Elements/Direct 2D Access enables you to visualize MI format 2D drawings generated by Creo Elements/Direct Annotation.

General Procedure

The general Creo Elements/Direct 2D Access Annotation procedure comprises the following steps:

• Prepare the Creo Elements/Direct 2D Access drawing environment for Creo Elements/Direct Annotation.

• Load an MI file generated in Creo Elements/Direct Annotation.

• View this file as appropriate. For example, you can manipulate views and you can hide or delete elements. Note that a warning message is displayed for any "structurally dangerous" operations that would impair 3D/2D associativity.

• Following 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 Creo Elements/Direct Annotation MI file by means of the Part Browser.

Creo Elements/Direct Annotation Rules

To ensure that the Creo Elements/Direct 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 2D Access **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 Annotation). Only use the **Sheet** commands provided by the Creo Elements/Direct 2D Access **Annotation** module.

• Do not change or delete info texts beginning with "SD_" or "DOCU" or "AM_ ".

• Only use the **View** commands provided by the Creo Elements/Direct 2D Access **Annotation** module.

- Do not scale view parts or 3D parts below view parts with the command in the **Parts** menu of Creo Elements/Direct 2D Access. Only use the **Scale** command provided by the Creo Elements/Direct 2D Access **Annotation** module.

Limitations

Some operations performed in the Creo Elements/Direct 2D Access **Annotation** module will not deliver the expected results after reloading and updating in Creo Elements/Direct Annotation.

- Parts created in 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 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 Drafting will not behave associatively in Creo Elements/Direct Annotation.

- Hatch holes automatically created in Drafting (forced by dimensions or texts when 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 Drafting when the text is changed in Drafting.

- Hatches that were automatically created in Creo Elements/Direct Annotation but deleted in 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 Drafting get their original name during the next update in Creo Elements/Direct Annotation.

- If you split or merge 2D geometry in 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 2D Access Drawing Environment

Before you load an MI file generated in Creo Elements/Direct 2D Access 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. Creo Elements/Direct Annotation does not update the preview data in the Creo Elements/Direct Annotation MI file loaded in Creo Elements/Direct 2D Access. Therefore, switch off the **Preview** mode by entering the following in the Creo Elements/Direct 2D Access user input line:
```
PRE_VIEW
OFF
```

3. Activate the Creo Elements/Direct 2D Access **Annotation** module by clicking **Modules**, **Annotation**.

Loading a Creo Elements/Direct Annotation Drawing

When you have prepared the Creo Elements/Direct 2D Access drawing environment as discussed in Preparing the Creo Elements/Direct 2D Access Drawing Environment on page 80, proceed as follows to load an MI file generated in Creo Elements/Direct Annotation.

1. In **Annotation**, click **Drawing ▶ Open**.
2. Select the MI file you want to load from the **Open Annotation** 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 **Annotation** module provides a number of commands for selecting and changing 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.

Hiding Hidden Lines

To hide hidden lines in the Creo Elements/Direct Annotation MI file:

1. In **Annotation**, click **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 Creo Elements/Direct Annotation MI file:

1. In **Annotation**, click **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 Creo Elements/Direct Annotation MI file:

1. In **Annotation**, click **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 Creo Elements/Direct Annotation MI file:

1. In **Annotation**, click **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 2D Access **Annotation** module is a 2D derivative of a 3D model view in Drafting. When the underlying 3D model changes in Drafting, you can update the associated view in Creo Elements/Direct 2D Access **Annotation** and check for differences in the view before and after this update.

Showing Differences

In **Annotation**, click **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 **Reset Show**. Note that **Reset Show** returns the view to the state prior to **Show Difference**.

Changing Hidden Line and Tangent Line Colors and Linetypes

In the **Annotation** module, you can change the color and linetype of hidden lines and tangent lines.

1. In **Annotation**, click **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.

12

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 2D Access spools all plots and screendumps 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 2D Access 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 screendump.

The Clipboard temporarily stores a plot or screendump 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 2D Access. 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 2D Access 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 2D Access 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.cfg`) 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 or Creo Elements/Direct 2D Access 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' END

WIN_PRT_MGR PAPER 'a4' ORIENTATION PORTRAIT COPIES 1 END
PLOT_SCALE AUTO
PLOTTER_TYPE ADD MSWINDOW_GDI_PRINTER PRT_DRIVER 'HP DeskJet'
'department_2' END

WIN_PRT_MGR PAPER 'a4' ORIENTATION LANDSCAPE COPIES 1 END
PLOT_SCALE AUTO
PLOTTER_TYPE ADD MSWINDOW_GDI_PRINTER PRT_DRIVER 'HP LaserJet
1100' 'department_3' END

PLOTTER_TYPE 'department_3'

{ plot it }
PLOT 0,0 0 all BW_IMG NORM_VIDEO
```

The settings in this `customize_p.m` print configuration file imply the following:

• One copy of the drawing is printed to A4 paper in landscape orientation.

• The drawing is scaled to fit.

• The PLOTTER_TYPE variable is set to `department_3`. This is the printer that will be used as the output device.

• The entire drawing is printed in normal (as opposed to inverse) video.

Re-Using a Saved Print Configuration

When you quit Creo Elements/Direct 2D Access, the system automatically saves your current print settings. This will be the setting used when you start Creo Elements/Direct 2D Access 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.
2. In the **Print** menu, Click **Settings.** The **Print Configurations** dialog box appears**.**
3. In the **Print Configurations** dialog box, Click **Load**. The **Load Print Configurations** dialog box appears.
4. In the **Load Print Configurations** dialog box, select the print configuration file (such as test.cfg) you want to load.
5. Click **Open**. The respective print configuration name appears in the **Name** selection list of **Print Configurations** dialog box.
6. Select the print configuration from the **Name** selection list.
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 2D Access 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. You can also define the output destination of a file by clicking **Browse** from the **Print** menu.

> ### Note
> Creo Elements/Direct 2D Access 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. Pull down the **Name** selection list in the **Printer** section of the **Print** menu.
2. Select the printer or plotter from this list.
3. 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.

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, be sure to select an HP-GL compatible plotter.
- Alternatively, if you want to create a PostScript file, be sure to select a PostScript printer.

Inspection with Creo Elements/Direct 2D Access Windows User Interface

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 **Size** 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

To set the print/plot orientation, click the **Portrait** or **Landscape** radio button.

> **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

The two **Position** options are:

- **Center**
- **Left**

Centering the Drawing

This is the default option.

To center your drawing on the page, click the **Center** radio button.

To center the drawing, Creo Elements/Direct 2D Access places an unseen box around all outermost dimension text, drawing lines, or text. This box is then centered on the paper.

Left Justifying the Drawing

To position your drawing at the lower left corner prior to printing/plotting, click the **Left** radio button.

Creo Elements/Direct 2D Access will place an unseen box around the outermost dimension text, drawing lines, or text. This box is then placed in the lower left corner of the paper.

Specifying Linear Offsets

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 2D Access environment.

Setting a Rotation Angle

If you want to rotate the drawing output to the printer/plotter, enter the angle in the **Rotate** field in the **Position** section of the **Print** menu.

The angle of rotation is interpreted in terms of the current angular units set up for your Creo Elements/Direct 2D Access environment.

Selecting the Drawing Range for Printing/Plotting

You can print/plot the complete drawing, the current viewport, or a boxed area. The three **Range** options are:

*   **All**
*   **Viewport**
*   **Box**

All the Drawing

Click the **All** radio button in the **Range** section of the **Print** menu to print/plot the entire drawing — irrespective of the current viewport contents. This is the default option.

The Current Viewport

Click the **Viewport** radio button in the **Range** section of the **Print** menu 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.

A Boxed Area

Click the **Box** radio button in the **Range** section of the **Print** menu to print/plot a specific area of a viewport only. Creo Elements/Direct 2D Access prompts you to specify the two opposite corners of a box. The contents of the boxed area will be printed/plotted irrespective of what is displayed on the screen.

For example, suppose you are currently viewing a part, and other parts/elements exist in the same area but are not displayed. If you click **Box**, all drawing elements existing in the boxed area, both displayed and not displayed, will be plotted.

Setting the Plot Scale

This section shows how to set the scale to be used for the subsequent print/plot output.

You can set a scale factor or automatically scale the drawing as large as possible within the limits of your paper.

Entering a Scale Factor

This is the default option. The entire print/plot output will be scaled, but dimension values are not affected.

Figure 9. Operation of Print/Plot Scale on page 91 shows the operation of print/plot scale.

Figure 9. Operation of Print/Plot Scale

FULL SCALE
(Scale Factor = 1)

HALF SCALE
(Scale Factor = 0.5)
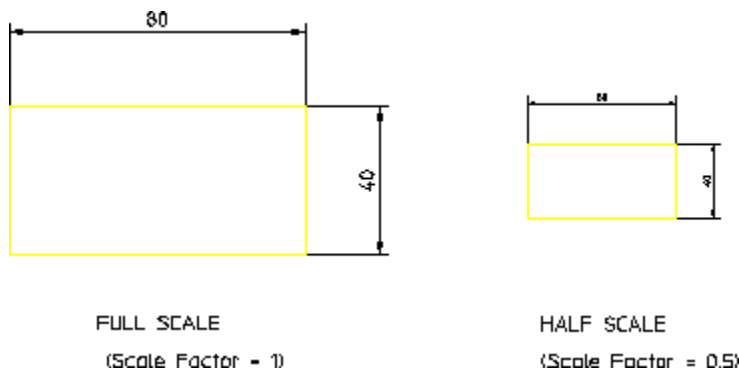
To set the print/plot scale factor:

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 2D Access 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 2D Access enables you to specify what will be included in the print/plot output:

- **Drawing**
- **Current Show**

Content: Drawing

Click the **Drawing** radio button in the **Content** section of the **Print** menu to print/plot the drawing using standard drawing conventions. Drawing lines, dimensions, text and hatching will be included in the print/plot output. Construction lines, spotlights for parts and so on 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).

Content: Current Show Attributes

Any combination of drawing lines, dimensions, hatching, text, or layers can be printed/plotted.

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 the **Current Show** radio button in the **Content** section of the **Print** menu to print/plot 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 and to set up a **Plot Transformation** map for your printer/plotter environment.

True Color

Switch on the **True Color** check box option 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 white, and specify truecolor plotting, the elements are drawn in white. Normally, when sending drawings to a pen-plotter, the white elements on the screen are converted to black

Plot Transformation

The **Plot Transformation** map enables you to map the colors and pensizes of elements in your drawing for output to a printer or plotter.

Proceed as follows to set up the **Plot Transformation** map:

1. Click **Plot Transformation** in the **Colors** section of the **Print** menu
2. In the **Plot Transformation** map, click the row you want to change. This row will be highlighted.
3. Click **Edit** to display the **Pen Format** dialog box.
4. 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 linteype 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.

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**.

Setting the Pixmap Color

When printing or plotting a drawing that includes embedded pixmaps, you can set the output of the image to be either **Black & White** or **Color**.

Retaining or Inverting Black and White Pixmaps

The **Reverse** radio button in the **B&W Pixmap** section of the **Print** menu enables you to reverse black and white pixmaps in the print/plot output.

Setting the Linetype Length

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. shows two DASH_CENTER lines drawn with different linetype lengths. The linetype length is shown arrowed.

Figure 10. Different Linetype Lengths
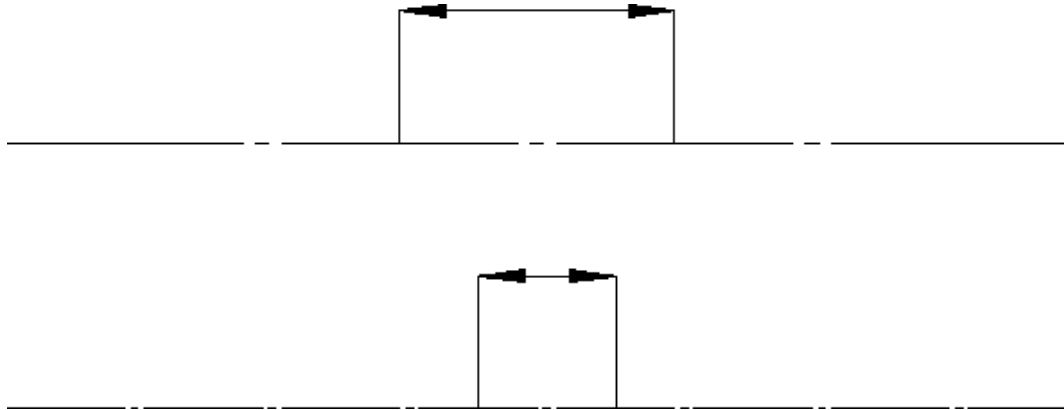
The seven broken linetypes that you can set are:

- 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 dispay 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.

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 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 2D Access Screen

To copy the entire Creo Elements/Direct 2D Access 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 screendump to the drawing area.

Writing a Creo Elements/Direct 2D Access Screendump to an Output File

Optionally, you can write a screendump of the current Creo Elements/Direct 2D Access window directly to an output file.

1. Click **File**

2. Click **Export**

3. Follow **Bitmap to File** across and choose an option:

    - BMP
    - JPEG
    - PNG
    - TIFF

4. You will be prompted through the rest of the 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.

Further Information

For further technical information about plotting or dumping, refer to the Creo Elements/Direct 2D Access help system:

Enter help plot or help bitmap_to_file at the Creo Elements/Direct 2D Access command line.

The screen will clear and Creo Elements/Direct 2D Access will display further information about plotting or dumping.

13

Customizing the Creo Elements/ Direct 2D Access User Interface

Creo Elements/Direct 2D Access provides a rich set of tools and options for user interface customization. Any customization changes in Creo Elements/Direct 2D Access are written to macros.

Displaying and Hiding Menus

To display or hide menus in the Creo Elements/Direct 2D Access 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.

Editing Toolbar Items

To change the name, type, action or other property of a menu item in the Creo Elements/Direct 2D Access 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 2D Access 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.
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 2D Access 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 105 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 2D Access, refer to the TOOLBAR_ITEM section in the Creo Elements/Direct 2D Access 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 2D Access 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 2D Access will look for your localized version (if any) of this menu item. Each toolbar and toolbar item in the Creo Elements/Direct 2D Access 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 2D Access 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 2D Access. 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 dispay **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 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 new cascaded menu:

1. Drag the **Background** menu to any position in the Creo Elements/Direct 2D Access 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 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 2D Access 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 **Image 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 2D Access when you press the key (or key combination). To execute the command, press **[Enter]**.

📝 **Note**

Note that 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 2D Access 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:

Inspection with Creo Elements/Direct 2D Access Windows User Interface

	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 2D Access. 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 an Creo Elements/Direct 2D Access 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 plain "show_popup_menu 'Arc'"#M
```

📝 **Note**

The Creo Elements/Direct 2D Access menu names are case sensitive.

Displaying Dialog Boxes Directly

The Creo Elements/Direct 2D Access 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 2D Access 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:

Managing Layouts

Via **Tools**, **Customize**, Creo Elements/Direct 2D Access 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 2D Access enables you to convert multi-column menus created with earlier Creo Elements/Direct Drafting or Creo Elements/Direct 2D Access releases.

Resetting the Layout

To return your Creo Elements/Direct 2D Access 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 2D Access layout:

```
toolbar_layout reset confirm
```

14

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.

Inspection with Creo Elements/Direct 2D Access Windows User Interface

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.

A

Tutorial

This tutorial provides you with a "step-by-step" guided tour of most of the Creo
Elements/Direct 2D Access functionality. The example drawing file used in this
tutorial is supplied with Creo Elements/Direct 2D Access.

📝 Note

In some of the following illustrations the dimensions have been switched off.
This is to make the illustrations clearer. It is not necessary for you to switch
off the dimensions.

Loading the Drawing

Proceed as follows to load the drawing file provided for this tutorial in Creo Elements/Direct 2D Access:

1. Start Creo Elements/Direct 2D Access.

2. Pull down the **File** menu and click **Load File**.

3. Select `example.mi` from the File Browser and then click **OK** to load this drawing file.

    The drawing will now load into viewport 1 as shown below.
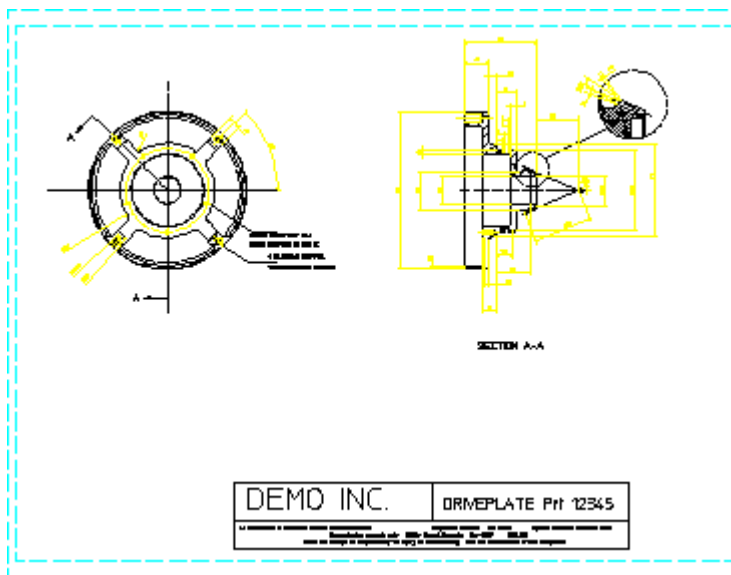
Creating Viewports

To prepare for the exercises presented in this tutorial, create three viewports similar to the ones shown in the following figure. Proceed as follows:

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

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

3. Click two diagonally opposite corner points for viewport 2. This new viewport should occupy the top left quarter of the Creo Elements/Direct 2D Access drawing area.

4. Use the same procedure to create viewport 3. This new viewport should occupy the top right quarter of the Creo Elements/Direct 2D Access drawing area.

5. Click **Fit** to fit the entire drawing into viewport 3.
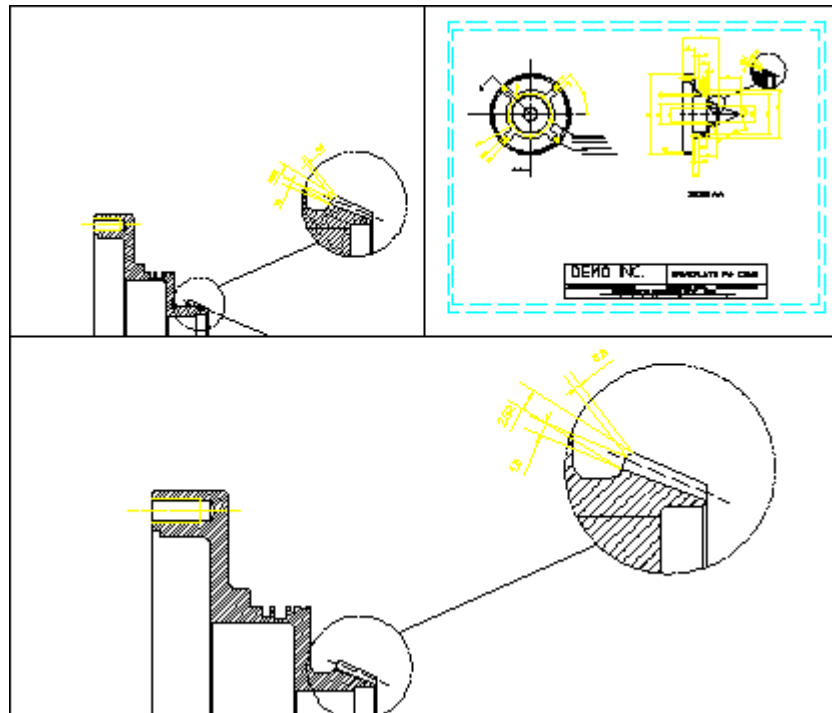
Note that the last viewport you created is the current (active) viewport.

Placing Detail Areas in the Viewports

Place a view of the right part of the drawing into viewport 1. To do this:

1. Pull down the **Setup** menu, click **Viewport** and **Current**.

2. Click on the **1** in the top right corner of viewport 1 to make this viewport the current viewport.

3. Click **New** (either in the **View** menu or on the right-hand menu bar).

4. Box the area (A) as shown below by picking the bottom left-hand corner and the top right-hand corner in viewport 1. Creo Elements/Direct 2D Access displays the boxed area in viewport 1.
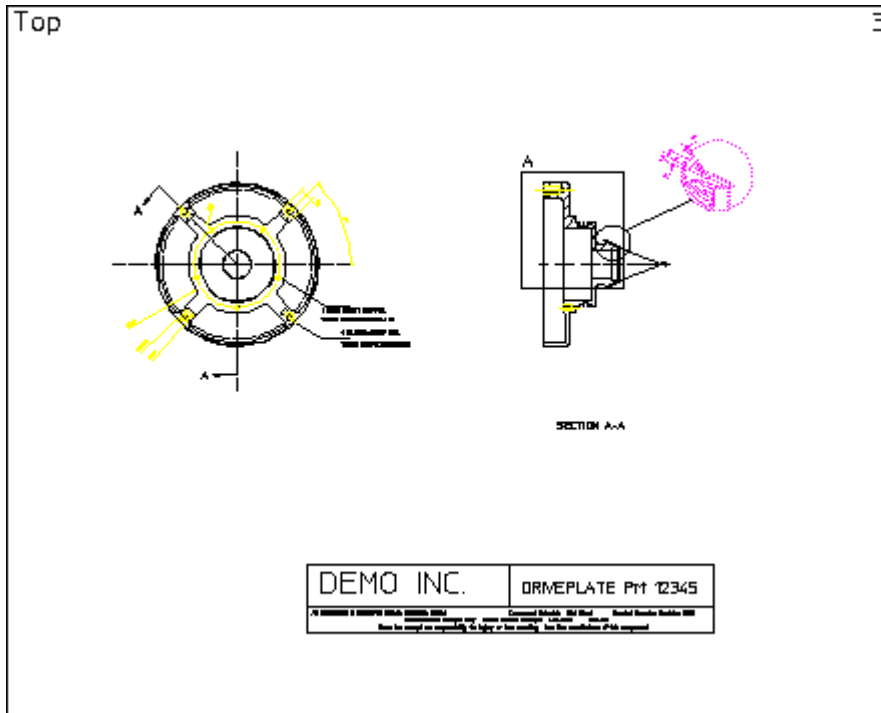
Now place a detail in each of the other two viewports:

1. Click on the **3** in the top right corner of viewport 3 to make this viewport the current viewport.

2. Click **New** and box the detail area (B) in viewport 3. Press CTRL and the left mouse button to pan. Press CTRL and the right mouse button to zoom.

   Creo Elements/Direct 2D Access displays the boxed area in viewport 3.

3. Make viewport 2 the active viewport.

4. Click **New** and box the second detail area (C) as shown below.

   Creo Elements/Direct 2D Access displays the boxed area in viewport 2.
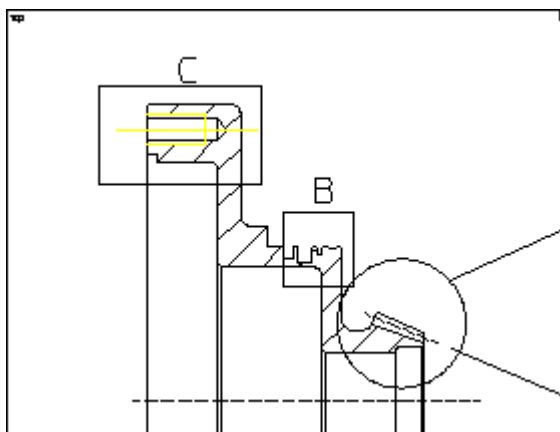
All three viewports will now contain a view as shown below.
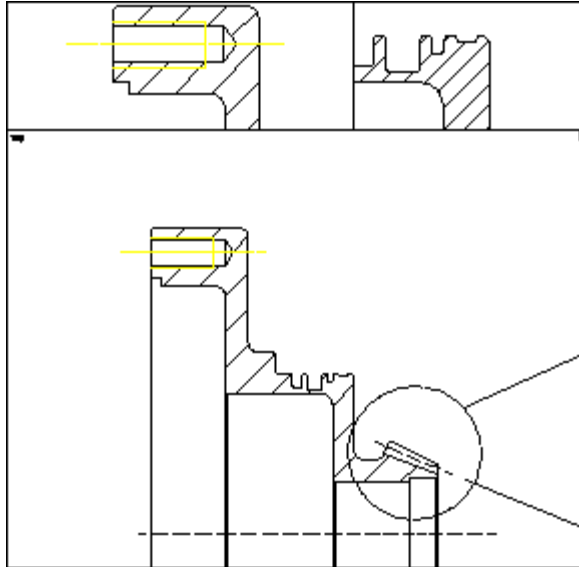
Measuring a Detail

Creo Elements/Direct 2D Access has a measuring facility which lets you accurately determine geometrical values from your drawing. The groups MEASURE and CATCH combine to form a powerful tool in the determination of geometrical values.

This part of the tutorial shows you how to measure basic drawing features.

In the drawing, a dimension has been accidently left out. The detail for this missing dimension is in the top right-hand viewport 3. To measure the length:

1. Pull down the **Tools** menu, click **Measure** and **Length**.

2. Pick line D as indicated in the following figure. Creo Elements/Direct 2D Access displays the length value in the User Prompt Line at the bottom of the screen.

3. Press ENTER to continue.
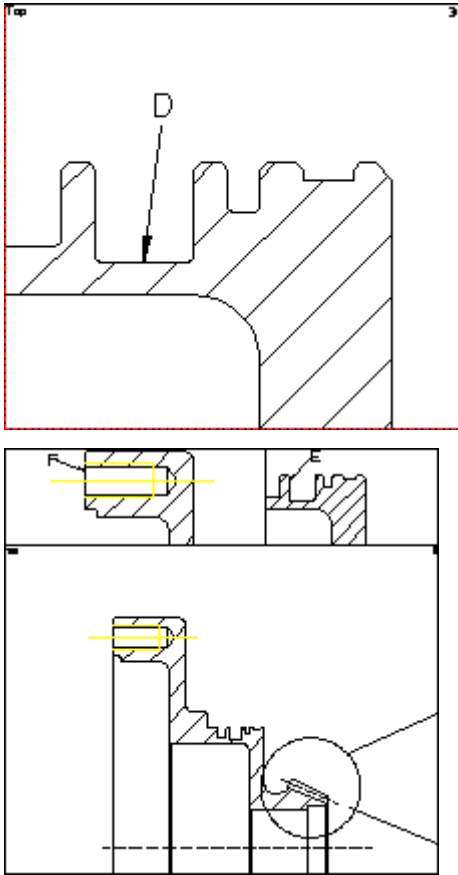
Now you need to find out the horizontal distance from the edge E to the detail edge F. This is shown in the viewports 2, and 3. To do this:

1. Pull down the **Tools** menu, click **Measure** and **Distance**.

2. Enter the qualifier **HORIZONTAL** into the User Input Line.

3. Pick the two points E and F as shown above. Creo Elements/Direct 2D Access displays the distance in the User Prompt Line.

4. Press ENTER to continue.

Using the Show options

Technical drawings often include large amounts of dimensioning, hatching, and text which can obscure some details. This part of the tutorial shows how you can selectively display what is in each of the viewports.

For this part of the tutorial you need to place a new detail area in two of the viewports. Do the following:

1. Make viewport 3 current and click **Fit**.

2. Make viewport 1 current, click **New** and box the area (G) in viewport 1.
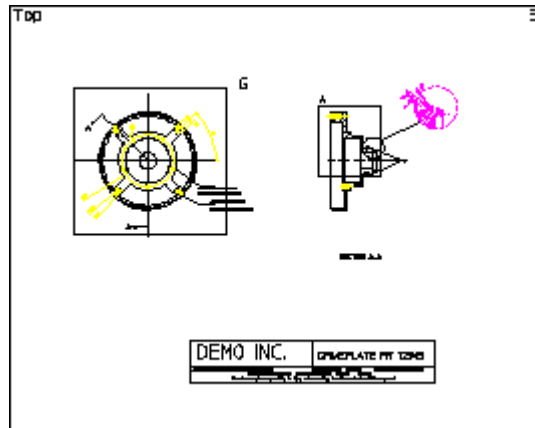
3. Make viewport 2 current, click **New** and box the area (H) in viewport 2.
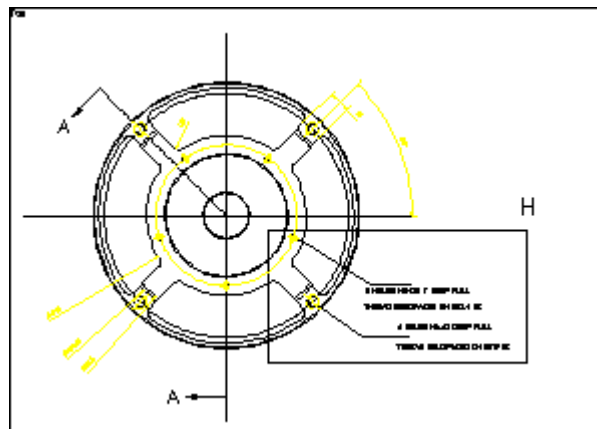
You can use the **Show** commands to display or hide entire classes of elements in a viewport. For example, you may want to view a drawing without its text. To do this:

4. Pull down the **Setup** menu and click **Show**.

5. In the **Show** dialog box, switch off **Text**.

6. Click **Redraw** in the **View** menu to see the geometry clearly.

Plotting a Drawing

You are now ready to print or plot your drawing. The print/plot options define how your drawing will be printed or plotted.

1. Pull down the **File** menu and click **Print**.

2. In the **Print** dialog box, select the appropriate printer (or plotter) and paper format.

3. Press the **Center** radio button in the **Position** section to place the drawing in the middle of the page to be printed or plotted.

4. Press the **Fit** radio button in the **Scale** section to scale the drawing to be as large as possible within the limits of the paper.

5. Optionally, save this print/plot configuration in a configuration file.

6. Click **OK** to print or plot your drawing.

Tutorial Complete

You have now finished the tutorial. If you want to, you can practice some further Creo Elements/Direct 2D Access operations using the example. You might want to try some commands and operations on your own drawings.

When you have finished with the tutorial example, click **Delete**, **All** and **Confirm** to delete the drawing from all viewports.

B

Customizing Creo Elements/Direct 2D Access

Creo Elements/Direct 2D Access can be customized to suit your individual requirements.

📋 Note

You need to be fully conversant with Creo Elements/Direct 2D Access before you start customizing Creo Elements/Direct 2D Access.

When Creo Elements/Direct 2D Access is started up, it reads and executes a number of files which define a variety of functions such as the screen menus and Creo Elements/Direct 2D Access environment. These files are executable programs called macros which you can edit and store under your own filenames to be reloaded and used as required. It is therefore possible to create your own alternative definitions for the screen menus and Creo Elements/Direct 2D Access environment. This process is known as customizing.

Customizing the Creo Elements/Direct 2D Access Environment

The Creo Elements/Direct 2D Access environment consists of such things as the the size and color of text, the plot scale, the drawing scale, and units.

All these are controlled by the settings of the environment functions, of which there are more than 70 in the system. An example of an environment function is DIM_COLOR. This can be set to produce dimensions with any of the colors available.

The current settings of the environment functions can be stored in a file. Here is a listing of part of an Creo Elements/Direct 2D Access environment file:

```
MAX_FEEDBACK 100
CONFIGURE_EDITOR '$' 1 79
UNITS 1 MM
UNITS 1 DEG
CS_REF_PT 0,0
CS_AXIS 1,0  0,1
FOLLOW OFF
GRID_FACTOR 10
CURRENT_FONT 'hp_block_v'
TEXT_FRAME OFF
TEXT_ANGLE 0
TEXT_ADJUST 1
TEXT_LINESPACE 2.2
TEXT_FILL OFF
TEXT_SIZE 1
TEXT_RATIO 1
TEXT_SLANT 0
LINE WHITE SOLID END
C_LINE RED DOTTED END
TEXT WHITE END
```

At startup, the functions are set as shown in the listing to provide a standard environment. During normal use, the settings are changed from the screen menu.

It is also possible to change the function settings by editing the Creo Elements/Direct 2D Access environment. This method is useful because it gives an overview of the status of all environment functions at once. It also gives access to those functions that are inaccessible from the screen menus.

A customized environment can be stored in a named file. This file can then be used to restore the same environment whenever it is required.

Changing the Creo Elements/Direct 2D Access Environment

To display the environment file:

1. Enter `edit_environment`.

The environment will be displayed and the function settings can be changed as required. When you have finished changing the environment, save the file and exit the text editor to implement all the changes to Creo Elements/Direct 2D Access.

The current environment can be stored for future use in a named file as follows:

1. Enter `save_environment`.

2. Enter `'env_filename'`. (The name of your own choice.)

To execute the stored file and redefine the system environment:

1. Enter INPUT `'env_filename'`.

Converting Binary Files to ASCII files

All the macros that correspond to each block are in the file `hp_macro_s`. We do not recommend that you change any of the macros in this file. However, you can copy the file under a different name, and then edit the copy.

The `hp_macro_s` file is a binary file. Before you can do any editing, you must create an ASCII copy of the file. Here's how to do it:

1. Enter the command:
   ```
   DELETE_MACRO ALL
   ```

   All macros within the workstation's memory will be deleted.

2. Load the macros within the binary file by entering the command:
   ```
   LOAD_MACRO 'sys_filename'
   ```

   `sys_filename` is the name of the system file, in this case `hp_macro_s`.

3. Store the macros as an ASCII file on disk by entering the command:
   ```
   SAVE_MACRO ALL 'newfile'
   ```

   `newfile` is a name of your choice. We recommend that this name be different from the system filename.

4. To restore all the macros that were previously deleted from the system, enter the command:
   ```
   INPUT 'startup'
   ```

You can now edit the file `newfile` that you created previously. To load the macros in this file, input the file. This will overwrite any macros of the same name that were loaded during startup.

There are two ways to ensure that your macro is loaded at startup:

- Keep the macro in a separate file. If this file is called `newfile`, then include the following line in a file called `customize`:
  ```
  INPUT 'newfile'
  ```
- Copy the altered macro to the file called `customize`. This file can be used to overwrite any of the system macros that you want to alter.

With both of these methods, you must remove the braces from the following line in the `startup` file:
```
{INPUT 'customize'}
```

Now the `customize` file will be input at startup. Any macros in this file will be loaded. Any INPUT statements will be executed, so macros in other files can also be loaded.

Writing Macros

A macro is a series of commands, executed sequentially, which are stored in a file. You can write macros to minimize the amount of time you spend on commonly used command sequences. You can use macros for activating commands that are not on the screen menu.

To create a macro enter `EDIT_FILE` and your filename, using 'single' or "double" quotation marks. If the file does not already exist, Creo Elements/Direct 2D Access creates a new one, the screen clears with the text cursor at the top of the screen. Type in your macro and save it.

The Structure of a Macro

The first line of a macro is mandatory and defines the name of the macro. The macroname does not have to be the name of the file in which you store the macro, it is possible to have any number of macros within the same file. For example, the system file `/Drafting [version]/hp_macro_s` contains several macros that Creo Elements/Direct 2D Access uses.

The first line of a macro looks like this:

- DEFINE macroname

Instructions about choosing a suitable macroname are in the `HELP` file under the keyword NAME.

Now you can define your macro. It can consist of:

- Creo Elements/Direct 2D Access commands.
- Macro operators, the help file describes these operators.
- Other macros.

An example of a macro could be:

```
DEFINE deload
```

```
DELETE ALL CONFIRM
LOAD
END_DEFINE
```

This macro deletes the current part and then prompts for the name of another drawing file that you want to load.

The last line (END_DEFINE) is also mandatory and marks the end of the macro.

Loading a Macro

You use the LOAD MACRO command to load a macro onto the system. The macro must have been stored using the STORE_MACRO command. To use this command:

1. Enter LOAD_MACRO.

2. Enter the name of the macro you want to load.

Creo Elements/Direct 2D Access loads the macro. To execute the macro, enter the macroname.

Editing a Macro

You can use the EDIT_MACRO command to:

• Edit, or list an existing macro.

• Create a macro by entering a new macroname.

To use this command:

1. Enter EDIT_MACRO.

2. Enter the name of the macro you want to edit, list, or create. If you enter a new macroname, Creo Elements/Direct 2D Access creates a new macro.

Creo Elements/Direct 2D Access displays the macro. If you have entered a new macroname then the screen will be blank as the macro is empty.

Inputting a Macro

Use the INPUT command to load a macro from an ASCII file. Once a macro is active, you can execute the macro by entering the macroname. For example, the example macro will be executed whenever deload is entered.

Executing and Running Macros

You now have a file that contains one or more macros. If you want to use any of these macros you can either enter INPUT filename or enter LOAD_MACRO. After this, you can execute the macro by typing in the appropriate macroname.

Customizing the Keyboard

When working with Creo Elements/Direct 2D Access, you may carry out some operations more frequently than others. This section describes how you can customize the keyboard to reduce the time you spend on these operations.

At the top of your keyboard there are normally eight function keys labeled [f1] to [f8] that you can customize.

For example, you may set several directories as current during an Creo Elements/Direct 2D Access session. To save time, you can customize the function keys to set each directory as current. Once you have done this, you only need to press this key, then press ENTER and the named directory will be set to current.

Any command can be implemented in this way and eight examples are given in this section.

You use the DEFINE_KEY command to customize the function keys.

The syntax for this command is described below.

DEFINE_KEY	key_number	'command_name'

The key_number defines which function key you are customizing (1 to 8).

The command_name is the command name.

Example

The following example describes how to customize the eight function keys:

1. Enter `EDIT_FILE`. The system will prompt for a filename.

2. Enter `"customize"`.

   Unless you have previously stored some customized files or macros in the file, the screen will now go blank (because the file is empty at the moment). Type in your first command line. An example could be:

   • DEFINE_KEY 1 "CURRENT_DIRECTORY 'c:/users/alan/proj1'"

   In the above line, 1 is the defined function key and `CURRENT_DIRECTORY 'c:/users/alan/proj1'` is the text to be displayed in the User Input Line.

   You can define the other function keys in exactly the same way. An example of all eight defined keys is shown below.

   ```
   DEFINE_KEY 1 "CURRENT_DIRECTORY 'c:/users/alan/proj1'"
   DEFINE_KEY 2 "CURRENT_DIRECTORY 'd:/users/larry/proj2'"
   DEFINE_KEY 3 "CURRENT_DIRECTORY 'd:/users/mary/proj3'"
   DEFINE_KEY 4 "CURRENT_DIRECTORY 'k:/users/geoff/proj4'"
   DEFINE_KEY 5 "CURRENT_DIRECTORY 'i:/users/steve/proj5'"
   ```

```
DEFINE_KEY 6 "CHANGE_VIEWPORT_SIZE"
DEFINE_KEY 7 "CHANGE_VIEWPORT_COLOR"
DEFINE_KEY 8 "EDIT_PART POINT"
```

The first five function keys set the named directories as current. Function keys 6 and 7 are used for altering the viewports. Function key 8 is used, in a drawing that contains parts, to make a picked part active.

3. After entering the command lines, press CTRL+D to store your changes in the file.

4. Input the file **customize** by entering `INPUT customize`.

The function keys will be defined and ready for your use.

Loading and Listing Text Fonts

A text font is a group of pre-defined geometrical patterns that can be placed on a drawing. Each pattern in a font is made up of straight lines joining points on a grid, and is associated with a keyboard character so that it can be easily called.

Although normally used for text (the standard system annotation and dimensioning texts are produced this way), text fonts can also produce special symbols. Because there can be many points in the grid, highly detailed symbols can be defined.

If you need special text or symbols, you can create them with your own customized text fonts and store them on disk ready for use. A good example of text font use is for producing company trademarks, logos, etc.

Text fonts are very versatile - these are some of the main features:

- Characters can be filled or unfilled
- The character width-to-height ratio can be set
- The character slant angle can be set (forwards or backwards)
- The line angle can be set (0 to 360 degrees)

The System Fonts

When the system is started up, a number of font files are loaded into memory. However, only one of them is the current font and available for immediate use.

Creo Elements/Direct Drafting's different internal and external encoding schemes were Unicode on page 109 incorporated for version 2007.

You can find out which fonts are in memory by entering `LIST_FONTS`. If no new fonts have been defined since system startup, the display will become as follows, and then you can return to the menu using ESC.

```
Fonts loaded:
```

```
    osd_default2
    osd_default3
    osd_default
    hp_def_font

 Font used:
    osd_default


 Current font:
    osd_default
```

This table gives the names of fonts that are in memory and has nothing to do with the files from which they are created. The fonts hp_Y14.5 and hp_i3098_v are loaded from files with the same names, but hp_def_font is created within the system and displays a box on the screen to represent the space that will be occupied by characters before they are drawn.

There are files available on the disk that create fonts when they are loaded. The complete list is:

osd_default	Default Creo Elements/Direct Drafting font containing hp_ i3098_v and hp_kanji2_c characters with variable character spacing
osd_default2	Default Creo Elements/Direct Drafting font containing hp_ i3098_c and hp_kanji2_c with fixed characters spacing
osd_default3	Default Creo Elements/Direct Drafting font containing hp_ d17_v and hp_kanji2_c with variable characters spacing
hp_Y14.5	Used for symbols only
hp_i3098_v	ISO 3098 font with variable character spacing
hp_i3098_c	ISO 3098 font with constant character spacing
hp_block_v	Filled font with variable character spacing
hp_block_c	Filled font with constant character spacing
hp_d17_v	DIN17 font with variable character spacing
hp_d17_c	DIN17 font with constant character spacing
hp_jasc_v	Kanji Japanese and Greek font with variable character spacing
hp_jasc_c	Kanji Japanese and Greek font with constant character spacing
hp_symbols	Special characters for dimensioning: °, ±, minutes, seconds

To increase the speed of access, all the files listed above are stored as binary files on the disk and cannot be input. They have to be loaded using:
LOAD_FONT 'bin_filename'

`bin_filename` is the name of the binary file. It is possible for a binary file to contain more than one font, in which case they will all be loaded. Each of the system files contains only one font that has the same name as the filename. The LOAD_FONT command will not change the current font.

Customizing the Startup File

📋 **Note**

To edit the Creo Elements/Direct 2D Access `startup` file, you need to be logged in as `root`. Your system administrator knows the root password for your workstation.

If you exit Creo Elements/Direct 2D Access, any customized features that you have entered, although stored in files, are not defined when you start Creo Elements/Direct 2D Access. To redefine the customization, the files containing the customization must be input again.

If you want your customization automatically defined whenever you start Creo Elements/Direct 2D Access, you should edit the Creo Elements/Direct 2D Access `startup` file. The `startup` file contains the commands that load the system files when you start Creo Elements/Direct 2D Access. To edit the `startup` file do the following:

1. Enter:

   • EDIT_FILE 'startup' and press ENTER.

   A listing of the startup file will be displayed on the screen. The listing of the file is shown below.

   ```
   INPUT 'passwords'
   LOAD_MODULE 'VIEW'
   LOAD_FONT 'hp_i3098_v'
   LOAD_FONT 'hp_Y14.5'
   LOAD_FONT 'hp_symbols'
   CURRENT_FONT 'hp_i3098_v'
   LET Menu_slot_border_distance_y 0
   LET Fill_up_menu 0
   LET Menu_position LOWER
   LOAD_MACRO 'hp_macro'
   INPUT 'defaults'
   Configure_system
   LOAD_MACRO Sys_macro_filename
   LOAD_MACRO Sys_tmacro_filename
   LOAD_MACRO Sys_menu_filename
   INPUT Sys_tmenu_filename
   {Enable_data_management}
   ```

```
{Enable_dm_classification}
{Enable_dm_bom}
Startup_menus
{INPUT 'customize'}
```

2. Remove the braces {} from around {INPUT 'customize'} so that it reads:

- INPUT 'customize'

3. Press CTRL+D to store your changes.

Check that all your files containing customized features are in the file called `customize`. If they are not, INPUT the filenames as described below.

For example, if you had two customized files: `Custm_fk`, and `Sm_menu_1` you would include the following lines at the end of the `customize` file:

- INPUT 'Custm_fk'

- INPUT 'Sm_menu_1'

During startup, the `startup` file automatically loads `customize` which in turn will load the customized files: `Custm_fk` and `Sm_menu_1`.

Creo Elements/Direct 2D Access Command and Function Names

The command or function name tells the system which operation to carry out. Each command or function has a unique name. The names for all of the Creo Elements/Direct 2D Access commands, functions, and macros are listed below. Some commands require Qualifiers, for example SCREEN. Qualifiers provide additional information about the specific operation that is to be carried out. Some commands precede others. For example, to use the LIST ITEMS command you first have to enter **CATALOG**.

Commands differ slightly from functions. Commands immediately terminate any previous command (or function). Function, however, can interrupt a command (or another function) without terminating it. Once a function has been executed, control is automatically returned to the previously unfinished command (or function) in which you were working.

General Commands/Functions

Command/Function	Option	Name
HELP		MEVIEW_HELP
DELETE	plus qualifiers	DELETE
DELETE	confirm	MEVIEW_DEL_ALL

Filing Commands/Functions

Command/Function	Option	Name
LOAD	drawing/part	MEVIEW_LOAD
CATALG	current dir.	MEVIEW_CATALOG
CATALOG	named directory	CATALOG `Name'
SET DIR		MEVIEW_SET_DIR
LOAD	plus qualifiers	LOAD
LIST ITEMS	(CATALOG first)	DETAIL_INFO
SELECT	(CATALOG first)	SELECT
SORT UP	(CATALOG first)	SORT
SORT DOWN	(CATALOG first)	REVERSE_SORT
CATALOG ITEMS	filenames	"FILE_NAME
	physical name	"PHYS_SIZE
	file description	"FILE_DESC"
	file type	"FILE_TYPE"
	file size	"FILE_SIZE"
	create date	"CREATE_DATE"
	modify date	"MODIFY_DATE"
	access date	"ACCESS_DATE"
	numerical access	"NUM_ACCESS"
CURRENT DIRECTORY		CURRENT_DIRECTORY
COPY FILE		COPY_FILE
EDIT FILE		EDIT_FILE

Setting Up Commands/Functions

Command/Function	Name
EDIT ENVIRONMENT	EDIT_ENVIRONMENT
SAVE ENVIRONMENT	SAVE_ENVIRONMENT
LOAD FONT	LOAD_FONT
LIST FONTS	LIST_FONTS
CURRENT FONT	CURRENT_FONT
LIST MACRO NAMES	LIST_MACRO_NAMES
LIST GLOBAL INFO	LIST_GLOBAL_INFO
DRAWING SCALE	DRAWING_SCALE

Hidden Line Commands/Functions

Command/Function	Option	Name
HIDD DRAW	on	HL_REDRAW_MODE On
NORM DRAW	off	HL_REDRAW_MODE Off
INQ Z-VAL		HL_INQ_Z_VALUE
GEN HIDDEN		Hl_generation
SHOW ON	on	HL_SHOW_HIDDEN On
SHOW OFF	off	HL_SHOW_HIDDEN Off

SHOW Commands/Functions

Command/Function	Option	Name
ALL	on	SHOW_ALL_ON
	off	SHOW_ALL_OFF
DIMENSION	on	DIMENSION_ON
	off	DIMENSION_OFF
HATCH	on	SHOW_HATCH_ON
	off	SHOW_HATCH_OFF
TEXT	on	SHOW_TEXT_ON
	off	SHOW_TEXT_OFF
CONSTRUCTION	on	C_GEO_ON
	off	C_GEO_OFF
GEOMETRY	on	SHOW GLOBAL GEO ALL ON
	off	SHOW GLOBAL GEO ALL OFF
LAYER	on	I_SHOW_LAYER_ON
	off	I_SHOW_LAYER_OFF
SPLIT	on	SHOW VERTEX ON
	off	SHOW VERTEX OFF
INFO	on	I_SHOW_INFO_ON
	off	I_SHOW_INFO_OFF

WINDOW Commands/Functions

Command/ Functions	Option	Name
FIT		WINDOW_FIT
PAN	norm	WINDOW_PAN
	dynam	WINDOW_PAN_DYNAMIC
NEW	norm	WINDOW_NEW
	dynam	WINDOW_NEW_DYNAMIC
LAST		WINDOW_LAST

Command/ Functions	Option	Name
REDRAW		WINDOW_REDRAW
ZOOM	norm	WINDOW_ZOOM
	dynam	WINDOW_ZOOM_DYNAMIC
LRG/SM		VIEWPORT_SWITCH
STORE		STORE_WINDOW
RECALL		WINDOW_RECALL
CENTER	norm	WINDOW_CENTER
	dynam	WINDOW_CENTER_DYNAMIC
DISPL LIST	on	WINDOW_DISPLAY_LIST On
	off	WINDOW_DISPLAY_LIST Off

VIEWPORT Commands/Functions

Command/Function	Name
CREATE	CREATE_VIEWPORT
DELETE	DELETE_VIEWPORT
DELETE ALL	DELETE_VIEWPORT ALL
REDRAW	NEW_SCREEN
CHANGE VIEWPORT SIZE	CHANGE_VIEWPORT_SIZE
CHANGE VIEWPORT COLOR	CHANGE_VIEWPORT_COLOR
CURRENT VIEWPORT	CURRENT_VIEWPORT

PARTS Commands/Functions

Command/ Function	Option	Name
EDIT	part	EDIT_PART
	top	EDIT_PART TOP
	parent	EDIT_PART PARENT
SHOW	part	SHOW_PART
	all	SHOW_PART ALL
	part boxed	SHOW_PART PART_BOX
SPOTLIGHT	on	SPOTLIGHT ON
	off	SPOTLIGHT OFF
VIEW	part	VIEW
	top	VIEW TOP
	parent	VIEW PARENT
PART LIST	Count Screen	PARTS_LIST TREE SCREEN
	Tree Screen	PARTS_LIST SCREEN

MEASURE Commands/Functions

Command/Function	Option	Name
DISTANCE	straight	MEASURE_DISTANCE
	horizontal	MEASURE_DISTANCE HORIZONTAL
	vertical	MEASURE_DISTANCE VERTICAL
LENGTH		MEASURE_LENGTH
ANGLE		MEASURE_ANGLE
RADIUS		MEASURE_RADIUS
POINT		MEASURE_COORDINATE
AREA		MEASURE_AREA
AREA PROPERTY		AREA_PROPERTY
GET PROPERTY		GET_PROPERTY

CATCH Commands/Functions

Command/Function	Option	Name
ALL	on	CATCH ALL
	off	CATCH OFF
INTERS		CATCH INTERSECTION
	permanent	CATCH PERMANENT INTERSECTION
CENTER		CATCH CENTER
	permanent	CATCH PERMANENT CENTER
VERTEX		CATCH VERTEX
	permanent	CATCH PERMANENT VERTEX
RULER/GRID		CATCH GRID
	permanent	CATCH PERMANENT GRID
ELEMENT		CATCH ELEM
	permanent	CATCH PERMANENT ELEM
RANGE		CATCH RANGE

RULER or GRID Commands/Functions

Command/Function	Option	Name
RULER	on	RULER ON
DOT GRID	on	DOT_GRID ON
LINE GRID	on	LINE_GRID ON
OFF		I_RULER_GRID_OFF
SPACING		GRID_FACTOR

Command/Function	Option	Name
ORIGIN		ORIGIN ON
PLACE	absolute	CS_SET ABSOLUTE
	relative	CS_SET RELATIVE
MOVE		CS_REF_PT
RESET		I_RULER_GRID_RESET
ISOMETRIC	top	CS_AXIS ABSOLUTE 30 1 150 1
	front	CS_AXIS ABSOLUTE 150 1 90 1
	side	CS_AXIS ABSOLUTE 30 1 90 1
CURSOR	large	CURSOR LARGE
	small	CURSOR SMALL
TURN	absolute	CS_ROTATE ABSOLUTE
	relative	CS_ROTATE RELATIVE

PLOT Commands

Command	Option	Name
POSITION	cent	Set_sys_plot_center On
	left	Set_sys_plot_center Off
SCALE	fit	Set_sys_plot_plotscale 0
	1	Plot_enter_plotscale_and_update
ROTATE ANG	0	Set_sys_plot_rot 0
	90	Set_sys_plot_rot PI/2
SIZE	A4	Set_sys_plot_format "A4"
	A3	Set_sys_plot_format "A3"
	A2	Set_sys_plot_format "A2"
	A1	Set_sys_plot_format "A1"
	A0	Set_sys_plot_format "A0"
	A	Set_sys_plot_format "A"
	B	Set_sys_plot_format "B"
	C	Set_sys_plot_format "C"
	D	Set_sys_plot_format "D"
	E	Set_sys_plot_format "E"
OUTPUT	all	Set_sys_plot_source All
	viewport	Set_sys_plot_source CURRENT WINDOW'
CONTENTS	drawing	Set_sys_plot_as_displayed (FALSE)
	as displayed	Set_sys_plot_as_displayed (TRUE)

Command	Option	Name
START PLOT		View_plot_start
SCREENDUMP	b & w	Dump_screen_bw
	color	Dump_screen_color
PLOTTERTYP		Plot_enter_plotter_type_and_update
PLOT DESTINATION		Plot_enter_filename_prefix_and_update

Creo Elements/Direct 2D Access Keywords

Keywords are reserved words within Creo Elements/Direct 2D Access used to access relevant technical information from the `help` file.

Each keyword in the list is followed by an abbreviation indicating which type of keyword it is. (Enter `help list_keywords` to display the list.)

There are four types of keywords:

- COMMANDS and FUNCTIONS (Cmd) or (Fnc): Command and function keywords such as DOT_GRID and REDRAW will start their corresponding action routine when typed by the user.

  Keep in mind that functions can interrupt a command (or another function) without terminating it. Once a function has been executed, control is automatically returned to the previously unfinished command in which you were working.

- QUALIFIERS (Qua): Qualifiers such as DEL_OLD and THREE_POINTS do not have an action routine of their own. They are used by other commands and functions to select one of the many options available to that command or function.

- PSEUDO COMMANDS (Psc): Pseudo commands such as IF and END_ LOOP are only used internally. They control the execution of macros.

- ARITHMETIC FUNCTIONS (Afn): Arithmetic functions such as SIN and SQRT are used within expressions. Their associated C function has to pick up the necessary parameters and calculate the function result.

The following keywords are available:

```
ABS                     (Afn)           ARCCOS
(Afn)
ABSOLUTE                (Qua)           ARCHIVE
(Cmd)
ADD                     (Qua)           ARCS
(Qua)
```

ADD_ATTRIBUTE	(Cmd)	ARCSIN
(Afn)		
AFFINE	(Qua)	ARCTAN
(Afn)		
ALL	(Qua)	ARC_RESOLUTION
(Fnc)		
ALLBASE	(Qua)	AREA
(Qua)		
AND	(Afn)	AREAS
(Qua)		
ANG	(Afn)	AREA_PROPERTY
(Cmd)		
ANG_BISECT	(Qua)	ARITHM_FUNCTION
(Qua)		
APPEND	(Qua)	ARROW_TYPE
(Qua)		

AS	(Qua)	CONNECT
(Cmd)		
AS_DISPLAYED	(Qua)	CONNECT_TABLE
(Fnc)		
ATTRIBUTE	(Qua)	CONTINUE
(Qua)		
ATTRIBUTES	(Qua)	CONTROLZ_IS_EOF
(Fnc)		
AUTO	(Qua)	CONVERT_TO_HP_SYMBOLS
(Fnc)		
AUTO_NEW_SCREEN	(Fnc)	COPY
(Qua)		
BACKGROUND_COLOR	(Qua)	COPY_FILE
(Fnc)		
BACKUP	(Cmd)	COS
(Afn)		
BALLOON	(Qua)	CREATE_DIRECTORY
(Fnc)		
BEEP	(Fnc)	CREATE_ELEMENT
(Cmd)		
BISECT	(Qua)	CREATE_LTAB
(Fnc)		
BITMAP_TO_FILE	(Fnc)	CREATE_STORAGE_AREA
(Cmd)		
BLACK	(Fnc)	CREATE_VIEWPORT
(Fnc)		
BLUE	(Fnc)	CS_AXIS
(Fnc)		
BORDER_COLOR	(Qua)	CS_MIRROR
(Fnc)		
BOX	(Qua)	CS_REF_PT
(Fnc)		

Inspection with Creo Elements/Direct 2D Access Windows User Interface

BREAK	(Qua)	CS_ROTATE	
(Fnc)			
CANCEL	(Cmd)	CS_SET	
(Fnc)			
CATALOG	(Fnc)	CURRENT	
(Qua)			
CATALOG_LAYOUT	(Cmd)	CURRENT_DIM_TEXTS	
(Fnc)			
CATCH	(Fnc)	CURRENT_DIRECTORY	
(Fnc)			
CENTER	(Qua)	CURRENT_FONT	
(Fnc)			
CENTER_DASH_DASH	(Fnc)	CURRENT_MENU	
(Fnc)			
CEN_BEG_END	(Qua)	CURRENT_TMENU	
(Fnc)			
CEN_RAD_ANG	(Qua)	CURRENT_VIEWPORT	
(Fnc)			
CHANGE_TABLE_SIZE	(Fnc)	CURRENT_WINDOW	
(Qua)			
CHANGE_VIEWPORT_COLOR	(Fnc)	CURSOR	
(Fnc)			
CHANGE_VIEWPORT_SIZE	(Fnc)	CURSOR_COLOR	
(Qua)			
CHAR_LAYOUT	(Cmd)	CYAN	
(Fnc)			
CHECKIN	(Cmd)	C_CIRCLES	
(Qua)			
CHECKOUT	(Cmd)	C_COLOR	
(Fnc)			
CHECK_ERROR	(Afn)	C_GEO	
(Qua)			
CHECK_WINDOW	(Fnc)	C_LINES	
(Qua)			
CHILD	(Qua)	C_LINETYPE	
(Fnc)			
CHR	(Afn)	DASHED	
(Fnc)			
CIRCLES	(Qua)	DASH_CENTER	
(Fnc)			
CLASS	(Qua)	DATE	
(Afn)			
CLASSES	(Qua)	DB_LOAD	
(Cmd)			
CLEAR	(Qua)	DB_SET_DEFAULT	
(Cmd)			
CLOSE_FILE	(Fnc)	DEFAULT	
(Qua)			
CM	(Qua)	DEFAULT_TABLE	
(Qua)			

COLOR (Fnc)	(Fnc)	DEFINE	
COLOR_LTAB (Fnc)	(Fnc)	DEFINE_CATALOG	
COLUMN (Cmd)	(Qua)	DEFINE_CLASS	
COLUMN_LAYOUT (Fnc)	(Qua)	DEFINE_FONT	
COMMAND (Fnc)	(Qua)	DEFINE_KEY	
CONFIGURE_EDITOR (Cmd)	(Fnc)	DEFINE_RELATION	
CONFIRM (Qua)	(Qua)	DEG	

DELETE (Cmd)	(Cmd)	END_PART	
DELETE_CLASS (Psc)	(Cmd)	END_WHILE	
DELETE_LTAB (Fnc)	(Fnc)	ENTER	
DELETE_LTAB_ROW (Fnc)	(Fnc)	ERROR_LOG	
DELETE_MACRO (Afn)	(Fnc)	ERROR_STR	
DELETE_MENU (Cmd)	(Fnc)	EXIT	
DELETE_TABLE (Psc)	(Cmd)	EXIT_IF	
DELETE_TMENU (Afn)	(Fnc)	EXOR	
DELETE_VIEWPORT (Afn)	(Fnc)	EXP	
DEL_OLD (Qua)	(Qua)	FACES	
DEL_VERSIONS (Afn)	(Qua)	FALSE	
DENSITY (Fnc)	(Qua)	FEEDBACK_INDICATOR	
DETAIL_INFO (Qua)	(Qua)	FEET	
DIAMETER (Qua)	(Qua)	FILE	
DIMENSIONS (Qua)	(Qua)	FILES	
DISCONNECT (Qua)	(Cmd)	FILES_TO_ELEMENTS	
DISPLAY (Qua)	(Fnc)	FILLETS	

DISPLAY_LIST	(Fnc)	FIRST	(Qua)
DISPLAY_NO_WAIT	(Fnc)	FIT	(Qua)
DIST_ANG	(Qua)	FOLLOW	(Fnc)
DIST_DIST	(Qua)	FONT	(Qua)
DIV	(Afn)	FORCE	(Qua)
DMS_VERSION	(Fnc)	FORMAT	(Qua)
DOTTED	(Fnc)	FRACT	(Afn)
DOT_CENTER	(Fnc)	VIEW_WIDTH	(Qua)
DOT_GRID	(Fnc)	FUNCTION	(Qua)
DOT_TYPE	(Qua)	GEO	(Qua)
DOWN	(Qua)	GET_ELEM_INFO	(Fnc)
DRAWING_SCALE	(Cmd)	GET_PROPERTIES	(Fnc)
DUMP	(Qua)	GLOBAL	(Qua)
DUMP_SCREEN	(Fnc)	GRAPHIC	(Qua)
DUMP_SCREEN_DEFAULTS	(Fnc)	GRD	(Qua)
DYNAMIC	(Qua)	GREEN	(Fnc)
ECHO	(Fnc)	GRID	(Qua)
EDIT_ENVIRONMENT	(Fnc)	GRID_FACTOR	(Fnc)
EDIT_FILE	(Fnc)	GROUP	(Qua)
EDIT_MACRO	(Fnc)	HATCHING	(Qua)
EDIT_PART	(Cmd)	HEIGHT	(Qua)
EDIT_PORT	(Fnc)	HELP	(Fnc)
ELEM	(Qua)	HELP_PORT	(Fnc)
ELEMENTS	(Qua)	HIGHLIGHT_LTAB	(Fnc)
ELEMENTS_TO_FILES	(Qua)	HL_GENERATE_HIDDEN	(Cmd)

ELSE (Fnc)	(Psc)	HL_INQ_FACE_COLOR
ELSE_IF (Fnc)	(Psc)	HL_INQ_Z_VALUE
END (Fnc)	(Cmd)	HL_REDRAW_MODE
END_DEFINE (Fnc)	(Psc)	HL_SET_COLOR
END_IF (Fnc)	(Psc)	HL_SET_LINETYPE
END_LOOP (Fnc)	(Psc)	HL_VISUALIZE
HORIZONTAL (Fnc)	(Qua)	LIST_MACRO_NAMES
HPGL (Fnc)	(Qua)	LIST_MENUS
HPGL2 (Qua)	(Qua)	LITERAL
HP_SQL (Afn)	(Qua)	LN
HSL_COLOR (Cmd)	(Fnc)	LOAD
IF (Cmd)	(Psc)	LOAD_FONT
ILLEGAL (Fnc)	(Qua)	LOAD_MACRO
IMAGE_FIT (Cmd)	(Qua)	LOAD_MODULE
INCHES (Psc)	(Qua)	LOCAL
INDEX (Fnc)	(Qua)	LONG_DASHED
INFOS (Psc)	(Qua)	LOOP
INPUT (Qua)	(Fnc)	LOWER
INQ (Afn)	(Afn)	LTAB_COLUMNS
INQ_DB (Afn)	(Fnc)	LTAB_ROWS
INQ_ELEM (Afn)	(Fnc)	LTAB_TITLES
INQ_ENV (Afn)	(Fnc)	LWC
INQ_SELECTED_ELEM (Qua)	(Fnc)	MACRO
INSERT_LTAB_ROW (Fnc)	(Fnc)	MACRO_STAT

INT (Fnc)	(Afn)	MAGENTA	
INTERSECTION (Cmd)	(Qua)	MAKE_QUERY_PAGE	
KEEP (Qua)	(Qua)	MARK	
KEEP_POINTS (Afn)	(Qua)	MATCH	
KEYWORD_STAT (Qua)	(Fnc)	MAXIMUM	
KM (Fnc)	(Qua)	MAX_FEEDBACK	
KNOB_BOX (Qua)	(Qua)	MAX_RADIUS	
KNOB_BOX_FACTOR (Fnc)	(Fnc)	MEASURE_ANGLE	
LARGE (Fnc)	(Qua)	MEASURE_AREA	
LAST (Fnc)	(Qua)	MEASURE_COORDINATE	
LAST_FEEDBACK (Fnc)	(Qua)	MEASURE_DISTANCE	
LAST_PROMPT (Fnc)	(Qua)	MEASURE_LENGTH	
LAST_QUERY_RESULTS (Fnc)	(Fnc)	MEASURE_RADIUS	
LAST_WINDOW (Fnc)	(Fnc)	MEMORY_STAT	
LAST_WINDOWS (Fnc)	(Qua)	MENU	
LEADER_ARROW (Fnc)	(Fnc)	MENU_BUFFER	
LEADER_LINE (Fnc)	(Cmd)	MENU_LAYOUT	
LEADER_LINES (Qua)	(Qua)	METERS	
LEFT (Qua)	(Qua)	MI	
LEN (Qua)	(Afn)	MILES	
LET (Qua)	(Fnc)	MILS	
LG (Qua)	(Afn)	MINIMUM	
LINEPATTERN (Qua)	(Fnc)	MIN_LENGTH	
LINES (Qua)	(Qua)	MIN_RADIUS	
LINETYPE (Afn)	(Fnc)	MIRR	

LINE_GRID (Qua)	(Fnc)	MIRROR
LINKS (Qua)	(Qua)	MIX
LIST_FONTS (Qua)	(Fnc)	MM
LIST_GLOBAL_INFO (Afn)	(Fnc)	MOD
LIST_KEYWORDS (Cmd)	(Fnc)	MODIFY_RELATIONSHIP

MODIFY_USER (Fnc)	(Cmd)	PICK_VP_PNT
MOVE (Cmd)	(Qua)	PLOT
MOVE_ELEMENT (Fnc)	(Cmd)	PLOTTER_TYPE
MOVE_TABLE (Fnc)	(Fnc)	PLOT_CENTER
MULTIPLE (Fnc)	(Qua)	PLOT_DESTINATION
NETUNAM (Fnc)	(Fnc)	PLOT_FORMAT
NEW (Fnc)	(Qua)	PLOT_LINETYPE_LENGTH
NEW_SCREEN (Fnc)	(Fnc)	PLOT_PEN_TABLE
NEXT (Fnc)	(Qua)	PLOT_SCALE
NONE (Fnc)	(Qua)	PLOT_TRANSFORMATION
NORMAL (Fnc)	(Qua)	PLOT_VIEWPORT
NOT (Qua)	(Afn)	PNT
NO_DELAY (Qua)	(Qua)	PNT3
NO_VIEWPORT_RANGE (Qua)	(Qua)	PNT_MM
NULL (Qua)	(Qua)	PNT_PIXEL
NUM (Afn)	(Afn)	PNT_RA
NUMBER (Afn)	(Qua)	PNT_XY
OF (Afn)	(Qua)	PNT_XYZ
OFF (Qua)	(Qua)	POINTS

OLD_SQL_QUERY	(Fnc)	POLYGON	(Qua)
ON	(Qua)	POP_DOWN_LTAB	(Fnc)
ON_ERROR	(Fnc)	POP_UP_LTAB	(Fnc)
OPEN_INFILE	(Fnc)	PORT	(Qua)
OPEN_OUTFILE	(Fnc)	POS	(Afn)
OR	(Afn)	PREVIOUS	(Qua)
ORACLE	(Qua)	PRINT_TABLE	(Fnc)
ORIGIN	(Fnc)	PROMPT	(Qua)
OUTPUT_HP15	(Fnc)	PROMPT_LINE	(Qua)
OUTPUT_HP16	(Fnc)	PROMPT_LIST	(Fnc)
OWNER	(Qua)	PSEUDO_COMMAND	(Qua)
PAN	(Qua)	PT_ANG	(Qua)
PAN_SEL	(Qua)	PT_ANG_DIST	(Qua)
PARALLEL	(Fnc)	PURGE_FILE	(Qua)
PARAMETER	(Psc)	QUALIFIER	(Qua)
PARENT	(Qua)	QUERY	(Fnc)
PART	(Qua)	QUERY_ADD_CLASS	(Fnc)
PARTS	(Qua)	QUERY_ADD_COLUMN	(Fnc)
PARTS_LIST	(Fnc)	QUERY_ADD_SELECT	(Fnc)
PART_BOX	(Qua)	QUERY_ADD_SORT	(Fnc)
PASSWORD	(Fnc)	QUERY_AREAS	(Fnc)
PERF_MON_OFF	(Fnc)	QUERY_CLASSES	(Fnc)
PERF_MON_ON	(Fnc)	QUERY_COPY	(Fnc)
PERMANENT	(Qua)	QUERY_CREATE	(Fnc)
PERMISSIONS	(Qua)	QUERY_CREATE_CLASSIFICATION	(Fnc)

PERPENDICULAR (Cmd)	(Qua)	QUERY_DELETE
PER_RAD_ANG (Fnc)	(Qua)	QUERY_DELETE_CLASS
PHANTOM (Fnc)	(Fnc)	QUERY_DELETE_COLUMN
PI (Fnc)	(Afn)	QUERY_DELETE_SELECT

QUERY_DELETE_SORT (Psc)	(Fnc)	REPEAT
QUERY_DESTINATION (Qua)	(Fnc)	RESERVE
QUERY_DIRECTION (Qua)	(Fnc)	RESET
QUERY_EXECUTE (Cmd)	(Fnc)	RESTORE
QUERY_FILES (Fnc)	(Fnc)	RETAIN_RASTER
QUERY_FILE_NAME (Cmd)	(Fnc)	RETAIN_RELATIONSHIP
QUERY_LEVELS (Qua)	(Fnc)	REVERSE
QUERY_LIST (Qua)	(Fnc)	REVERSE_SORT
QUERY_MODIFY_CLASS (Qua)	(Fnc)	REVISION
QUERY_MODIFY_COLUMN (Qua)	(Fnc)	REVISIONING
QUERY_MODIFY_SELECT (Fnc)	(Fnc)	RGB_COLOR
QUERY_MODIFY_SORT (Qua)	(Fnc)	RIGHT
QUERY_REFRESH (Afn)	(Fnc)	RND
QUERY_RENAME (Afn)	(Fnc)	ROT
QUERY_SAVE (Qua)	(Fnc)	ROTATE
QUERY_SELECT_TABLE_NAME (Afn)	(Fnc)	ROUND
QUERY_SET_SELECT (Qua)	(Fnc)	ROW
QUERY_SORT_TABLE_NAME (Qua)	(Fnc)	ROWS
QUERY_START_ELEMENT (Afn)	(Fnc)	RPT
QUERY_TABLE_NAME (Qua)	(Fnc)	RUBBER_ARC_BEG_END

QUERY_TYPE	(Fnc)	RUBBER_ARC_CEN_BEG	(Qua)
QUERY_USER_TABLE_NAME	(Fnc)	RUBBER_ARC_CEN_END	(Qua)
RAD	(Qua)	RUBBER_BOX	(Qua)
RANGE	(Qua)	RUBBER_CIRCLE_2_PTS	(Qua)
READ	(Fnc)	RUBBER_CIRCLE_CEN	(Qua)
READ_FILE	(Fnc)	RUBBER_LINE	(Qua)
READ_LTAB	(Afn)	RUBBER_LINE_ANG	(Qua)
READ_ONLY	(Qua)	RUBBER_LINE_HORIZONTAL	(Qua)
RECALL_BUFFER	(Fnc)	RUBBER_LINE_VERTICAL	(Qua)
RECALL_WINDOW	(Fnc)	RULER	
RECALL_WINDOWS	(Qua)	RUN	
RECTANGLE	(Qua)	SAME	
RECURSIVE	(Fnc)	SAVE_ENVIRONMENT	
RECURSIVE_TO	(Qua)	SAVE_LTAB	
RED	(Fnc)	SAVE_MACRO	
REDRAW	(Fnc)	SAVE_MENU	
REF_PT	(Qua)	SAVE_TABLE	
RELATE	(Cmd)	SAVE_TMENU	
RELATED_CLASS	(Qua)	SAVE_VIEWPORT	
RELATED_CLASSES	(Qua)	SCALE	
RELATIVE	(Qua)	SCREEN	
RELEASABLE	(Qua)	SCREEN_TRANSFORMATION	
RELEASE	(Cmd)	SCROLL_BAR	
REMOVE	(Cmd)	SCROLL_LTAB	
REMOVE_QUERY_PAGE	(Cmd)	SEARCH	

REMOVE_USER	(Cmd)	SECURE_LTAB	
(Fnc)			
RENAME_ELEMENT	(Cmd)	SECURE_MACRO	
(Fnc)			
RENOVATE	(Fnc)	SECURE_TABLE	
(Fnc)			

SELECT	(Qua)	TAN_PT	
(Qua)			
SELECT_FROM_LTAB	(Fnc)	TAN_PT_PT	
(Qua)			
SELECT_KNOB	(Qua)	TB_GET_DRAW_VALUES	
(Fnc)			
SELECT_PORT	(Qua)	TB_SETUP_TABLE	
(Fnc)			
SEL_OLD	(Qua)	TECHO	
(Fnc)			
SGN	(Afn)	TEXT	
(Cmd)			
SHARED_PART	(Qua)	TEXTS	
(Qua)			
SHOW	(Fnc)	TEXT_ADJUST	
(Fnc)			
SHOW_PART	(Fnc)	TEXT_ANGLE	
(Fnc)			
SHOW_TABLE	(Fnc)	TEXT_ATTR	
(Qua)			
SHOW_TABLE_PAGE	(Fnc)	TEXT_COLOR	
(Qua)			
SIMILAR	(Qua)	TEXT_FILL	
(Fnc)			
SIN	(Afn)	TEXT_FORMAT	
(Qua)			
SLASH_TYPE	(Qua)	TEXT_FRAME	
(Fnc)			
SMALL	(Qua)	TEXT_LINESPACE	
(Fnc)			
SMOOTH	(Qua)	TEXT_RATIO	
(Fnc)			
SNID	(Afn)	TEXT_SIZE	
(Fnc)			
SOLID	(Fnc)	TEXT_SLANT	
(Fnc)			
SORT	(Qua)	THREE_PTS	
(Qua)			
SORT_LTAB	(Fnc)	TIME	
(Afn)			
SPLINES	(Qua)	TINPUT	
(Fnc)			

Inspection with Creo Elements/Direct 2D Access Windows User Interface

SPOTLIGHT	(Fnc)	TITLE	(Qua)
SQL_QUERY	(Fnc)	TITLE_LAYOUT	(Qua)
SQR	(Afn)	TMENU	(Fnc)
SQRT	(Afn)	TONE	(Fnc)
STATLINE_RESET	(Fnc)	TOP	(Qua)
STORE_MACRO	(Fnc)	TRACE	(Fnc)
STORE_WINDOW	(Fnc)	TRACKING	(Qua)
STORE_WINDOWS	(Qua)	TRAP_ERROR	(Fnc)
STR	(Afn)	TREE	(Qua)
STRING	(Qua)	TRIANGLE_TYPE	(Qua)
STRUCTURE	(Fnc)	TRIM	(Afn)
STRUCTURES	(Qua)	TRUE	(Afn)
SUBPART	(Qua)	TRUNC	(Afn)
SUBSTR	(Afn)	TWO_PNT	(Qua)
SUBTRACT	(Qua)	TWO_PTS	(Qua)
SYMBOL	(Qua)	TYPE	(Afn)
TABLE	(Qua)	UINCHES	(Qua)
TABLET_RESET	(Fnc)	UM	(Qua)
TABLE_COLUMN	(Fnc)	UNDO	(Qua)
TABLE_LAYOUT	(Cmd)	UNITS	(Fnc)
TABLE_SCROLL_STEP	(Fnc)	UNRELEASE	(Cmd)
TABLE_TITLE	(Fnc)	UNRESERVE	(Cmd)
TAN	(Afn)	UNTIL	(Psc)
TAN2	(Qua)	UP	(Qua)
TAN2_PT	(Qua)	UPC	(Afn)

TAN3 (Cmd)	(Qua)	UPDATE_ELEMENT	
TAN_CENTER (Fnc)	(Qua)	UPDATE_SCREEN	
UPDATE_STATISTICS (Psc)	(Cmd)	WHILE	
UPPER (Fnc)	(Qua)	WHITE	
USER_TABLE (Qua)	(Qua)	WIDTH	
VAL (Fnc)	(Afn)	WINDOW	
VERSION (Fnc)	(Fnc)	WRITE_FILE	
VERSIONING (Fnc)	(Qua)	WRITE_LTAB	
VERTEX (Afn)	(Qua)	X_OF	
VERTEX_3D (Qua)	(Qua)	YARDS	
VERTICAL (Fnc)	(Qua)	YELLOW	
VIEW (Afn)	(Fnc)	Y_OF	
WAIT (Qua)	(Fnc)	ZOOM	
WARNING (Afn)	(Qua)	Z_OF	
WHERE_USED (Qua)	(Fnc)	Z_POSITION	

Creating Menus

To create a custom menu:

1. Pull down the **Tools** menu, click **Customize** and **Customize Layout**.

2. In the **Customize Layout** dialog box, click the **Toolbar** tab.

3. Click **New**.

4. Enter a name for the new menu in the **New Toolbar** dialog box.

5. Click **OK** to create a new (initially empty) container.

6. In the **Customize Layout** dialog box, click the **Commands** tab.

7. In the **Categories** pane, select the existing container from which you want to copy commands.

8. Select the required commands from the **Buttons** pane.

9. Drag these commands to the newly created container.

10. When the container holds all of the required commands, you can drag and drop it to any location in the Creo Elements/Direct 2D Access user interface.

Adding Icons to the Toolbar

To add an existing 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 **Customize Layout** dialog box, click the **Macro Commands** tab.

3. Click **Add New**. In the **Add New Macro Command** dialog box:

   • Enter a new name for the new macro.

   • Enter a macro command (such as: `delete all confirm`) into the **Action Text** field.

   • Optionally, enter a tool tip (such as: `Delete everyting on the screen`).

   • Click **Browse** to select an icon.

4. Click **OK** to add the new macro command plus its associated icon to your list in **Customize Layout**.

5. In **Customize Layout**, click the **Commands** tab.

6. Scroll down to the bottom of the **Categories** list until you see User Commands.

7. Click on **User Commands**. In the **Buttons** pane, select a button and drag it to any point in the Creo Elements/Direct Drafting user interface (such as the toolbar).

Creating Multi-Column Menus

Creo Elements/Direct 2D Access enables you to define and use your own multiple-column menus.

Creating a Multi-Column Menu Definition File

To define a multiple-column menu, you create a simple text file which contains the required definitions. This text file must contain the following sections:

menu	Contains global information like menu name, position etc.
rowX	Defines which buttons (with their position) are contained in row X (where X is a natural number). A separate section is required for each row.
button_ btnname	Contains information about the button named btnname: label, width and associated action. Each button requires its own section.

📝 **Note**

Section names are enclosed in square brackets.

Be sure to specify the filename extension `mnu` when saving your multi-column menu definition file.

General Syntax

The general syntax of multi-column menu definition file is presented below.

```
[menu]
menu_name=MMM
menu_shown_rows=ROWS
menu_shown_columns=COLS
menu_positionx=POSX
menu_positiony=POSY
[rowX]
row_buttons=PP1;btn1#PP2;btn2...
[button_btn1]
button_label=LBL
button_width=WIDTH
button_actiontext=ACT
[button_btn2]
button_label=LBL
button_width=WIDTH
button_actiontext=ACT
[rowY]
...
```

where:

MMM	stands for name of the menu. This name string will also be displayed as the banner title of the menu window.
ROWS	specifies the menu's height (in lines/rows).
COLS	specifies the menu's width in characters.

POSX	specifies the horizontal position of the menu in pixels.
POSY	specifies the vertical position of the menu in pixels.
PP1;btn1#PP2;btn2	There are two kinds of separators: # for buttons and ; for the data (position and name) of a single button. In this example, PP1 stands for the position of the first button (btn1), and PP2 stands for the position of the second button (btn2). The units are columns. One column is approximately equal to the width of one character.
button_label	This string will be shown on the button. The user will see only this text.
button_width	Width of the button in columns. 1 unit is equal approximately to the width of 1 character.
button_actiontext	This is the action text which will be executed when the button is pressed. If the user need to open a submenu with pressing this button then the action text should begin with '##' and afterwards the name of the new multi column menu file. The application will look for this file in the directory from which is loaded the current menu. The action text can have one more value — *X — which means that when the user presses this button, the current menu will be closed.

📝 **Note**

Vertical and/or horizontal scrollbars are handled automatically.

Loading a Multi-Column Menu Definition File

To display and use a multi-column menu in Creo Elements/Direct 2D Access, load its underlying menu definition file (extension mnu) in one of two ways:

• Enter LOAD_MULTICOL_MENU in the User Input Line in Then specify the filename.

Alternatively:

• In Creo Elements/Direct 2D Access click **Tools ▶ Customize ▶ Customize Layout**. Then click on the **Multi Column Menus** tab, click **Add** and select the menu definition file.

Index

E

EDIT MACRO command, 126
EDIT_PART
    Parent option, 49
    Top option, 48
Editing Toolbar Items, 100
elements
    adding to your selection, 12
    boxing, 13
    deleting, 11
    enclosing within a polyline, 13
    preselecting, 16
    removing them from your selection, 13
    selecting, 12
    selecting all, 14
enclosing
    customizing the, 123
    elements, 13
environment
    changing the, 124
exiting Creo Elements/Direct 2D
  Access, 18

F

face
    measuring the area, 41
faces, what are they?, 55
features
    showing all, 35
file browser, 114
File Browser
    opening a drawing, 20
files
    how to find, 19
filing
    introduction, 19
fitting, 29
Follow on/off
    using, 44
font

importing, 19
fonts
    system, 128
    text, 128
function names, 131

G

generating
    hidden lines, 57
Geo command, 15
global
    selection, 11
graphics performance
    enhancing, 36
grids
    displaying, 43

H

hidden lines
    displaying as dashed lines, 57
    generating, 57
    introduction, 55
    plotting, 58
Hidden Mode
    On or Off, 58
hiding
    ruler/grid, 45
horizontal
    measuring distances, 40
HP-GL file
    importing, 27

I

icon
    adding to toolbar, 105, 151
    creating, 105
icons
    in Part Browser, 51
importing
    font, 26

the ruler/grid, 44
rotation
    print/plot output, 90
ruler
    displaying, 43
    moving, 44
ruler/grid
    drawing an isometric, 45
    hiding, 45
    placing, 45
    rotating, 44
    setting the spacing, 45
    using, 43

S

scale
    factor for plot, 91
    setting, 91
scaling
    OLE object, 75
screendunp, 97
Select command, 12
Select mechanism, 11
selecting
    what is displayed, 34
selection
    a vertex, 14
    boxing elements, 13
    confirming, 17
    construction geometry, 15
    dimensions, 16
    drawing elements, 12
    drawing geometry, 15
    elements within a polyline, 13
    general, 11
    global, 11
    infos, 14
    layers, 15
    linesize, 15
    parts, 15
    pensize, 15

reference lines, 15
removing elements, 13
subtracting elements, 13
subtree, 15
text, 15
URLs, 16
Set Color command (revision
  checking), 66
setting the units of measurement, 42
Sheet Browser, 81
Show
    functions with layers, 71
SHOW
    All option, 50
    Boxed option, 50
    function, 34
    Part option, 49
Show Difference command, 67
Show Hidden, 58
Show, using, 118
showing
    all features, 35
    dimensions, 36
    points, 35
sorted catalogs, 24
SPOTLIGHT ON/OFF command, 53
spotlighting, 53
starting Creo Elements/Direct 2D
  Access, 8
startup file
    customizing, 130
    editing, 130
storing
    a window, 30
structure
    macros, 125
subtree
    selecting, 15
switching
    URLs on and off, 71
system fonts, 128

T

tangent
  constructing, 62
text
  selecting, 15
text fonts
  loading and listing, 128
toolbar
  adding a button, 151
  adding items, 101
  creating, 101
  loading, 108
  saving, 108
toolbar items
  editing, 100
truecolor plotting, 93
truecolor printing, 93
TrueType font
  importing, 26
  Resolution, 26
tutorial
  using Creo Elements/Direct 2D
    Access, 113

U

Undo, 17
Unicode, 109
units of measurement, 42
URL
  color, 71
URLs
  selecting, 16
user interface
  classic, 10
  customizing, 99
using
  Show commands, 118
  viewports, 115

V

values
  Z, 55
vertex
  selecting a, 14
  show, 35
vertical
  measuring distances, 40
vertices
  show, 35
View command
  Parent option, 50
  Top option, 50
View commands
  parts, 50
VIEW commands, 50
viewport
  changing size, 33
  creating, 32
  creating, example, 114
  deleting, 34
  making it current, 32
  maximizing, 33
  moving, 33
  placing a detail, 115
  redrawing, 33
  settings, 34

W

what are macros?, 125
wildcards, 25
window
  creating, 29
  recalling, 30
  restoring last view, 29
  storing, 30
  using normal functions, 29
  zooming, 29
windows
  multiple Creo Elements/Direct 2D
    Access instances, 22

working
   basic methods, 11
writing
   macros, 125

Z

z-value
   checking, 56
   checking maximum, 57
   checking minimum, 57
   of a shared part, 57
   what is it?, 55
zooming
   in or out, 29