



**Inspection with Creo
Elements/Direct 2D Access
Classic User Interface**
Creo Elements/Direct 2D Access 20.2.4.0

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Preface

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

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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 66](#).

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

Starting Creo Elements/Direct 2D Access

To start Creo Elements/Direct 2D Access:

On an MS-Windows PC (with Creo Elements/Direct 2D Access set to Classic User Interface) Click: **Start ► Programs ► PTC ► Creo Elements Direct 2D Access [version]**.

Note

Be sure that the License Server is running before you start Creo Elements/Direct 2D Access. See your Installing and Configuring Creo Elements/Direct Drafting manual for instructions.

Optionally, you can run multiple Creo Elements/Direct Drafting instances (in separate Drafting windows) at the same time. Refer to [Creating a New Instance of Creo Elements/Direct 2D Access on page 16](#) 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	<code>change_locale 'en'</code>
French	<code>change_locale 'fr'</code>
German	<code>change_locale 'de'</code>
Italian	<code>change_locale 'it'</code>

Language	Command
Spanish	<code>change_locale 'es'</code>
Japanese	<code>change_locale 'ja'</code>

Alternatively, specify this command above the `Startup_menus` string in the `startup.m` file 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 Between Flat and Raided Menu Buttons

The `MEPELOOK` variable in the `me10.ini` file (located in the Creo Elements/Direct Drafting installation directory) enables you to select either the "flat" look or the "raised menu buttons" look of the Creo Elements/Direct Drafting Classic user interface.

Undo and Redo

In Creo Elements/Direct 2D Access, you can undo mistakes with **UNDO** / **REDO**.

Click **UNDO** to undo an operation (before you click **CONFIRM**).

Click **REDO** to redo an operation (after you have clicked **UNDO**).

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

You can set a limit on the number of steps to keep in the **UNDO/REDO** history in the System Setting dialog.

1. In the main menu, click **Setup ▶ Edit Environment ▶ System**.
2. You can select:
 - **-1**: no limit.
 - **0**: disable the Undo/Redo option.

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

Limitations:

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

Undo and Redo with state history

Each time you enter a command and change your drawing, Creo Elements/Direct Drafting adds a new state to the undo/redo history. Creo Elements/Direct Drafting marks the new state with a unique ID. With this information, you can:

- Use `UR_GET_CURRENT_ID` to see the ID of the current undo/redo state.
- Use `UR_ID_EXISTS` to see if a specified ID exists in the undo/redo history.
- Use `UR_GET_FIRST_ID` to see the ID of the first state in the undo/redo history.
- Use `UR_GET_LAST_ID` to see the ID of the last state in the undo/redo history.
- Use `UR_MOVE` to undo/redo the drawing to a specified state.

Exiting Creo Elements/Direct 2D Access and Logout

To exit Creo Elements/Direct 2D Access:

1. Click **EXIT**.

Creo Elements/Direct 2D Access will prompt for you to confirm your decision. The **CONFIRM** slot will now appear in the **EXIT** slot.

2. Click **CONFIRM**.

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:

Manuals\htm\english\viewer\appb.html

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Finding, Loading, and Handling Files

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

Setting the Current Directory

Setting a directory as current lets you load the files in that directory without having to enter the full path name for each command or filename.

To set a directory as current:

1. Pick **SET DIR** from the screen menu.
2. Enter the full path name to the directory containing the file.

Loading a Drawing

To load a drawing in Creo Elements/Direct 2D Access, pick **LOAD**, and specify the name of the drawing file (in single quotation marks). Optionally, repeat this procedure to load additional files in the same Creo Elements/Direct 2D Access session.

If this file is not located in the current directory, enter its full pathname. Alternatively, you can use the File Browser. See [Using the File Browser on page 13](#) for instructions.

If you want to clear your Creo Elements/Direct 2D Access workspace before you load a drawing file, pick **DELETE ALL** and **CONFIRM**.

The following file formats can be loaded in Creo Elements/Direct 2D Access:

- 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 files (*.bdl) (Note that only the 2D portions of such files are extracted, while any 3D portions are ignored.)
- HP-GL and HP-GL2. Enter `LOAD_HPGL` and then specify the name of the HP-GL or HP-GL2 file you want to load.
- Bitmap files in the formats TIFF (*.tif), JPEG (*.jpg) and BMP (*.bmp).

Loading a Subpart

Optionally, you can use the `LOAD SUBPART` command to add a drawing as a subpart to a drawing that is already loaded.

Proceed as follows:

1. Enter `LOAD SUBPART`.
2. Enter the name of the drawing file you want to load as a subpart drawing.

-
3. Position the reference point of the loaded part relative to the drawing's coordinate system. This can be done in two ways:
 - a. Pick the location of the part's reference point, or
 - b. Enter the qualifier `ABSOLUTE` to load several drawings onto each other using the standard or absolute coordinates. Creo Elements/Direct 2D Access automatically loads the part back into the same position, relative to the origin, from which it was originally stored.

The system will now load the subpart drawing file beneath the top part. Refer to [Spotlighting Parts on page 39](#) for further information on viewing parts in Creo Elements/Direct 2D Access.

Using the File Browser

You can load and store drawings quickly and easily by means of the File Browser. You do not need to remember long filenames and subdirectories as these are displayed.

To load a drawing in the File Browser, pick `FBROWS` and then pick the desired directory. Pick the desired file in that directory. The file is highlighted. Either pick `Ok` or pick the highlighted file again to load it.

Optionally, repeat this procedure to load additional files in the same Creo Elements/Direct 2D Access session.

Using the File Browser Options

The File Browser lets you set various options to search, sort, and set filters.

Directory	Displays list of directories/ subdirectories
Searchpath	Displays list of searchpaths to be searched
Sort Up	Sorts the catalog in alphabetical order
Sort Down	Sorts the catalog in reverse alphabetical order
Reread	Updates the window to reflect any change in directory contents
Filter On	Switches on filtering
Filter Off	Switches off filtering
Preview	Displays a preview image of the drawing file you selected

To set a pattern, pick the field marked with an asterisk (*) and enter the pattern on the command line.

Using the CATALOG Command to List Files

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

1. Enter CATALOG
2. Specify the directory to be listed.
3. Specify the applicable output option: SCREEN, APPEND, DEL_OLD, or prt (printer).
4. To return to the Creo Elements/Direct 2D Access screen menu, press [Ctrl] [D].

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.

Creating a New Instance of Creo Elements/Direct 2D Access

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 the **FILE 1** menu, pick ToNewInst.
2. To open an empty Creo Elements/Direct 2D Access instance (without any drawing file), enter an empty string (two apostrophes) into the user input line:
' '

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 Drafting 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 2D Access instance are available in that instance only.
-

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 the **FILE 1** menu, pick **ToNewInst**.
2. Specify the filename (complete pathname) of the drawing file you want to load.

3

Viewing and Measuring Drawings with Creo Elements/Direct 2D Access

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

Many of the commands and functions described in this chapter can be picked
directly from the screen with your mouse. Other command and function names
must be entered using the keyboard.

Note

The following instruction sequences assume that your screen is displaying
three smaller viewports (the SM viewport layout). This means that you have to
pick a viewport for many commands and functions. If you use the LRG
viewport mode (one large viewport), you do not need to pick a viewport. Press
LRG/SM to toggle between these two viewport modes.

The Catch, Measure, Show, Window, Viewports, and Ruler or Grid commands serve as "interrupt functions". This means that they can interrupt a command or another function without terminating it. This is helpful because it allows you to use these functions at any time and yet still return to the unfinished command in which you were working.

For example, while using the ANGLE command to measure an angle, you may want to draw a new window to view your drawing in detail. Since NEW in WINDOW is a function, you can draw the new window and then continue measuring the angle.

Using Window Functions

The window functions help you view your drawing. For example, using NEW, you can place a detail area from your drawing in another viewport and inspect it closer. Other commands allow you to pan, redraw, and fit your drawing.

The window menu is displayed in the lower half of the screen menu. When you first start up Creo Elements/Direct 2D Access, the window menu is automatically displayed. If the lower half of the screen menu displays another menu, pick WINDOW to display the window menu again.

This section describes the default norm (normal) window functions, that is, they are not dynamic. To use dynam window functions, see [Using Dynamic Window Functions on page 22](#).

Fitting a Drawing

Use the FIT function to fit your entire drawing into a picked viewport. To view your entire drawing in a viewport:

1. Pick FIT in WINDOW.
2. Pick the viewport for your drawing to fill.

Creo Elements/Direct 2D Access fills the picked viewport with the entire drawing.

Redrawing

Use the REDRAW function to redraw the displayed part of your drawing in a picked viewport. This is sometimes necessary if you use a SHOW function, and part of the drawing is obscured. To redraw:

1. Pick REDRAW in WINDOW.
2. Pick the viewport to be redrawn.

Switching Between Large and Small Viewport Layouts

Use the LRG/SM function in WINDOW to toggle between the large and the small viewport layouts. The LRG (large) viewport layout displays a single viewport using all the viewing area and the SM (small) viewport layout displays three viewports in the viewing area.

Making a Window Current or Active

To make a window current or active:

1. Enter CURRENT_VIEWPORT.

-
2. Pick the viewport.

Creo Elements/Direct 2D Access makes the picked window the active viewport.

Restoring the Last View in a Viewport

Use the LAST function to redraw the previous view in a picked viewport. If there is no previous view, the current view is redrawn. To redraw the previous view:

1. Pick LAST in WINDOW.
2. Pick the viewport for the previous view.

Creating a New Window

Use the NEW function to box a specific area of your drawing (for example, a detail) and place it in a viewport.

To create a new view:

1. Pick NEW in WINDOW.
2. Pick the bottom left-hand corner and then the top right-hand corner of the detail area you would like to place in the viewport.
3. Pick the viewport to place the detail in. You may also enter ALL to place the new view in all viewports.

Panning Across a Drawing

Use the PAN function to move across the drawing. You can pan in any direction.

To pan across your drawing:

1. Pick PAN in WINDOW.
2. Pick the viewport you want to pan across.
3. Pick a first point on your drawing.
4. Pick a point where you want the first point to go.

Alternatively, press the [Ctrl] key and the left mouse button. Then move the mouse to pan the drawing.

Zooming In or Out

Use the ZOOM function to expand or contract a view by a zoom factor (scale factor). A positive zoom factor zooms in on the view; a negative zoom factor zooms out of the view. To zoom in or out:

1. Pick ZOOM in WINDOW.

-
2. Pick the viewport to be zoomed.
 3. Enter the zoom factor or pick a point when the box on the screen contains the desired area to be zoomed.

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

Creating a New Center Point for Your Drawing

The following function lets you center the drawing on any point you pick. To center your drawing:

1. Pick CENTER in WINDOW.
2. Pick a new center point on the drawing.
3. Pick the viewport for the new view or enter ALL if you want all viewports to display the new view.

Storing and Recalling Windows

Use the STORE function to store a view you use often so it can be recalled later.

To store your view, enter STORE_WINDOW.

Use the RECALL function to recall the stored view.

To recall your stored view, enter RECALL_WINDOW.

Using Dynamic Window Functions

Dynamic window functions give you immediate feedback on how your drawing will look after you complete the command sequence. These functions are especially useful when you are using accelerated graphics hardware.

You will find that using display lists together with dynamic functions causes your drawing to be displayed faster on the screen.

If you are using accelerated graphics hardware, you can achieve smooth control of your displayed drawing using real-time zooming and panning.

If you are not using accelerated graphics hardware, you may not notice much difference between normal and dynamic window functions. In this case we suggest using the normal window functions as described in the previous sections.

Generating Display Lists

This subsection describes display lists. Display lists are helpful when using dynamic window functions because they allow the drawing to be quickly redrawn on the screen.

A display list is graphical information that is independent from the main drawing data structure. When display lists are switched on, very fast redraw, zoom, and pan operations are performed using the data in the display list.

Note

To prevent a decrease in system performance when using display lists we recommend:

- Not changing the spotlight setting for parts when display lists are on.
- If you change the show status (for example, switching off hatching) while display lists are on, regenerate them regularly to reduce their size to a minimum. To regenerate a display list in the current viewport, pick on in DISPL LIST and then pick the viewport.

Because display lists can consume large amounts of memory, selectively choose the viewports in which they should be generated. When you finish using the display list in a viewport, you can switch it off by picking off in DISPL LIST and then picking the viewport.

Note

Although dynamic window functions can be used without switching display lists on, we recommend using display lists with dynamic functions to produce quick screen redraws.

To generate or regenerate a display list for the current viewport:

1. Pick on in DISPL LIST in the WINDOW menu.
2. Pick the viewport to generate the display list in.

Creo Elements/Direct 2D Access generates a display list and displays two numbers in the lower left-hand corner of that viewport. The left number is the total size of the display list in bytes. The right number is the size in bytes of the part of the display list used, for example, for turning SHOW functions off or using the move or delete commands. If the right-hand number becomes large, regenerate the display list.

Display list remains active until you switch it off by picking off in DISPL LIST and then picking the viewport. Once the display list is switched on in a viewport, you can use any of the dynamic window functions in that viewport without generating the display list again.

 **Note**

Be sure that the viewport you choose for dynamic functions always has DISPL LIST on.

Creating a New Window Using the Dynamic Function

To create a new window using the dynamic function:

1. Pick dynam in NEW.
2. Pick the viewport for your new window. (Make sure it has DISPL LIST on.)
3. Pick the bottom left-hand corner and then the top right-hand corner of the detail area of any viewport you would like to place in your picked viewport.

Creo Elements/Direct 2D Access redraws the detail area to fill the picked viewport.

Panning Your View Using the Dynamic Function

Dynamic panning gives you immediate feedback of how your drawing will look after you complete the panning function. Use dynamic panning as follows:

1. Pick dynam in PAN.
2. Pick the viewport you want to pan across.
3. Pick a first point on your drawing. (Make sure the viewport has DISPL LIST on.)
4. Pick a point where you want the first point to go.

Creo Elements/Direct 2D Access redraws your drawing at the new position in the viewport.

Zooming In or Out Using the Dynamic Function

The dynamic ZOOM function can be used to expand or contract your view. To use the dynamic ZOOM function:

1. Pick dynam in ZOOM.
2. Pick viewport to be zoomed. (Make sure it has DISPL LIST on.)
3. Enter the zoom factor or pick a point.

Creo Elements/Direct 2D Access expands or contracts the view in the viewport.

Creating a New Center Point Using the Dynamic Function

To create a view of your drawing with a new center point using the dynamic function:

1. Pick dynam in CENTER.
2. Pick the viewport to be recentered. (Make sure it has DISPL LIST on.)
3. Pick a new center point on the drawing.

Creo Elements/Direct 2D Access centers the view in the picked viewport.

Selecting What is Shown On Your Screen

The SHOW command enables you to hide or display entire item classes in the current viewport.

Switching Dimensions, Hatching, Text or Everything On/Off

Pick on or off to switch all dimensions (DIM), hatching (HAT), text (TXT) or everything (ALL) on or off in the current viewport.

Switching Construction Lines On/Off

- To switch off all construction lines in the active viewport, enter SHOW GLOBAL C _GEO ALL OFF.
- To switch on all construction lines in the active viewport, enter SHOW GLOBAL C _GEO ALL ON.

Switching Geometry Lines On/Off

- To switch off all geometry in the active viewport, enter SHOW GLOBAL GEO ALL OFF.
- To switch on all geometry in the active viewport, enter SHOW GLOBAL GEO ALL ON.

Switching Layers On/Off

Design engineers use layers in complex drawings where it is useful to associate drawing elements with a definable plane of reference. Switching a layer off will remove visually any elements associated with the layer.

After entering the command, enter the layer number you want to switch off. You can use the command several times to turn off several layers.

To switch a layer off in the active viewport:

1. Enter `I_SHOW_LAYER_OFF`.
2. Enter the layer number.

To switch a layer on in the active viewport:

1. Enter `I_SHOW_LAYER_ON`.
2. Enter the layer number.

Switching Vertices On/Off

You can use `SPLIT` on for checking whether lines are split or overlapping.

- To switch off vertices in the active viewport, enter `SHOW VERTEX OFF`.
- To switch on vertices in the active viewport, enter `SHOW VERTEX ON`.

Switching Info On/Off

An info is a piece of information attributed to an element or group of elements.

- To switch off info in the active viewport, enter `I_SHOW_INFO_ON`.
- To switch on info in the active viewport, enter `I_SHOW_INFO_OFF`.

Changing the Catch Mode

Catching allows you to select a point on the screen without having to put the cursor directly on that point. The system will catch to a given point only if the point lies within the (imaginary) circle around the edges of the cursor.

Combining the `CATCH` and `MEASURE` functions provides a powerful tool in determining geometrical values.

To change the catch mode so the cursor catches to a specified point, choose one of the following options:

- Pick **ALL** to set default catch mode as described in the following section.
- Pick **INTERS** for the nearest intersection point.
- Pick **CENTER** for the nearest center point.
- Enter `CATCH GRID` for the nearest ruler or grid point.
- Enter `CATCH VERTEX` for the nearest vertex point.
- Enter `CATCH ELEM` for the nearest element.

The catching modes are active for one catch only, after which the system switches automatically back to **ALL**. If you want the chosen catch mode to remain active until you change it, enter the appropriate permanent option as follows:

- CATCH PERMANENT INTERS
- CATCH PERMANENT CENTER
- CATCH PERMANENT GRID
- CATCH PERMANENT VERTEX
- CATCH PERMANENT ELEM

Creo Elements/Direct 2D Access will set the chosen catch mode active until you choose another catch mode.

Resetting the Default Catch Mode

The default catch mode is **ALL**. Pick **ALL** and the cursor will catch using the following order of priority:

1. Vertex
2. Inters
3. Ruler/Grid
4. Elem

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

To switch catching off completely, enter `CATCH OFF`.

The current catch mode is always displayed in the System Status Line towards the bottom of the screen.

Setting the Catch Range

To alter the size of the catch range:

1. Enter `CATCH RANGE` and the current value will be displayed.
2. Enter the new value (the size in pixels).
3. Press `[Return]`.

The size of the small cursor will change to cover the new catching range. This allows you to visually check if the catching range is suitable for your purposes.

Making Measurements

You can measure dimensions on your drawing with Creo Elements/Direct 2D Access. This option is useful if a dimension is unclear or missing on an existing drawing feature. You can also use this option to check any existing dimension. Creo Elements/Direct 2D Access displays the geometrical value in the User Input Line at the bottom of the screen. When you have noted the value, press [Return] to continue.

Setting the Units of Measurement

The current linear and angular units of measurement are displayed on the System Status Line at the bottom of the Creo Elements/Direct 2D Access window.

To change these units:

1. Enter UNITS.

Creo Elements/Direct 2D Access prompts for the units you want to set.

2. Enter the applicable linear unit:

- CM, METERS, KM, MM, and UM (micro-meter).
- INCHES, FEET, YARDS, MILE, MILS, and UINCHES (micro-inch).

3. Enter the applicable angular unit:

- DEG, GRD (gradian), and RAD (radian).

Creo Elements/Direct 2D Access displays the current units in the System Status Line.

The MEASURE functions are dependent on the current catch mode. The catch mode you choose allows you to select a point on the screen without having to put the cursor directly on that point.

Measuring Distances (Horizontal and Vertical)

To measure the distance between two points:

1. Pick DIST in MEASURE.
2. If the two points are either horizontal or vertical from each other, enter HORIZONTAL or VERTICAL using the keyboard.
3. Pick the two points.

Measuring Lengths

Use the LENGTH function to measure the length of an element. An element can be a line, circle, arc, fillet, chamfer, or spline. To measure the length:

-
1. Pick LENGTH in MEASURE.
 2. Pick an element.

Measuring Angles

Use the ANGLE function to measure the angle between two lines. The angle measured is in degrees and is the counterclockwise value. To measure the angle:

1. Pick ANGLE in MEASURE.
2. Pick a point on the first line.
3. Pick a point on the second line.

Measuring a Radius

Use the RADIUS function to measure the radius of a curved element. A curved element can be a fillet, arc, or circle. To measure the radius:

1. Pick RADIUS in MEASURE.
2. Pick the curved element.

Measuring an Area

Use the AREA function to calculate the area and area properties of a picked face. The face must be completely bounded by elements. Creo Elements/Direct 2D Access measures the area of both single faces and multiple faces.

For a single face, Creo Elements/Direct 2D Access calculates and displays the following area properties:

- number of holes in the face area
- 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 with respect to the center of area
- angle between coordinate system and principal axes

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.

For multiple faces, Creo Elements/Direct 2D Access calculates and displays the following total values:

- 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 measure the area properties:

1. Pick AREA in MEASURE.

Creo Elements/Direct 2D Access prompts you to enter a density (the default is 1) or identify a face(s).

2. Enter the density value if known. If not, go to the next step.
3. Pick a face(s).

Creo Elements/Direct 2D Access prompts for an output option SCREEN, DEL_OLD, APPEND, or filename.

4. Enter SCREEN to display the information.

The screen clears and Creo Elements/Direct 2D Access displays the area properties information.

-
- The density value is reset to 1 after using an output option.
5. Pick [Reset] or enter [Ctrl] [D] to return to the Creo Elements/Direct 2D Access screen.

 **Note**

Creo Elements/Direct 2D Access marks the Center of Mass on the faces you identify. If you identify multiple faces when using Area, then Creo Elements/Direct 2D Access also marks the Common Center of Area (+) and the Common Center of Mass (o). To remove these marks, pick REDRAW

Displaying Point Coordinates

Use the POINT function to display the exact coordinates of a picked point. The coordinates are relative to your drawing's datum reference point.

To display the coordinates:

1. Pick POINT.
2. Pick a point.

Creo Elements/Direct 2D Access displays the geometrical value at the bottom of the screen.

Using the Ruler or Grid

The ruler and grid help you to accurately place drawings onto one another.

The ruler simulates the T-square traditionally used with manual drafting methods. The grid can be displayed either as a dot grid or a line grid. When you use the ruler together with the gridpoint catch mode, the ruler allows you to accurately place drawings with the help of a variable grid.

The commands for the ruler and grid are not included in the screen menu but may be used by entering the command using the keyboard.

Displaying the Ruler

To switch on the ruler in the active viewport, enter RULER ON. To switch it off, enter I_RULER_GRID_OFF.

Displaying the Dot Grid

To display a dot grid in the active viewport, enter `DOT_GRID ON`. To switch it off, enter `I_RULER_GRID_OFF`.

Displaying the Line Grid

To display a line grid in the active viewport, enter `LINE_GRID ON`. To switch it off, enter `I_RULER_GRID_OFF`.

Switching the Ruler or Grid Off

To switch off the ruler, the dot grid, or the line grid in the active viewport, enter `I_RULER_GRID_OFF`.

Displaying the Origin

Use the following command to mark the coordinate axes with an origin symbol.

- To switch on the origin symbol at the axes origin, enter `ORIGIN ON`.
- To switch off the origin symbol, enter `ORIGIN OFF`.

Setting the Cursor Size

To toggle from a small to a full-screen cursor or vice versa, enter `I_RULER_GRID_CURSOR_LG_SMALL`. Note that this command does not change the catching range of the cursor.

Moving the Ruler

Use the following command to move the ruler to a new position. The orientation of the ruler axes will remain the same. To move the ruler:

1. Enter `CS_REF_PT`.
2. Pick the new position for the origin.

Rotating the Ruler or Grid with Respect to the X-Axis

Use the following command to rotate the ruler or grid so that the x-axis is at a given angle to the horizontal. To use this command:

1. Enter `CS_ROTATE ABSOLUTE`.
2. Enter an angle of rotation.

Rotating the Ruler or Grid Through a Given Angle

Use the following command to rotate the ruler or grid through a given angle. A positive angle rotates the ruler or grid counterclockwise. To rotate the ruler or grid:

1. Enter `CS_ROTATE RELATIVE`.
2. Enter an angle of rotation.

Setting the Ruler or Grid Spacing

Use the following command to set how many units are between each stroke marked on the ruler or grid. If the divisions are very small, they are not shown on the screen. Creo Elements/Direct 2D Access will still use the divisions for catching purposes. The current spacing is shown in the System Status Line. To set ruler or grid units:

1. Enter `GRID_FACTOR`.
2. Enter a grid spacing value.

Creo Elements/Direct 2D Access displays the set spacing as an entry in the System Status Line.

Drawing an Isometric Ruler or Grid

Use the following commands to redraw the ruler or grid according to the isometric grid required for positioning top, front, or side views.

For top isometric views:

- Enter `CS_AXIS ABSOLUTE 30 1 150 1`.

For front isometric views:

- Enter `CS_AXIS ABSOLUTE 150 1 90 1`.

For side isometric views:

- Enter `CS_AXIS ABSOLUTE 30 1 90 1`.

Resetting the Ruler or Grid

To reset the ruler or grid, enter `I_RULER_GRID_RESET`. This command resets the ruler or grid to the default system orientation and origin. This command also resets the ruler axes to a mutually orthogonal position.

Placing the Ruler or Grid

Use the following commands to change the orientation of the x-axis and the y-axis. The cursor cross-hairs will change to the new orientation. Two command options are available: by three points and by relative orientation.

By Three Points

To place the ruler or grid by three points:

1. Enter `CS_SET`.
2. Pick the origin, a point on the x-axis, and a point on the y-axis.

This option changes the axes orientation relative to the current orientation (this may not be the same as the original orientation). To place the ruler or grid by relative orientation:

1. Enter `CS_SET RELATIVE`.
2. Pick two points that coincide with the new x-axis of the ruler.

Creo Elements/Direct 2D Access changes the cross-hairs to the new orientation.

Using Viewports to Display Several Views

Creo Elements/Direct 2D Access uses viewports to display areas of your drawing. Each viewport has a viewport number. Creo Elements/Direct 2D Access displays the viewport number in the top right-hand corner of the viewport.

If a viewport is partially or totally covered by other viewports, it is made inactive. This means that Creo Elements/Direct 2D Access does not update the viewport's visual contents. If you make a viewport active or you enter the command `NEW_SCREEN`, Creo Elements/Direct 2D Access updates the visual contents of the viewport.

You can divide the viewing area of the screen to contain up to sixteen viewports.

The `VIEWPORT` commands are not included in the Creo Elements/Direct 2D Access screen menu. You can use the following commands by entering the command name using the keyboard.

Creating a Viewport

Use the following command to create a new viewport. You can also define the color of the viewport and its size. To create a new viewport:

1. Enter `CREATE_VIEWPORT`.
2. Enter a viewport background color. The colors available are: `BLACK`, `WHITE`, `YELLOW`, `BLUE`, `GREEN`, `RED`, `MAGENTA`, and `CYAN`.

-
- Pick the bottom left-hand corner and then the top right-hand corner of the new viewport.

Making a Viewport Active

Most of the Creo Elements/Direct 2D Access commands not available on the screen only work in the active viewport. The number of the active viewport is always highlighted.

There are two methods of making a viewport active or current.

- Pick the viewport number in the upper right corner of the viewport you want to set as active.
- The second method uses the following command to set a viewport active:
 - Enter `CURRENT_VIEWPORT`.
 - Enter the viewport number, or pick the viewport.

Deleting a Viewport

Use the following command to delete a viewport. If you delete the active viewport, you must make another viewport active before you can continue working. To delete a viewport:

- Enter `DELETE_VIEWPORT`.
- Enter the viewport number, or pick the viewport. To delete all the viewports enter `ALL`.

Creo Elements/Direct 2D Access deletes the viewport.

Changing the Current Viewport Size

To change the size of the active viewport:

- Enter `CHANGE_VIEWPORT_SIZE`.
- Pick the bottom left-hand corner and then the top right-hand corner of the new viewport.

Creo Elements/Direct 2D Access alters the size of the active viewport.

Changing the Current Viewport Color

To change the background color of the active viewport:

- Enter `CHANGE_VIEWPORT_COLOR`.

-
2. Enter a color. The colors available are: BLACK, WHITE, YELLOW, BLUE, GREEN RED, MAGENTA, and CYAN.

Creo Elements/Direct 2D Access changes the color of the active viewport.

Redrawing All Viewports

To redraw and update all the viewports and the menu areas, enter `NEW_SCREEN`.

Using Parts

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

Introduction to 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.

There are several groups of commands for viewing or highlighting an individual part or a group of parts. These commands are called the PARTS commands and they are not included in the Creo Elements/Direct 2D Access screen menu. You can use the commands by entering the command name using the keyboard.

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 pick an element from the part.

Creo Elements/Direct 2D Access sets the picked 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 SCREEN`.

Creo Elements/Direct 2D Access displays the list of parts.

-
2. Press [Ctrl] [D] to return to the Creo Elements/Direct 2D Access screen menu.

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.
2. Press [Ctrl] [D] to return to the Creo Elements/Direct 2D Access screen menu.

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. Pick 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 pick an element of the part.

Creo Elements/Direct 2D Access draws only the picked 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.

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.

4

Viewing Hidden Lines

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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.

Pick 2DHL 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.

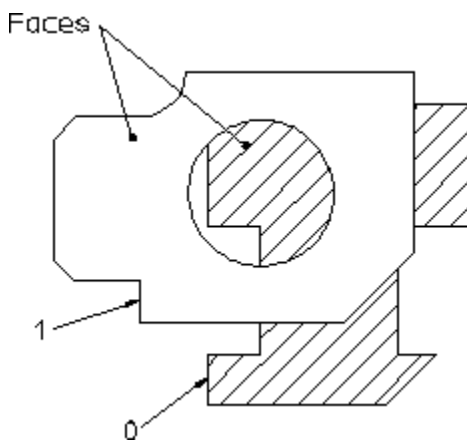
What are Faces?

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 1. Faces - With Z-Values 0 and 1 on page 41](#) shows the faces of two components (one with z-value 0 and the other with z-value 1).

Figure 1. Faces - With Z-Values 0 and 1

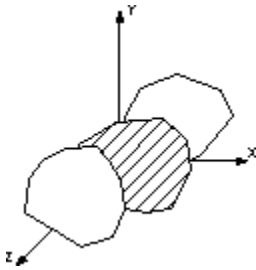


What are Z-Values?

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 2. Visualizing Z-Values on page 42](#).

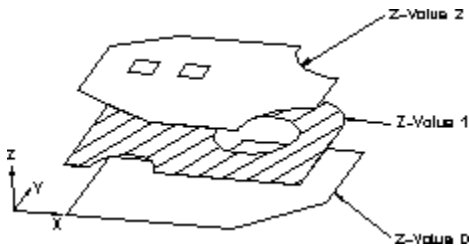
Figure 2. Visualizing Z-Values



A face or line with a higher z-value covers a face or line with a lower z-value.

Figure 3. Z-Values 0, 1 and 2 on page 42 shows 3 faces assigned the z-values 0, 1 and 2.

Figure 3. Z-Values 0, 1 and 2



Hatches are treated as faces. This means that 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

To check the z-value of an individual element:

1. Pick **2DHL**.
2. Pick **INQ Z-VAL**.
3. Identify the element.

Creo Elements/Direct 2D Access displays the element's z-value in the System Input Line. The value is also written into the System Inquiry Array.

If the specified element has no assigned z-value, an error beep is given. If your assembly drawing has no z-values assigned, you will not be able to generate hidden lines.

Previewing Hidden Lines

If your assembly drawing has z-values assigned to its faces, you can preview the drawing to see how it will look after you generate hidden lines.

This preview mode redraws your assembly drawing on the screen, removing those lines that will later be generated as hidden.

The preview mode does not compute hidden lines, change the structure of the assembly or alter geometry. It only affects the view.

Note

HIDD DRAW affects all WINDOW functions as each takes the z-value information into account. You cannot use display lists (Displ List On in WINDOW) together with hidden line preview mode in the same viewport.

To preview your assembly drawing before you actually generate hidden lines:

1. Click **2DHL**
2. Click **HIDD DRAW**
3. Select the viewport

Creo Elements/Direct 2D Access displays the drawing partially covered (depending on the z-values associated with the assembly drawing).

Note that Creo Elements/Direct 2D Access displays **-H-** in the upper left-hand corner to indicate hidden mode.

Note

If you try to plot your drawing while in the preview mode, Creo Elements/Direct 2D Access will plot the normal drawing, not the hidden line view you see on your screen.

To view your drawing the way it normally looks without previewing hidden lines, use the **NORM DRAW** command. This is the default mode. To redraw normally:

-
1. Pick **2DHL**.
 2. Pick **NORM DRAW**.
 3. Identify the appropriate viewport.

Creo Elements/Direct 2D Access redraws your assembly drawing the way it normally looks, without hidden lines displayed.

Generating Hidden Lines

In contrast to the preview mode, the generation mode actually computes hidden lines and changes the drawing in the Creo Elements/Direct 2D Access workspace as follows:

- Geometry is modified by splitting geometric components if they are covered by faces with a higher z-value.
- Splines are converted if necessary.
- Shared parts are unshared.
- The parts tree, starting from the current part, is completely smashed.

However, note that the original drawing file remains unchanged.

To generate hidden lines:

1. Pick **2DHL**
2. Pick **GEN HIDDEN**
3. Pick **CONFIRM**

Creo Elements/Direct 2D Access generates hidden lines.

Note

You cannot go back to the preview mode once you have generated hidden lines. Generating hidden lines is not reversible.

Showing Hidden Lines On/Off

Once you have generated hidden lines, the hidden lines can be switched on and off by picking **SHOW ON** or **SHOW OFF** in **2DHL**.

To switch hidden lines on or off:

1. Pick **2DHL**
2. Choose from the following:

-
- a. Pick **SHOW ON** to display your assembly drawing with hidden lines.
 - b. Pick **SHOW OFF** to display your assembly drawing without hidden lines.

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.

Dumping Hidden Lines

You can use the screen dumping commands to produce a copy of your drawing that includes face colors (if you have a color printer). Screen dumps can be made at any stage in the drawing process. Refer to [Plotting Drawings and Dumping Screens on page 46](#).

5

Plotting Drawings and Dumping Screens

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This chapter shows you how to:

- Plot drawings on a plotter.
- Dump screens to a printer.

Pick **PLOT**. The **PLOT** menu appears on the lower half of the screen menu.

Pick **CONFIG** at the bottom of the **PLOT** menu. The **PLOT CONFIGURATION** menu appears on the lower half of the screen menu. It contains a table of predefined plot configurations.

To display the **PLOT** menu again, either pick **MENU 1** from the **CONFIG** menu, or pick **PLOT** from the top of the screen menu.

Depending on the system configuration, the spooler can:

- Continuously send plots to the plotter which plots them automatically.
- Queue up the plots. This means you must initiate each plot.

The spooler can store several drawings at one time so each user can send plots instantly. The plotting queue can, however, cause the drawing not to be plotted for several minutes.

To plot your drawing you must do the following:

1. Pick **PLOT** to display the **PLOT** screen menu.

-
2. Set up the desired plot parameters: **POSITION**, **SCALE**, **ROTATE ANG**, and **SIZE**. (Refer to the appropriate sections in this chapter).
 3. Select a predefined plot configuration from the **PLOT CONFIGURATION** menu. (Refer to [Selecting a Predefined Plot Configuration on page 53](#).) If your plotter is not on the menu, you can define it by following the instructions in [Selecting the Plotter Type on page 54](#) and [Naming the Plot Destination on page 55](#).
 4. Start the plot by picking **START PLOT**. Refer to [Starting a Plot on page 58](#).

Defining the Plotting Parameters

Before plotting a drawing you should define exactly what will be plotted, and how the drawing will be plotted.

The following subsections show you how to do this.

Positioning Your Drawing

This section shows how to position your drawing on the paper before plotting.

The two **POSITION** options are:

- cent
- left

Centering the Drawing

This is the default option.

To position your drawing at the center of the plot limits prior to plotting:

1. Pick **PLOT**
2. Pick cent in **POSITION**

The subsequent plots will be centered.

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 of the plot limits prior to plotting:

1. Pick **PLOT**
2. Pick left in **POSITION**

The subsequent plots will be placed in the lower left corner of the paper.

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.

Setting the Plot Scale

This section shows how to set the plotting scale to be used for the subsequent plots.

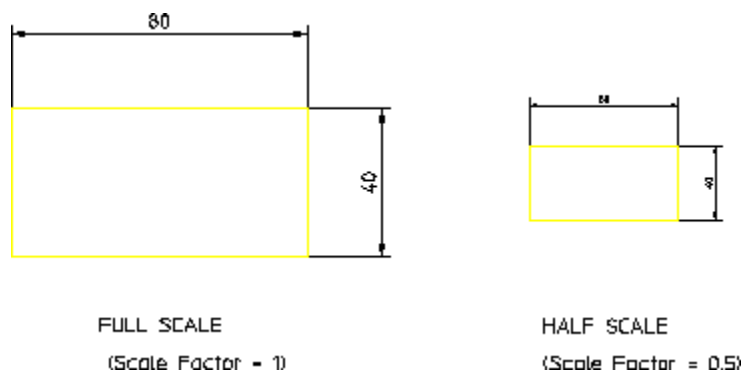
You can set a plotting scale factor or choose to scale the drawing to be as large as possible within the limits of your paper.

Entering a Scale Factor

This is the default option. The entire plot will be scaled. Note that actual dimensions are not affected.

Figure 4. Operation of Plot Scale on page 49 shows the operation of plot scale.

Figure 4. Operation of Plot Scale



To set the plot scale factor:

1. Pick **PLOT**.
2. Pick **SCALE**.

The current plotting scale is displayed both:

- In the User Input Line.
- In the menu slot immediately to the right of the **SCALE** command.

3. Enter the required plotting scale.

The new scale factor is displayed in the menu field immediately to the right of the **SCALE** command.

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 [BREAK] 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

To plot your drawing as large as possible within the limits of the paper, pick **PLOT** to display the **PLOT** screen menu and then pick fit.

Selecting a Plot Source

This section shows how to select a source for your plot. The plot source can be the complete drawing, the current viewport, or a boxed area.

All the Drawing

This is the default option.

Enter `Set_sys_plot_source ALL` and your complete drawing will be plotted irrespective of what is presently displayed in the viewports.

The Current Viewport

Enter `Set_sys_plot_source CURRENT_WINDOW` and the contents of the current viewport will be plotted.

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 plotted.

A Boxed Area

Enter `Plot_enter_two_pts_and_update`. Creo Elements/Direct 2D Access prompts you to pick the two opposite corners of a box. The contents of the boxed area will be plotted irrespective of what is displayed on the screen.

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

Setting the Plot Appearance

This section shows how to set the appearance of the plot. You can either use the standard drawing conventions or select your own. These commands only determine how the plot will appear. The plotting area is unaffected.

With Drawing Conventions

You can plot the drawing using standard drawing conventions. Drawing lines, dimensions, text, and hatching will be plotted. 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).

Enter `Set_sys_plot_as_displayed false`, and the subsequent plots will conform to standard drawing conventions.

With Current Show Attributes

You can select the conventions used to plot the drawing. Thus any combination of drawing lines, dimensions, hatching, text, or layers can be 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. Enter `Set_sys_plot_as_displayed true`.

The subsequent plots will contain whatever drawing conventions you have selected in the current viewport.

Setting the Rotation Angle

Use **ROTATE ANG** to rotate the drawing on the plotter paper through 90 degrees. The default angle is 0 degrees.

To set the angle of rotation to zero or 90 degrees:

1. Pick **PLOT** to display the PLOT screen menu.
2. Pick either 90 to rotate the drawing through 90 degrees at the plotter or pick 0 to reset the angle rotation back to 0 degrees.

The current slot is highlighted in blue.

To rotate the drawing through any angle:

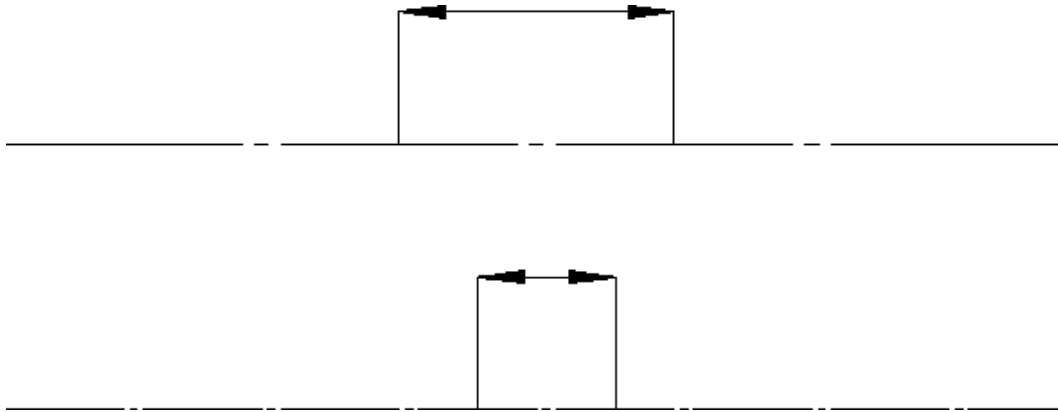
1. Enter `PLOT_ENTER_ROTATE`.
2. Enter the angle value in the current angular units.

Creo Elements/Direct 2D Access plots your drawing at that angle.

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. Using the linetype length command you can specify the length of each pattern and thus alter the fineness of the line. [Figure 5. Different Linetype Lengths on page 52](#) shows two `DASH_CENTER` lines drawn with different linetype lengths. The linetype length is shown arrowed.

Figure 5. 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:

1. Enter `PLOT_LINETYPE_LENGTH`.
2. Enter the name of the linetype.
3. Enter the required pattern length (the units used are the current length units).

If you wish to change all the linetype lengths:

1. Enter `PLOT_LINETYPE_LENGTH`.
2. Enter `ALL`.
3. Enter a single value for the linetype and all seven broken linetypes will be set to the required length.

Selecting the Paper Size

This section shows how to tell Creo Elements/Direct 2D Access what size of paper you will be using for your plot.

Each option sets the correct plotter limits for the paper to be loaded in the plotter (a full explanation of hard limits and other plotter terminology can be found in the manual supplied with your plotter).

To set the paper size:

Pick PLOT and pick the required format in SIZE.

[Table 2 on page](#) gives the ten paper size options.

Table 2. Paper Size Options.

A4	A3	A2	A1	A0
A	B	C	D	E

If you want to use paper sizes other than those listed above, pick SIZE and enter the following format information:

1. Width
2. Height
3. X offset
4. Y offset

Selecting a Predefined Plot Configuration

To save time when configuring a plot, you can select from the list of predefined plot configurations. If the plot configuration you need is not in the list, you will need to define it yourself or ask your system administrator for help. To configure a new plotter, follow the instructions in [Selecting the Plotter Type on page 54](#) and [Naming the Plot Destination on page 55](#).

Each predefined configuration contains settings for the following:

- PLOTTER TYPE
- PLOT DESTINATION
- delete old
- gener name

Note

If you manually change one of the above settings the new value will overwrite the predefined value (until you restart Creo Elements/Direct 2D Access).

To select a predefined plot configuration:

1. Pick **PLOT**.
2. Pick **CONFIG**.
3. Pick the required configuration in the table of plot configurations.

If you have many entries in the plot configuration table you can scroll through them using the scroll bar to the right of the table. Use the scroll bar:

- To scroll to the next page of the plot configuration table; pick the white square at the bottom of the scroll bar.
- To scroll to the previous page of the plot configuration table; pick the white square at the top of the scroll bar.
- To scroll anywhere between the first and last pages; pick anywhere between the top and bottom white squares. Creo Elements/Direct 2D Access will display the corresponding part of the plot configuration table.

If you scroll up or down one page, there is always one line overlap between one page and the other.

The scroll bar locator shows your position in the table. For example, if the locator is half-way down the scroll bar, then the information you see on the top line of the table is halfway through the plot configuration table.

Selecting the Plotter Type

If the plotter you want to use is not in the list of predefined plot configurations, you will need to select your plotter as described in this section.

Pick **PLOTTERTYP** to select the type of plotter you want to use. On startup the system default plotter type is '7550'.

When you select a plotter you also define which plotter language Creo Elements/Direct 2D Access will use:

- Hewlett-Packard Graphics Language - HP-GL.
- Hewlett-Packard Graphics Language - HP-GL/2.
- PostScript® Language



Note

The plotter names must be enclosed in quotes.

To select the plotter type:

1. Pick **PLOT**
2. Pick **CONFIG**
3. Pick **PLOTTERTYP**

The present plotter name will be displayed:

- In the User Input Line.

-
- In the menu field immediately below the PLOTTERTYP menu slot.
4. Enter a valid plotter name enclosed in quotes.
Creo Elements/Direct 2D Access now displays the new plotter name below the PLOTTERTYP menu slot.

Naming the Plot Destination

If you are defining your own plot configuration, you will need to name the destination to which drawings are plotted. On system startup the default plot destination is `'/usr/Drafting [version]/spooldir1/p'`.

Note

Plot destination can be a file or the standard input of another process. If the plot destination is a file, Creo Elements/Direct 2D Access automatically derives the extension for the output file according to the selected printer/plotter.

If you have a multi-sheet drawing, type `USE_SHEET_SETTINGS ON` in the User Input Line to automatically set the format and orientation of each sheet based on the sheet size. Refer to the *Creo Elements/Direct Drafting / Creo Elements/Direct 2D Access Programming Reference Guide* for the details of the command.

To name the plot destination:

1. Pick **PLOT** to display the **PLOT** screen menu.
2. Pick **CONFIG**.
3. Pick **PLOT DESTINATION**.

The present plot destination name will be displayed:

- In the User Input Line.
 - In the menu field immediately below the PLOT DESTINATION command.
4. Enter the new name then press `[RETURN]`.

The new plot destination name is displayed in the menu field immediately below the PLOT DESTINATION command.

Note

To set the orientation of a plot, use the `SET_PLOT_ORIENTATION` command with option `LANDSCAPE` or `PORTRAIT`. If you set the paper size such that the height and width are equal, the orientation is set to landscape by default.

Creating a New Plotfile

If you send your plot to a file, you can avoid accidentally overwriting an existing plotfile:

1. Pick **PLOT** to display the **PLOT** screen menu
2. Pick **CONFIG** in **PLOT**
3. Pick **New File**

When you pick **START PLOT**, Creo Elements/Direct 2D Access informs you that you cannot write to the existing plotfile name unless you first pick **Delete Old**. This option is the system default.

Note that the **New File** option has no effect when piping the plot data.

Deleting the Old Plotfile

If you send your plot to a file, you can overwrite the existing plotfile each time you plot:

1. Pick **PLOT** to display the **PLOT** screen menu.
2. Pick **CONFIG** in **PLOT**.
3. Pick **Delete Old**.

When you pick **START PLOT**, Creo Elements/Direct 2D Access will overwrite the existing plotfile.

Note that the **Delete Old** option has no effect when piping the plot data.

Plotting with the Original Plot Destination

To send all subsequent plots to the original user defined plot destination:

1. Pick **PLOT**
2. Pick **CONFIG** in **PLOT**
3. Pick **Orig Name**

When you pick **START PLOT**, Creo Elements/Direct 2D Access sends your plot to the original (existing) plot destination.

Generating Destination Names

Creo Elements/Direct 2D Access can generate a unique filename for each plot. This option is the default. The name is generated each time you pick **START PLOT** in the **PLOT** menu and consists of:

- The user defined plot name
- The paper format
- Your workstation identification name
- A time stamp

The following example shows how a typical plotfile name might appear:

```
'/usr/PE/Drafting [version]/spooldir1/pA4_ID_093723'
```

To generate plot destination names:

1. Pick **PLOT** to display the **PLOT** screen menu
2. Pick **CONFIG**
3. Pick **Gener Name**

When you pick **START PLOT**, Creo Elements/Direct 2D Access generates a unique plot destination name for your plot.

Plotting Pixmap Images

If a drawing contains pixmaps (imported images) you can specify how they are plotted with the following commands:

IMAGE QA	Specifies a high or low resolution.
IMAGE	Specifies the output of the image to be either black and white (B & W) or color (Color).
COLO	

Previewing a Print/Plot

Open the Plot Preview system viewport to see print/plot output on your screen. You see how output will look as you work on your drawing, or after you select all the necessary print settings.

To preview output:

1. Click **PLOT**.
2. Click a **PREVIEW** color setting for the viewport only:
 - **Original**: to preview color printer/plotter output.

-
- Grey-scaled: to preview black and white printer/plotter with dithering capability output.
 - B&W: to preview black and white printer/plotter output.

3. The **Plot Preview** viewport appears.

The **Plot Preview** viewport remains open until you uncheck it in the **PLOT** dialog box.

Starting a Plot

Use the **START PLOT** command to send your drawing to one of the available spoolers. The spooler queues the drawing and then sends it to a plotter.

To initiate a plot:

1. Pick **PLOT** to display the **PLOT** screen menu
2. Pick **CONFIG**
3. Pick a plot configuration from the table of predefined plot configurations. Refer to [Selecting a Predefined Plot Configuration on page 53](#). If you wish to define your own configuration refer to [Selecting the Plotter Type on page 54](#) and [Naming the Plot Destination on page 55](#).

4. Pick **MENU 1**.

The current plot configuration is displayed in the menu field to the right of the **START PLOT** command.

5. Set up the desired plot parameters: **POSITION**, **SCALE**, **ROTATE ANG**, and **SIZE**. (Refer to the appropriate sections in this chapter).
6. Pick **START PLOT**

Creo Elements/Direct 2D Access sends the drawing to the spooler, which then queues your plot request and sends your plot data to a plotter.

Dumping a Screen to a Printer

This section shows how to obtain a printout of your screen on a printer.

To obtain a screen dump:

1. Pick **PLOT** to display the **PLOT** screen menu.
2. Pick **B & W** in **IMAGE COL** if you want a black and white printout. Pick **COLOR** in **IMAGE COL** if you have a color printer.

Creo Elements/Direct 2D Access will print out the entire screen.

Dumping a Viewport or Boxed Area to a Printer

This section shows how to obtain a printout of the contents of a viewport or a boxed area of the screen on a printer.

To printout a viewport or boxed area, you will need to make changes in your `dump_screen` parameters. Before you make changes, you will need to know:

- If you want a color or b & w screen dump.
- If you want a black background on the screen dump.
- The number for printer resolution, paper length, and paper width. (You can find the default values for these numbers by entering `HELP DUMP_SCREEN_DEFAULTS`. Note the default values for resolution, paper length, and paper width for your specific printer type. You will enter these values when you change your screen dump default file as described below. Press `[Ctrl] [D]` to return to the Creo Elements/Direct 2D Access screen.)

Now you are ready to print a viewport or a boxed area.

1. Enter `DUMP_SCREEN_DEFAULTS`. Follow the screen prompts and enter your choices to set up the defaults.
2. Enter `DUMP_SCREEN`.
3. Enter `PORT` to print the contents of a viewport or `BOX` to print the contents of a boxed area.
4. Pick the viewport you want printed, or pick the lower left-hand corner of the boxed area and then the upper right-hand corner.

Creo Elements/Direct 2D Access will print out the contents of the selected viewport or boxed area.

Further Information

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

1. Enter `help plot` or `help dump_screen`.
2. Creo Elements/Direct 2D Access will display further information about plotting or dumping.

6

Creo Elements/Direct Annotation

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This chapter discusses how Creo Elements/Direct 2D Access handles Creo Elements/Direct Annotation drawings.

Introduction

The **Sheet** menu included in Creo Elements/Direct 2D Access enables you to visualize MI format drawings generated by Creo Elements/Direct Annotation, an integrated 2D module in Creo Elements/Direct Modeling for the creation and management of associative drawings.

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, annotate and edit 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 Modeling, update the relevant view(s) to visualize and review the resulting changes in 2D.
- Edit the 2D views as appropriate.

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 Drafting **Annotation** module:

- Do not copy one drawing into another drawing.
- Do not manually insert one or more sheets into another drawing.
- Do not manually insert views into a sheet (using the view definition commands of Creo Elements/Direct Annotation). Only use the **Sheet** commands provided by the Creo Elements/Direct Drafting **Annotation** module.
- Do not change or delete info texts beginning with SD or DOCU or AM.
- Do not 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 Drafting **Annotation** module.

Limitations

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

- Parts created in Creo Elements/Direct Drafting as sub-parts of the TOP part named `SD_drawing` will not be loaded into Creo Elements/Direct Annotation.
- Geometry or any other additional elements created in the TOP part named `SD_drawing` will not be loaded into Creo Elements/Direct Annotation.
- Subparts created in Creo Elements/Direct Drafting (and which are not owned by the TOP part) will not be displayed in the Drawing Browser of Creo Elements/Direct Annotation. However, their geometry is displayed in the viewport.
- Leader lines (or text with leaderlines) created in Creo Elements/Direct Drafting will not behave associatively in Creo Elements/Direct Annotation.
- Hatch holes automatically created in Creo Elements/Direct Drafting (forced by dimensions or texts when Creo Elements/Direct Drafting option `TEXT_HOLE_INSERTION` is set to ON by default) do not update associatively in Creo Elements/Direct Annotation.
- Hatch holes automatically created in Creo Elements/Direct Annotation for text do not update associatively in Creo Elements/Direct Drafting when the text is changed in Creo Elements/Direct Drafting.
- Hatches that were automatically created in Creo Elements/Direct Annotation but deleted in Creo Elements/Direct Drafting will re-appear after the next update in Creo Elements/Direct Annotation.
- 2D view parts created in Creo Elements/Direct Annotation that are renamed in Creo Elements/Direct Drafting get their original name during the next update in Creo Elements/Direct Annotation.
- If you split or merge 2D geometry in Creo Elements/Direct Drafting which was derived from the 3D geometry during an update in Creo Elements/Direct Annotation, the geometry will look like the original one after the next update in Creo Elements/Direct Annotation.

Preparing the Creo Elements/Direct 2D Access Drawing Environment

Before you load an MI file generated in Creo Elements/Direct Annotation, be sure to complete the following steps:

-
1. To make sure that the top part is empty, enter the following into the user input line:
RESET_SYSTEM
CONFIRM
 2. Creo Elements/Direct Annotation does not update the preview data in the Creo Elements/Direct Annotation MI file loaded in Creo Elements/Direct Drafting. 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 **ANNO**.

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 62](#), proceed as follows to load an MI file generated in Creo Elements/Direct Annotation.

1. In **ANNOTATION SHEETS**, click **OPEN**.
2. Select the MI file you want to load from the 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 SHEETS**, click **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 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 Sheet Browser.

Views

The Creo Elements/Direct 2D Access **Annotation** module provides a number of commands for selecting, changing and manipulating views.

Modifying Views

You can change the position of views on a sheet. Also, you can move views between sheets. Note that a warning message is displayed for any "structurally dangerous" operations that would impair 3D/2D associativity.

Hiding Hidden Lines

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

1. In **ANNOTATION SHEETS**, click **HIDE** below **HID 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 SHEETS**, click **HIDE** below **TAN 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 SHEETS**, click **SHOW** below **HID 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 SHEETS**, click **SHOW** below **TAN 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 **Annotation** module is a 2D derivative of a 3D model view in Modeling. When the underlying 3D model changes in Modeling, 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

To show differences:

1. In **ANNOTATION SHEETS**, click **SHOW DIFF**.
 - 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 DIFF**.

Changing Colors and Linetypes

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

1. In **Annotation**, click the option for the change you want to perform:
 - In **HID LINE** to change the color and/or linetype of hidden lines.
 - In **TAN LINE** to change the color and/or linetype of tangent lines.

A

Tutorial

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

Setting a Directory and Loading a Drawing

The first step of the tutorial shows you how to set a directory, catalog the directory, and load a drawing onto the screen.

1. Start 2D Access.
2. Pick FBROWS.
3. In the FILE BROWSER - DIRECTORIES window, switch to the 2D Access installation directory, for example:

```
<application data path>\PTC\Creo Elements Direct Drafting  
[version]\[version]
```

or

```
C:\Program Files\PTC\Creo Elements\Direct Drafting [version]
```

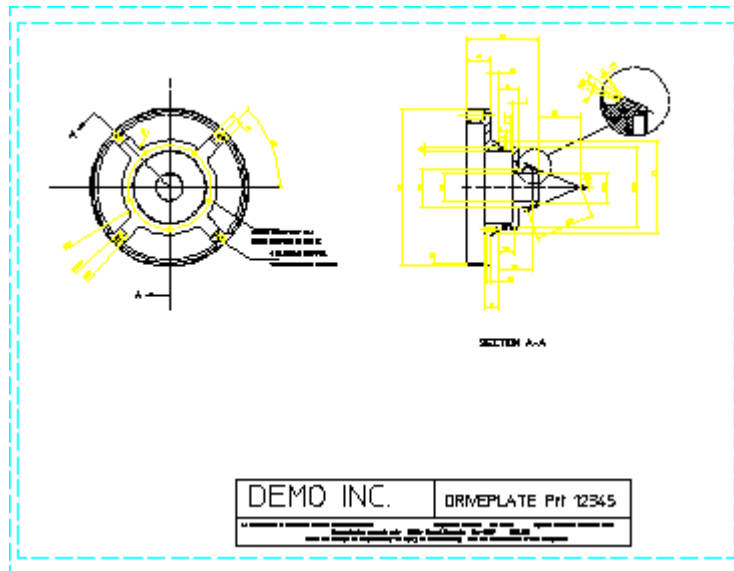
or

```
C:\Program Files\PTC\Creo Elements\Direct Drafting
```

Look at the list of files in this directory in this directory.

1. Highlight the example file and pick Ok.

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

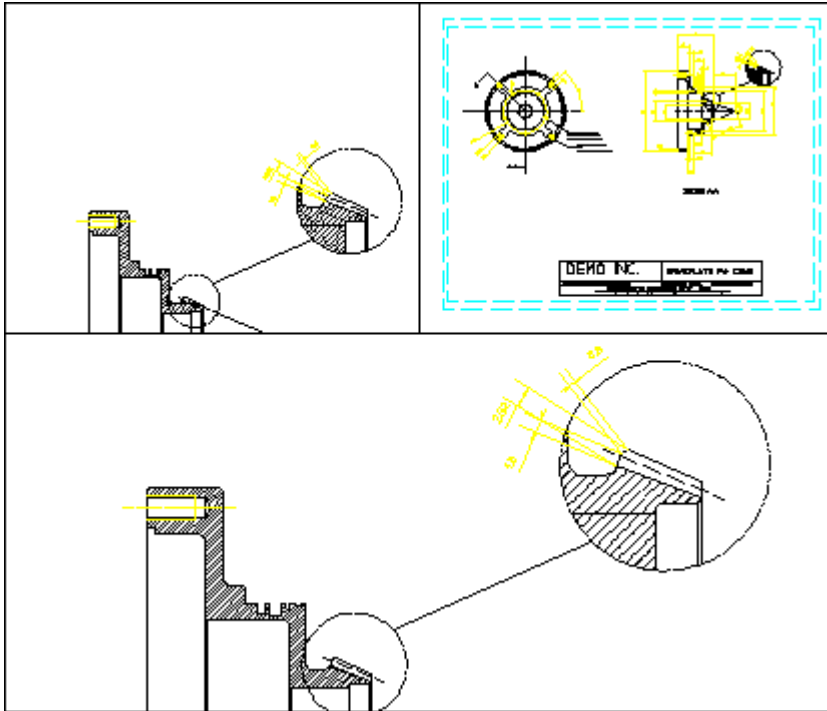


Placing Detail Areas in the Viewports

As discussed earlier, 2D Access has a large and a small viewport layout. This part of the tutorial shows you how to place detail areas in each viewport.

1. Pick LRG/SM in the WINDOW menu block.

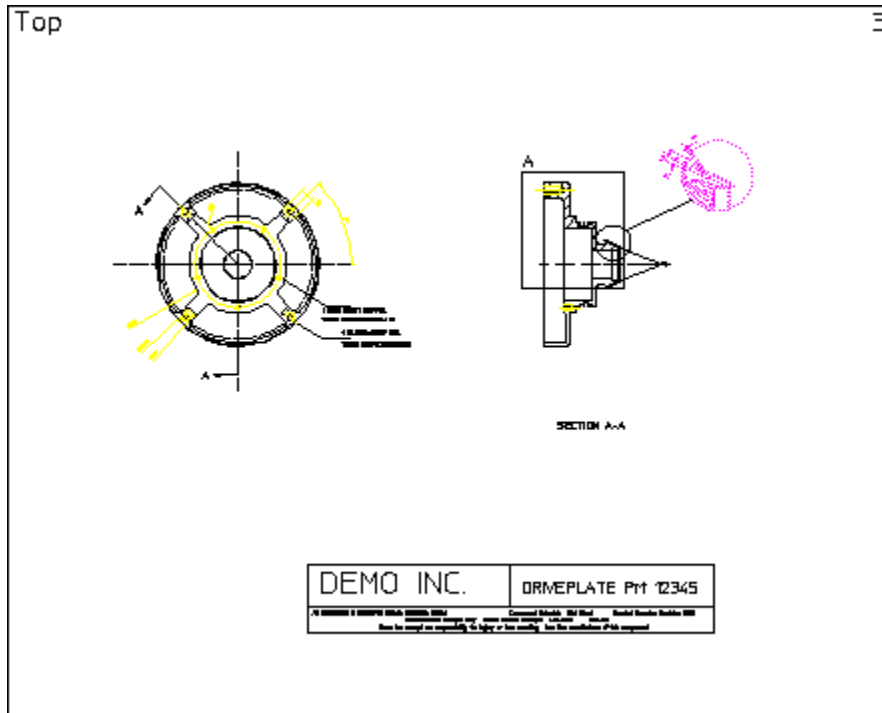
The small viewport layout will appear. There are three pre-defined viewports: viewport 1 , viewport 2, and viewport 3, as shown below.



Place a view of the right part of the drawing (viewport 3) into viewport 1. To do this:

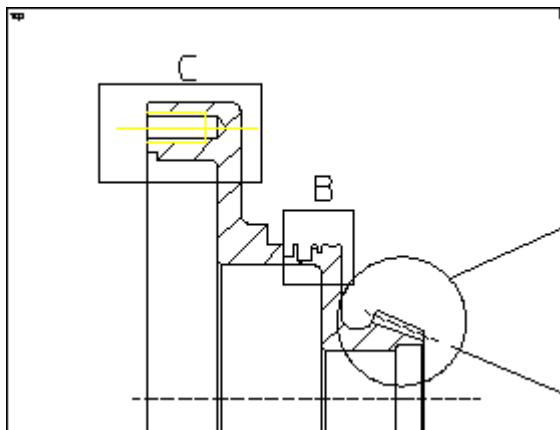
1. Pick NEW in the WINDOW menu block.
2. Box the area (A) as shown below by picking the bottom left-hand corner and the top right-hand corner. Then pick viewport 1 to indicate which viewport you want it drawn in.

2D Access displays the boxed area in viewport 1.

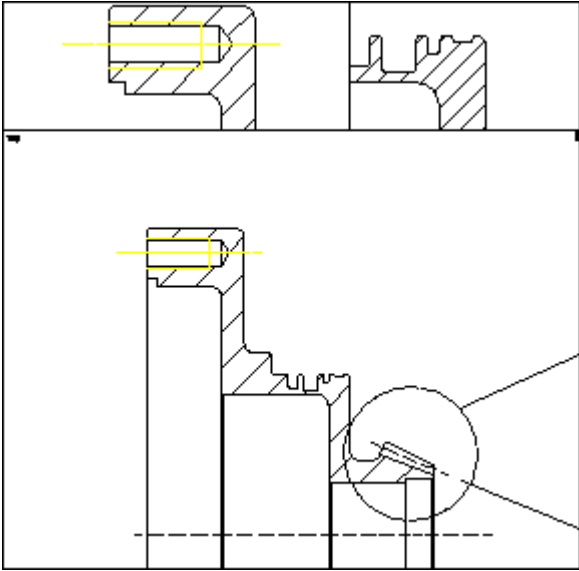


Now place a detail in each viewport:

1. Pick **NEW** in WINDOW menu block.
2. Box the detail area (B) as shown below, then pick viewport 3.
2D Access displays the boxed area in viewport 3.
3. Pick **NEW**.
4. Box the second detail area (C) as shown below, then pick viewport 2.
2D Access displays the boxed area in viewport 2.



All three viewports will now contain a view as shown below.



Measuring a Detail

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.

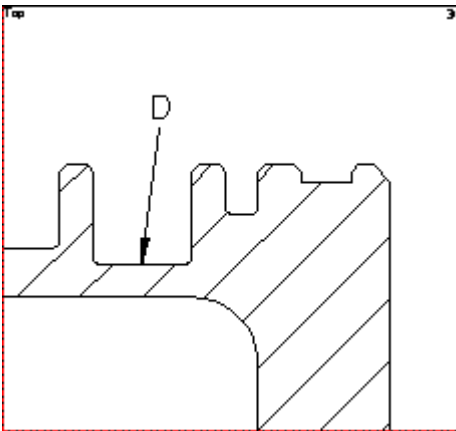
This part of the tutorial shows you how to measure basic drawing features.

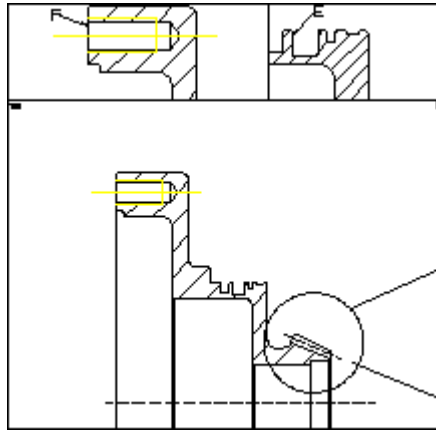
In the drawing, a dimension has been accidentally left out. The detail for this missing dimension is in the top right-hand viewport 3. To measure the length:

1. Pick **LENGTH**.
2. Pick line D shown in the following screen display.

2D Access displays the value for the length in the User Input Line at the bottom of the screen.

3. Press [Return] 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:

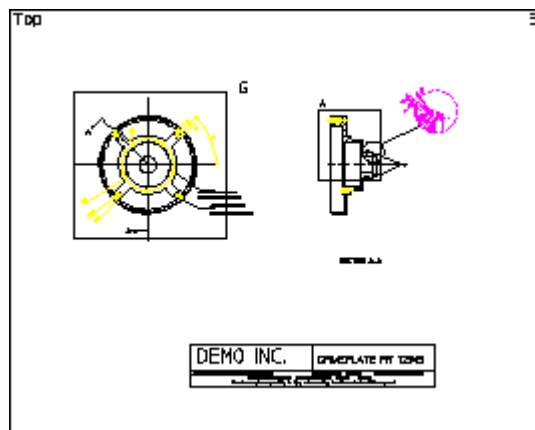
1. Pick **DIST**.
2. Enter the qualifier **HORIZONTAL** using the keyboard.
3. Pick the two points E and F as shown above.
 - 2D Access displays the distance in the User Input Line.
4. Press [Return] to continue.

Using the SHOW options

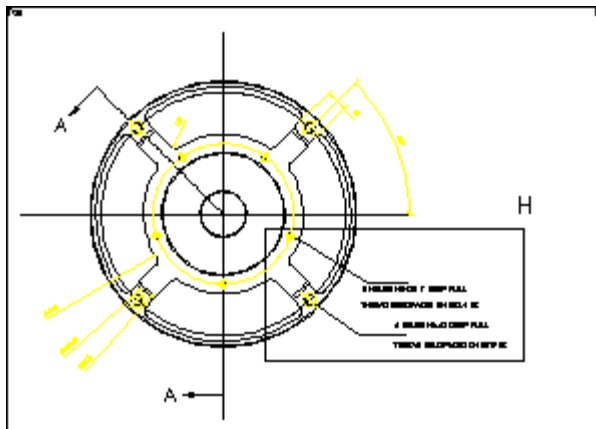
Technical drawings often include large amounts of dimensioning, hatching, and text which can obscure some details. This part of the tutorial shows how you can selectively display what is in each of the viewports.

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

1. Pick **FIT**, then pick the top right-hand viewport 3.
2. Pick **NEW**, box the area (G), and pick the bottom viewport 1.



- Pick NEW, box the area (H) and pick the top left-hand viewport 2.



You can selectively view what is in a viewport using the SHOW commands. In this case you can view the area without text. To do this:

- Pick TXT off.
- Pick the viewport.
- Pick REDRAW to see the geometry clearly.

Plotting a Drawing

You are now ready to plot your drawing. Before you plot, you need to set the plot options. The plot options define how your drawing will be plotted.

- Pick PLOT. The plot menu will appear on the lower half of the screen menu.
- Pick cent in POSITION to place the drawing in the middle of the plotting paper.
- Pick fit in SCALE to scale the drawing to be as large as possible within the limits of the paper.
- Pick 0 in ROTATE ANG if you don't want the plot to be rotated.
- Pick A4 in SIZE to plot the drawing on A4 size paper.

Now you are ready to choose the plotter and begin plotting your drawing:

- Pick **CONFIG** in the **PLOT** menu to access the **CONFIG** menu which appears on the lower half of the screen menu.
- Pick a plot configuration from the table of predefined plot configurations. If your plotter is not yet configured, your system administrator can configure it following the instructions in chapter 5, especially the sections "Selecting the Plotter Type" and "Naming the Plot Destination".
- Pick **MENU 1**. The current plot configuration is displayed in the menu field to the right of the **START PLOT** command.

4. Pick **START PLOT**

2D Access sends the drawing to the spooler, which then queues your plot request and sends your plot data to a plotter.

Note

This part of the tutorial depends on your system configuration. If you think 2D Access isn't plotting correctly, contact your system administrator or refer to chapter 5 to make sure your system is configured correctly.

Dumping a Screen

To print what you see on the screen:

1. Pick **IMAGE COL** from the **PLOT** menu.
2. Pick b & w for a black and white printout, or color if you have a color printer.

2D Access creates a file of the screen display and sends it to a locally connected printer for printing.

Note

This part of the tutorial depends on your system configuration. If you think 2D Access isn't printing correctly, contact your system administrator.

Tutorial Complete

You have now finished the tutorial. If you want to, you can practice some further 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, you need to delete the contents of the viewports:

1. Pick **DEL ALL**.

2D Access prompts you to confirm your decision. A red **CONFIRM** slot now appears in place of the **DEL ALL** slot on the screen menu.

2. Pick **CONFIRM**.

2D Access deletes the contents of all the viewports.

Now you need to reset the system defaults:

1. Pick **RESET**.

2D Access prompts you to confirm your decision.

2. Pick **CONFIRM**.

B

Customizing 2D Access

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2D Access allows full customization to suit your individual requirements. This chapter describes how to:

- Customize the 2D Access Environment.
- Customize screen menus.
- Write macros.

-
- Customize the keyboard function keys.
 - Create and store text fonts.
 - Edit the `startup` file to automatically load any customization.

This chapter also lists the 2D Access command names and keywords.

 **Note**

You need to be fully conversant with 2D Access before you start customizing 2D Access.

When 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 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 2D Access environment. This process is known as customizing.

Customizing the 2D Access Environment

The 2D Access environment consists of such things as 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 a 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 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 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, press [CTRL] [D] to implement all the changes on the system.

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.

Customizing the Screen Menu

Here are some of the ways that you can customize the screen menus:

- New commands can be made available in the existing menus. Add the display text, and insert the new command as the action text.
- You can activate one of your macros from an empty menu slot. Add the display text, and insert the name of your macro as the action text.
- New menus can be created using the existing menu layout.
- New menus can be created using a new menu layout.
- Customized viewport arrangements can be defined and called when wanted.

How Screen Menus are Created

The screen layout consists of a menu from which commands can be given and a working area in which drawings are produced. The purpose of the menu layout is to provide active areas called slots so that commands can be called directly from them. Names and mnemonics associated with these commands can be displayed in the slots. A slot is selected by positioning the cursor on it and pressing the mouse button.

The lines of a typical menu listing are shown below. If you study this listing, you can see that, for each row and column, the "display" text appears first and then the "action text".

```
DEFINE Sm_window_menu_layout
  IF (Data_management_menu_activ)
    LET Data_management_menu_activ 0
    Sm_data_management_reset_main
  END_IF
  LET Hl_active 0
  CURRENT_MENU 'Window_menu'
  Sm_screen_main_use_current
  MENU
  BLACK
  YELLOW '          WINDOW          ' 13 1
  MENU
  WHITE
  BLACK 'FIT' 'Window_fit' 14 1
END_DEFINE
```

The names of the system macros that display the screen menus can be found by entering `EDIT_MACRO` and picking one of the screen menu slots. The display will become as follows:

```
DEFINE action_text
  action
END_DEFINE
```

In this case `action` will be the name of the macro that displays the screen menu. Each of these macros can be edited and stored in a file for future use.

Here is how to save the macro that displays a screen menu. Let's use the `WINDOW` menu again as an example:

1. Enter `EDIT_MACRO` and pick the `WINDOW` slot on the screen menu. You will find that the macro for the `WINDOW` menu is `Sm_window_menu_layout`.
2. Enter `SAVE_MACRO Sm_window_menu_layout 'filename'`.
`filename` is a name of your choice.

Now you can edit the file. You may want to add the contents of the file to your customize file.

Menu Variables

Here are the first few lines of a sample menu, showing the menu variables:

```
DEFINE Sm_window_menu_layout
  IF (Data_management_menu_activ)
    LET Data_management_menu_activ 0
    Sm_data_management_reset_main
  END_IF
  LET Hl_active 0
  CURRENT_MENU 'Window_menu'
  S_clear_menu
  Sm_screen_main_use_current
  MENU
  BLACK
  YELLOW '          WINDOW          ' 13 1
  MENU
  WHITE
  BLACK 'FIT' 'Window_fit' 14 1
```

The meaning of each of these variables is as follows:

- `CURRENT_MENU ' '`
This sets the name of the current screen menu to a default empty string. All menu layouts and menu commands will be related to this current menu, until a current menu having a different name is used.
- `S_clear_menu`
This macro clears all text from the menu, so that new text can be entered.

Physical Layout of Menu Slots

You have probably noticed that, with only a few exceptions, all your screen menus have the same layout of rows and columns. This is because most of the menus use the same "template". This template is created using a macro called `Scr_men_view_layout`. This macro is called once during startup, and is not used again. The macro defines the layout of `CURRENT_MENU` ' Here is a listing of the macro:

The screen consists of a screen menu from which you can pick commands and a viewing area in which you view your drawing.

The screen menu is made up of active areas called `slots`. When you pick a slot the corresponding command is activated. Names and mnemonics related to the command are displayed in each slot.

```
DEFINE Scr_men_layout_view_top
  Headline_height      '          '
  Text_slot_height    '      |      |      '
  Text_slot_height    '      |      |      '
  Text_slot_height    '      |      |      '
  Text_slot_height    '      |      |      '
  Text_slot_height    '  |  |  |  |  |  '
  Text_slot_height    '  |  |  |      '
  Text_slot_height    '      |      |      '
  Text_slot_height    '      |      |      '
  Text_slot_height    '      |      |      '
  Text_slot_height    '      |      |      '
  Text_slot_height    '      |      |      '
END_DEFINE

DEFINE Scr_men_layout_view_body
  Text_slot_height    '          '
  Text_slot_height    '          |      '
  Text_slot_height    '          |      '
  Text_slot_height    '          |      '
  Text_slot_height    '          |      |      '
  Text_slot_height    '          |      |      '
  Text_slot_height    '          |      |      '
  Text_slot_height    '          |      |      '
  Text_slot_height    '          |      |      '
  Text_slot_height    '          |      |      '
  Text_slot_height    '          |      |      '
END_DEFINE

DEFINE Scr_men_layout_view_bottom
  Bottom_headline_height '          '
  Text_slot_height      '      |      |      '
  Text_slot_height      '      |      |      '
END_DEFINE

DEFINE Scr_men_view_layout
  MENU_LAYOUT Menu_position RIGHT
```

```
    Scr_men_layout_view_top
    Scr_men_layout_view_body
    Scr_men_layout_view_bottom
END
END_DEFINE
```

The `MENU_LAYOUT` command causes the menu to be displayed at the right of the screen. The `MENU_LAYOUT` command is terminated by `END`.

Here is an explanation of the variables used in the above macro:

- `Menu_position` This is a macro containing the qualifier `UPPER`. This can be set to `LOWER`, `LEFT`, `RIGHT`, or combinations of these, such as `LOWER LEFT`, if required.
- `Headline_height` This is the height of the menu title slot.
- `Text_slot_height` This is the height of the slots containing the display text.
- `Bottom_slot_height` is the height of the small boxes at the bottom of each menu.

The slot height values are calculated during startup. This allows the menu size to be adapted to suit the window or display size in which 2D Access was started. The values are defined in the `hp_macro` file. These definitions can be used as examples of how the slot sizes can be made self-adapting for different window or display sizes.

Using Tables for Screen Menus

Some of the screen menus are defined using the `TABLE` series of commands, such as `TABLE_LAYOUT` and `SHOW_TABLE`. Tables are used because they allow the display of system status variables. Table scrolling allows long lists to be displayed. The `PLOT CONFIGURATION` menu uses tables. This menu is created using `MENU` and `TABLE` commands.

The definition of these menus can be seen in ASCII versions of the files `hp_macro_s`, and `hp_menu_s`.

If you want to change these menus, use copies of the existing menus to make changes or additions.

Menus that are defined using tables are secured against deletion or overwriting. This avoids potential problems caused by commands such as `DELETE_TABLE_ALL`.

If you want to use your own menu tables, you should move the `SECURE_TABLE` statements in the previously mentioned files to the end of your table definitions. The logical tables for the default menus should not be changed because they are accessed by the 2D Access code.

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, 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 by pressing `[Ctrl] [D]`. The 2D Access screen will then re-appear.

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 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:

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.

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`, 2D Access creates a new macro.

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 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 a 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 [Return] 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 '/users/alan/proj1'"
```

In the above line, 1 is the defined function key and `CURRENT_DIRECTORY` `'/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 '/users/alan/proj1'"
DEFINE_KEY 2 "CURRENT_DIRECTORY '/users/larry/proj2'"
DEFINE_KEY 3 "CURRENT_DIRECTORY '/users/mary/proj3'"
DEFINE_KEY 4 "CURRENT_DIRECTORY '/users/geoff/proj4'"
DEFINE_KEY 5 "CURRENT_DIRECTORY '/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.

1. After entering the command lines, press `[Ctrl] [D]` to store your changes in the file.
2. 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 us 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, but only one of them is the current font and immediately available for 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:	° ±

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` 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 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 2D Access, any customized features that you have entered, although stored in files, are not defined when you start 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 2D Access, you should edit the 2D Access startup file. The startup file contains the commands that load the system files when you start 2D Access. To edit the `/usr/PE/Drafting [version]/startup` file do the following:

1. Enter:

- `EDIT_FILE '/usr/PE/Drafting [version]/startup'` and press [Return]

A listing of the `/usr/PE/Drafting [version]/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 `/usr/PE/Drafting [version]/startup` file automatically loads `customize` which in turn will load the customized files: `Custm_fk` and `Sm_menu_1`.

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 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		
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		
LOAD	plus qualifiers	LOAD
LIST ITEMS	(DETAIL_INFO

Command/Function	Option	Name
SELECT	CATALOG first) (CATALOG first)	SELECT
SORT UP	(CATALOG first)	SORT
SORT DOWN	(CATALOG first)	REVERSE_SORT
CATALOG ITEMS	filenames	"FILE_NAME
name	"PHYS_SIZE	
description	"FILE_DESC"	
type	"FILE_TYPE"	
size	"FILE_SIZE"	
date	"CREATE_DATE"	
date	"MODIFY_DATE"	
date	"ACCESS_DATE"	
access	"NUM_ACCESS"	
CURRENT DIRECTORY COPY 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		

Command/Function	Option	Name
GEN HIDDEN		
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
DIMENSION	on	DIMENSION_ON
HATCH	on	SHOW_HATCH_ON
TEXT	on	SHOW_TEXT_ON
CONSTRUCTION	on	C_GEO_ON
GEOMETRY	on	SHOW GLOBAL GEO ALL
GLOBAL GEO ALL		ON
OFF		
LAYER	on	I_SHOW_LAYER_ON
SPLIT	on	SHOW VERTEX ON
VERTEX OFF		
INFO	on	I_SHOW_INFO_ON

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
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

Command/Functions	Option	Name
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 PARENT SHOW	part	SHOW_PART
ALL boxed	SHOW_PART PART_BOX	
SPOTLIGHT OFF	on	SPOTLIGHT ON
VIEW TOP PARENT	part	VIEW
PART LIST	Count Screen	PARTS_LIST TREE SCREEN
Screen	PARTS_LIST SCREEN	

MEASURE Commands/Functions

Command/Function	Option	Name
DISTANCE	straight	MEASURE_DISTANCE
HORIZONTAL		
VERTICAL		
LENGTH		
ANGLE		
RADIUS		

Command/Function	Option	Name
POINT		
AREA		
AREA PROPERTY		
GET PROPERTY		

CATCH Commands/Functions

Command/Function	Option	Name
ALL	on	CATCH ALL
OFF		
INTERS	INTERSECTION	
PERMANENT INTERSECTION		
CENTER	CENTER	
PERMANENT CENTER		
VERTEX	VERTEX	
PERMANENT VERTEX		
RULER/GRID	GRID	
PERMANENT GRID		
ELEMENT	ELEM	
PERMANENT ELEM		
RANGE	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		
SPACING		
ORIGIN	ON	
PLACE	absolute	CS_SET ABSOLUTE
RELATIVE		
MOVE		
RESET		
ISOMETRIC	top	CS_AXIS ABSOLUTE 30 1 150 1
ABSOLUTE 150 1 90 1		
ABSOLUTE 30 1 90 1		
CURSOR	large	CURSOR LARGE
SMALL		

Command/Function	Option	Name
TURN	absolute	CS_ROTATE ABSOLUTE
	CS_ROTATE RELATIVE	

PLOT Commands

Command	Option	Name
POSITION	cent	Set_sys_plot_center On
Off		
SCALE	fit	Set_sys_plot_plotscale 0
ROTATE ANG	0	Set_sys_plot_rot 0
PI/2		
SIZE	A4	Set_sys_plot_format "A4"
"A3"		
"A2"		
"A1"		
"A0"		
"A"		
"B"		
"C"		
"D"		
"E"		
OUTPUT	all	Set_sys_plot_source All
CURRENT WINDOW'		
CONTENTS	drawing	Set_sys_plot_as_displayed (FALSE)
displayed	Set_sys_plot_as_displayed (TRUE)	
START PLOT		
SCREENDUMP	b & w	Dump_screen_bw
PLOTTERTYP		
PLOT DESTINATION		

2D Access Keywords

Keywords are reserved words within 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:

<code>ABS</code> (Afn)	(Afn)	<code>ARCCOS</code>
<code>ABSOLUTE</code> (Cmd)	(Qua)	<code>ARCHIVE</code>
<code>ADD</code> (Qua)	(Qua)	<code>ARCS</code>
<code>ADD_ATTRIBUTE</code> (Afn)	(Cmd)	<code>ARCSIN</code>
<code>AFFINE</code> (Afn)	(Qua)	<code>ARCTAN</code>
<code>ALL</code> (Fnc)	(Qua)	<code>ARC_RESOLUTION</code>
<code>ALLBASE</code> (Qua)	(Qua)	<code>AREA</code>
<code>AND</code> (Qua)	(Afn)	<code>AREAS</code>
<code>ANG</code> (Cmd)	(Afn)	<code>AREA_PROPERTY</code>
<code>ANG_BISECT</code> (Qua)	(Qua)	<code>ARITHM_FUNCTION</code>
<code>APPEND</code> (Qua)	(Qua)	<code>ARROW_TYPE</code>
<code>AS</code> (Cmd)	(Qua)	<code>CONNECT</code>

AS_DISPLAYED (Fnc)	(Qua)	CONNECT_TABLE
ATTRIBUTE (Qua)	(Qua)	CONTINUE
ATTRIBUTES (Fnc)	(Qua)	CONTROLZ_IS_EOF
AUTO (Fnc)	(Qua)	CONVERT_TO_HP_SYMBOLS
AUTO_NEW_SCREEN (Qua)	(Fnc)	COPY
BACKGROUND_COLOR (Fnc)	(Qua)	COPY_FILE
BACKUP (Afn)	(Cmd)	COS
BALLOON (Fnc)	(Qua)	CREATE_DIRECTORY
BEEP (Cmd)	(Fnc)	CREATE_ELEMENT
BISECT (Fnc)	(Qua)	CREATE_LTAB
BITMAP_TO_FILE (Cmd)	(Fnc)	CREATE_STORAGE_AREA
BLACK (Fnc)	(Fnc)	CREATE_VIEWPORT
BLUE (Fnc)	(Fnc)	CS_AXIS
BORDER_COLOR (Fnc)	(Qua)	CS_MIRROR
BOX (Fnc)	(Qua)	CS_REF_PT
BREAK (Fnc)	(Qua)	CS_ROTATE
CANCEL (Fnc)	(Cmd)	CS_SET
CATALOG (Qua)	(Fnc)	CURRENT
CATALOG_LAYOUT (Fnc)	(Cmd)	CURRENT_DIM_TEXTS
CATCH (Fnc)	(Fnc)	CURRENT_DIRECTORY
CENTER (Fnc)	(Qua)	CURRENT_FONT
CENTER_DASH_DASH (Fnc)	(Fnc)	CURRENT_MENU
CEN_BEG_END (Fnc)	(Qua)	CURRENT_TMENU
CEN_RAD_ANG (Fnc)	(Qua)	CURRENT_VIEWPORT
CHANGE_TABLE_SIZE (Qua)	(Fnc)	CURRENT_WINDOW

CHANGE_VIEWPORT_COLOR (Fnc)	(Fnc)	CURSOR
CHANGE_VIEWPORT_SIZE (Qua)	(Fnc)	CURSOR_COLOR
CHAR_LAYOUT (Fnc)	(Cmd)	CYAN
CHECKIN (Qua)	(Cmd)	C_CIRCLES
CHECKOUT (Fnc)	(Cmd)	C_COLOR
CHECK_ERROR (Qua)	(Afn)	C_GEO
CHECK_WINDOW (Qua)	(Fnc)	C_LINES
CHILD (Fnc)	(Qua)	C_LINETYPE
CHR (Fnc)	(Afn)	DASHED
CIRCLES (Fnc)	(Qua)	DASH_CENTER
CLASS (Afn)	(Qua)	DATE
CLASSES (Cmd)	(Qua)	DB_LOAD
CLEAR (Cmd)	(Qua)	DB_SET_DEFAULT
CLOSE_FILE (Qua)	(Fnc)	DEFAULT
CM (Qua)	(Qua)	DEFAULT_TABLE
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 (Qua)	(Fnc)	FIRST
DISPLAY_NO_WAIT (Qua)	(Fnc)	FIT
DIST_ANG (Fnc)	(Qua)	FOLLOW
DIST_DIST (Qua)	(Qua)	FONT
DIV (Qua)	(Afn)	FORCE
DMS_VERSION (Qua)	(Fnc)	FORMAT
DOTTED (Afn)	(Fnc)	FRACT
DOT_CENTER (Qua)	(Fnc)	VIEW_WIDTH
DOT_GRID (Qua)	(Fnc)	FUNCTION
DOT_TYPE (Qua)	(Qua)	GEO

DOWN (Fnc)	(Qua)	GET_ELEM_INFO
DRAWING_SCALE (Fnc)	(Cmd)	GET_PROPERTIES
DUMP (Qua)	(Qua)	GLOBAL
DUMP_SCREEN (Qua)	(Fnc)	GRAPHIC
DUMP_SCREEN_DEFAULTS (Qua)	(Fnc)	GRD
DYNAMIC (Fnc)	(Qua)	GREEN
ECHO (Qua)	(Fnc)	GRID
EDIT_ENVIRONMENT (Fnc)	(Fnc)	GRID_FACTOR
EDIT_FILE (Qua)	(Fnc)	GROUP
EDIT_MACRO (Qua)	(Fnc)	HATCHING
EDIT_PART (Qua)	(Cmd)	HEIGHT
EDIT_PORT (Fnc)	(Fnc)	HELP
ELEM (Fnc)	(Qua)	HELP_PORT
ELEMENTS (Fnc)	(Qua)	HIGHLIGHT_LTAB
ELEMENTS_TO_FILES (Cmd)	(Qua)	HL_GENERATE_HIDDEN
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 (Qua)	(Fnc)	POLYGON
ON (Fnc)	(Qua)	POP_DOWN_LTAB
ON_ERROR (Fnc)	(Fnc)	POP_UP_LTAB
OPEN_INFILE (Qua)	(Fnc)	PORT
OPEN_OUTFILE (Afn)	(Fnc)	POS
OR (Qua)	(Afn)	PREVIOUS
ORACLE (Fnc)	(Qua)	PRINT_TABLE
ORIGIN (Qua)	(Fnc)	PROMPT
OUTPUT_HP15 (Qua)	(Fnc)	PROMPT_LINE
OUTPUT_HP16 (Fnc)	(Fnc)	PROMPT_LIST

OWNER (Qua)	(Qua)	PSEUDO_COMMAND
PAN (Qua)	(Qua)	PT_ANG
PAN_SEL (Qua)	(Qua)	PT_ANG_DIST
PARALLEL (Fnc)	(Qua)	PURGE_FILE
PARAMETER (Qua)	(Psc)	QUALIFIER
PARENT (Fnc)	(Qua)	QUERY
PART (Fnc)	(Qua)	QUERY_ADD_CLASS
PARTS (Fnc)	(Qua)	QUERY_ADD_COLUMN
PARTS_LIST (Fnc)	(Fnc)	QUERY_ADD_SELECT
PART_BOX (Fnc)	(Qua)	QUERY_ADD_SORT
PASSWORD (Fnc)	(Fnc)	QUERY_AREAS
PERF_MON_OFF (Fnc)	(Fnc)	QUERY_CLASSES
PERF_MON_ON (Fnc)	(Fnc)	QUERY_COPY
PERMANENT (Fnc)	(Qua)	QUERY_CREATE
PERMISSIONS (Fnc)	(Qua)	QUERY_CREATE_CLASSIFICATION
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 (Qua)	(Fnc)	RUBBER_ARC_CEN_BEG
QUERY_USER_TABLE_NAME (Qua)	(Fnc)	RUBBER_ARC_CEN_END
RAD (Qua)	(Qua)	RUBBER_BOX
RANGE (Qua)	(Qua)	RUBBER_CIRCLE_2_PTS
READ (Qua)	(Fnc)	RUBBER_CIRCLE_CEN
READ_FILE (Qua)	(Fnc)	RUBBER_LINE
READ_LTAB (Qua)	(Afn)	RUBBER_LINE_ANG
READ_ONLY (Qua)	(Qua)	RUBBER_LINE_HORIZONTAL
RECALL_BUFFER (Qua)	(Fnc)	RUBBER_LINE_VERTICAL
RECALL_WINDOW (Fnc)	(Fnc)	RULER

RECALL_WINDOWS (Fnc)	(Qua)	RUN
RECTANGLE (Qua)	(Qua)	SAME
RECURSIVE (Fnc)	(Qua)	SAVE_ENVIRONMENT
RECURSIVE_TO (Fnc)	(Qua)	SAVE_LTAB
RED (Fnc)	(Fnc)	SAVE_MACRO
REDRAW (Fnc)	(Fnc)	SAVE_MENU
REF_PT (Fnc)	(Qua)	SAVE_TABLE
RELATE (Fnc)	(Cmd)	SAVE_TMENU
RELATED_CLASS (Fnc)	(Qua)	SAVE_VIEWPORT
RELATED_CLASSES (Qua)	(Qua)	SCALE
RELATIVE (Qua)	(Qua)	SCREEN
RELEASABLE (Fnc)	(Qua)	SCREEN_TRANSFORMATION
RELEASE (Qua)	(Cmd)	SCROLL_BAR
REMOVE (Fnc)	(Cmd)	SCROLL_LTAB
REMOVE_QUERY_PAGE (Cmd)	(Cmd)	SEARCH
REMOVE_USER (Fnc)	(Cmd)	SECURE_LTAB
RENAME_ELEMENT (Fnc)	(Cmd)	SECURE_MACRO
RENOVATE (Fnc)	(Fnc)	SECURE_TABLE
SELECT (Qua)	(Qua)	TAN_PT
SELECT_FROM_LTAB (Qua)	(Fnc)	TAN_PT_PT
SELECT_KNOB (Fnc)	(Qua)	TB_GET_DRAW_VALUES
SELECT_PORT (Fnc)	(Qua)	TB_SETUP_TABLE
SEL_OLD (Fnc)	(Qua)	TECHO
SGN (Cmd)	(Afn)	TEXT

SHARED_PART (Qua)	(Qua)	TEXTS
SHOW (Fnc)	(Fnc)	TEXT_ADJUST
SHOW_PART (Fnc)	(Fnc)	TEXT_ANGLE
SHOW_TABLE (Qua)	(Fnc)	TEXT_ATTR
SHOW_TABLE_PAGE (Qua)	(Fnc)	TEXT_COLOR
SIMILAR (Fnc)	(Qua)	TEXT_FILL
SIN (Qua)	(Afn)	TEXT_FORMAT
SLASH_TYPE (Fnc)	(Qua)	TEXT_FRAME
SMALL (Fnc)	(Qua)	TEXT_LINESPACE
SMOOTH (Fnc)	(Qua)	TEXT_RATIO
SNID (Fnc)	(Afn)	TEXT_SIZE
SOLID (Fnc)	(Fnc)	TEXT_SLANT
SORT (Qua)	(Qua)	THREE_PTS
SORT_LTAB (Afn)	(Fnc)	TIME
SPLINES (Fnc)	(Qua)	TINPUT
SPOTLIGHT (Qua)	(Fnc)	TITLE
SQL_QUERY (Qua)	(Fnc)	TITLE_LAYOUT
SQR (Fnc)	(Afn)	TMENU
SQRT (Fnc)	(Afn)	TONE
STATLINE_RESET (Qua)	(Fnc)	TOP
STORE_MACRO (Fnc)	(Fnc)	TRACE
STORE_WINDOW (Qua)	(Fnc)	TRACKING
STORE_WINDOWS (Fnc)	(Qua)	TRAP_ERROR
STR (Qua)	(Afn)	TREE
STRING (Qua)	(Qua)	TRIANGLE_TYPE

STRUCTURE (Afn)	(Fnc)	TRIM
STRUCTURES (Afn)	(Qua)	TRUE
SUBPART (Afn)	(Qua)	TRUNC
SUBSTR (Qua)	(Afn)	TWO_PNT
SUBTRACT (Qua)	(Qua)	TWO_PTS
SYMBOL (Afn)	(Qua)	TYPE
TABLE (Qua)	(Qua)	UINCHES
TABLET_RESET (Qua)	(Fnc)	UM
TABLE_COLUMN (Qua)	(Fnc)	UNDO
TABLE_LAYOUT (Fnc)	(Cmd)	UNITS
TABLE_SCROLL_STEP (Cmd)	(Fnc)	UNRELEASE
TABLE_TITLE (Cmd)	(Fnc)	UNRESERVE
TAN (Psc)	(Afn)	UNTIL
TAN2 (Qua)	(Qua)	UP
TAN2_PT (Afn)	(Qua)	UPC
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

C

Unicode

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Introduction

Creo Elements/Direct Drafting's different internal and external encoding schemes were Unicode incorporated for version 2007. Unicode allows text and symbols from all languages to be consistently represented and manipulated by computers.

All textual and string data processed by Creo Elements/Direct Drafting is held internally in Unicode (UTF-8) encoding. That means most of Creo Elements/Direct Drafting's functionality processes user input from almost any language, and string data from different languages can be mixed. For example, it is now possible to have German, Japanese, and Russian characters within a single string, such as a part name.

Drafting's font list contains all the Creo Elements/Direct true type fonts that are dynamically converted to internal font format when referenced. An appropriate font that contains all desired characters must be selected to display texts and dimension texts correctly. Fonts containing characters from the whole Unicode range are relatively rare, and they usually contain the word Unicode in their name (for example, Arial Unicode MS).

Symbols are a special Creo Elements/Direct Drafting feature and display with `hp_symbols` and `hp_symbols2` fonts. They can still be composed in old-style escape sequences (`#15#XY#16`) but for compatibility with Unicode are mapped to a private-use Unicode area [E000-E1FF]:

- `15#XY#16` → `0xE000 + XY`
- `30#XY#31` → `0xE100 + XY`

MI file format

Version 3.20 MI files are encoded in Unicode (UTF-8). Files stored to older MI versions are encoded as before, depending on the current locale. When loading older MI formats, the Creo Elements/Direct Drafting loader uses the information recorded in the info section (~1) of the MI file. MI formats without encoding information in their info sections are treated as if encoded in ROMAN8 or SJIS (Japanese locale).

Macro file encoding

Old macro files did not contain any encoding information. They were assumed to be encoded for current locales (e.g. ROMAN8 for English and SJIS for Japanese locales). However, this causes problems when using the same macros in different locales.

Version 2007 supports macro Unicode but maintains backward compatibility. We recommend you change old macro files such that their encoding is explicitly specified with the `DEFINE_ENCODING` directive.

Text macro files containing Unicode UTF-8 or UTF-16 BOMs (Byte Order Markers), supported by most popular Windows text editors, are read as Unicode.

Files containing the `DEFINE_ENCODING` directive are always read in the specified encoding. Otherwise they are treated as old macro files and encodings are locale dependent:

- SJIS for Japanese
- BIG5 for traditional Chinese
- GB2312 for simplified Chinese
- ROMAN8 for all else

Limitations

The displaying of Unicode texts is limited. Features such as combining characters, Arabic cursive connection, and bidirectional algorithm are not fully implemented.

`CHR`, `NUM` and `#` still work in old locale-dependent internal encodings (ROMAN8 & SJIS). They return different results in different locales. They are obsolete; their use is only for maintaining backward compatibility. For example, `((CHR 154)+(CHR 223))` or its equivalent `(#154#223)` displays a Japanese character with Unicode code 22756 (hex 58E4) on a Japanese locale and two Latin characters on other locales. To avoid problems and to display the same character regardless of locale, use Unicode versions of these functions: `(UCHR 22756)`, `(UCHR '58E4')`, `#u22756`, or `#ux58E4`.

Combinations of `#` and `CHR` to compose 2-byte characters and symbols are not allowed. For example, `((CHR 154)+#223)` does not work.

Strings that begin with `(CHR 255)` are icons and are not encoded in UTF-8. Pre-2007 CoCreate Drafting versions let you compose icons using a large variety of combinations (for example `#255'abc'#212+CHR 12`). In version 2007, only `CHRs` can be used to compose icons.

MI files of versions prior to 2.90 that do not contain encoding information in their info section (~1) load correctly only if loaded in the same locale as they were drawn. For example, you have 2 options to correctly load a Japanese MI file of version 2.80 in English Creo Elements/Direct Drafting:

1. Load the file in japanese locale and store it back in new format:
 - `CHANGE_LOCALE 'ja'`
 - `LOAD 'japanese.mi'`
 - `STORE MI ALL DEL_OLD 'japanese.mi'`
 - `CHANGE_LOCALE 'en'`
2. Load the MI file in a text editor and manually add section ~1 at the beginning of MI file:
 - `#~1`
 - `ENCODING: SJIS`

-
- Such files load correctly in any locale.

AI 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 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), 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`.

True Type fonts converted to CoCreate Drafting fonts contain all defined characters. Separate symbol fonts are used

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