Pro/ENGINEER® Wildfire® 4.0

Detailed Drawings
Help Topic Collection

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Detailed Drawings

Detailed Drawings Overview

The Detailed Drawings Help describes how to pass the dimensions, notes, GTOLs, and other detailing items from the 3D model directly to views on the plotted sheet in Pro/ENGINEER. You can prepare drawings of parts and assemblies to pass information forward and backward in the design workflow.

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Detailed Drawings Basics

The Pro/ENGINEER Drawing Modes

Pro/ENGINEER offers functionality for working with engineering drawings in Drawing mode and in the Detailed Drawings module.

Drawing Mode

Using the Pro/ENGINEER Drawing mode, you can create drawings of all Pro/ENGINEER models, or import drawing files from other systems. You can annotate the drawing with notes, manipulate the dimensions, and use layers to manage the display of different items. All views in the drawing are associative: if you change a dimensional value in one view, the system updates other drawing views accordingly. Moreover, Pro/ENGINEER associates drawings with their parent models. The model automatically reflects any dimensional changes that you make to a drawing. In addition, corresponding drawings also reflect any changes that you make to a model (such as the addition or deletion of features and dimensional changes) in Part, Sheetmetal, Assembly, or Manufacturing modes.

Detailed Drawings Module

Detailed Drawings extends the drawing capability of Drawing mode. You can use it with Pro/ENGINEER to create, view, and annotate models and drawings.

Detailed Drawings supports additional view types and multisheets, offers numerous commands for manipulating items in a drawing, and lets you add and modify different kinds of textual and symbolic information. In addition, you can use it to customize engineering drawings with sketched geometry, create custom drawing formats, and make multiple cosmetic changes to drawings.

With Detailed Drawings, you can also use a shortcut menu to modify any object in a drawing from anywhere in the Model Tree. At any time when a drawing window is active, you can interrupt your current process and activate a drawing object for modification.

With Pro/ENGINEER Interface or Detailed Drawings, you can access various interface commands for exporting drawing files to other systems and importing files into Drawing mode.

Selecting Objects in Drawings

To perform operations within your drawings you must know to select objects. You can use preselection highlighting, drawing object filters, and several selection methods.

Preselection Highlighting

Preselection highlighting allows you to visually confirm the item that will be selected before you select it. By default, as the pointer passes over an object in the graphics area, the object is highlighted. The object's name is displayed in a tooltip on the drawing and on the status bar in lower left-hand corner of the application. For example, `d92: F9(HOLE)` displays to indicate that the pointer is over the diameter dimension for the ninth model feature, a hole.

Drawing Object Filters

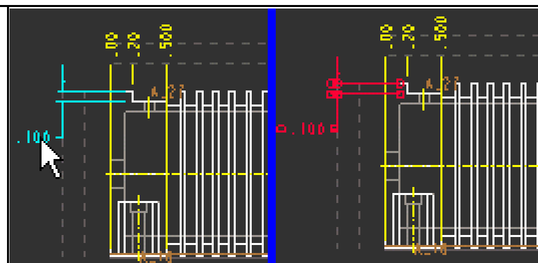
You can limit which objects are prehighlighted and, ultimately, what is selectable using the drawing filter. Filters allow you to change the type of entities you can select, which simplifies graphic selection. So, when cleaning up the placement of dimensions, you can set the filter to make only dimensions available for selection.

The filter is a drop down list on the right side of the status bar. While an object is designated in the filter, all other object types are disallowed from selection.

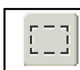
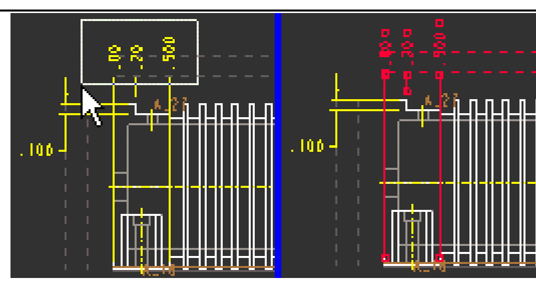
Methods of Selecting

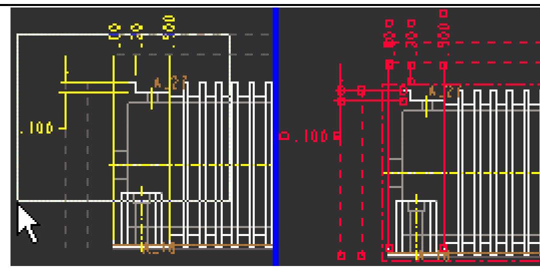
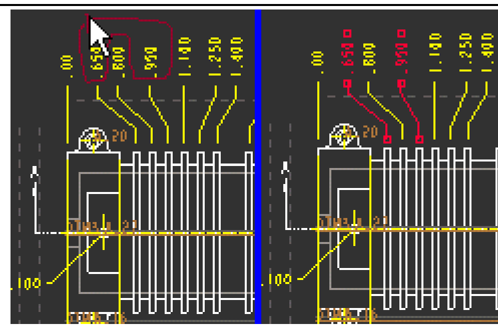
Depending on the type of operation you are completing, you may need to select a specific drawing object, or even multiple object types simultaneously. The following selection methods enable you to efficiently select the objects:

- **Individual**—Select drawing items individually. Move the pointer over the desired drawing element and click to select.

Note: By holding down CTRL you can select multiple elements, one at a time.	

- **Region**—Select multiple elements simultaneously without pressing CTRL. Select the appropriate region selection tool (**Edit** > **Select** > **Preferences**), then drag to designate the area in which to select items. Generally, only top-level items are selected. For example, you can select a note but not the note text. The following region selection tools are available:

	**Inside Box** Selects drawing items that lie completely inside the rectangular region.	 Note: Only the items entirely enclosed in the box are selected.

	**Across Box** Selects drawing items that partially or lie completely inside the rectangular region.	 Note: In this example, both the dimensions and the drawing view are selected because the region selection box crossed the edge of the drawing view.
	**Inside Polygon** Selects drawing items that lie completely inside the polygon region.	 Note: Only the items entirely enclosed in the polygon region are selected. Drawing views and tables (including cells, rows, and columns) cannot be selected.

- **Model Tree**—Select the feature or part from the model tree rather than identifying the element in the graphics pane.

Note: When the `selection_of_removed_entities` configuration file option is set to `yes`, you can select entities and activate them for performing such actions as showing dimensions and placing items on a layer.

You can also activate entities for choosing features that are in front of a cross section (planar or offset), have been removed via Z-Clipping, or have been erased using **Erase Line** in the **EDGE DISP** menu.

About Cut, Copy and Paste

When you cut or copy items, they are copied temporarily to a clipboard. They can then be pasted onto the same sheet, a different sheet, or into a different drawing.

Parameters in notes are copied as parameters and are evaluated on the target drawing. If a parameter cannot be evaluated on the target drawing, three asterisks are displayed to indicate this.

Tables with regular text (no repeat regions) are copied as text. When pasted, the text looks as it does on the source drawing. Tables with repeat regions are copied with the report symbol names and have to be updated on the target drawing using **Update Tables**.

Rules for Copying Between Drawings

The following rules and restrictions apply to copying detail items from drawing to drawing:

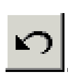
- The source and target drawings must use the same units.
- Notes, balloons, and symbols with leaders are copied with their leader. The leader is copied as it looks on the source drawing. You can move the end of the leader on the target drawing by dragging its endpoint and jog to the desired locations.
- Notes that contain dimension parameters (that is, &d23) cannot be copied. Notes that contain other parameters can be copied.
- Tables are copied as follows:
    - Tables without repeat regions are copied as text and look the same as they do on the source drawing.
    - Tables with repeat regions are copied with the report symbol names and have to be updated on the target drawing using **Update Tables**.

Undoing and Redoing Drawing Operations

As you dimension and detail your drawings you can undo and redo some general operations. Such flexibility helps to ensure that the appropriate action is taken and also allows you to further explore some of the detailing capabilities.

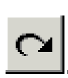
To Undo a Drawing Operation

After completing the operation

- Click .
- Press `CTRL+Z`.
- Click **Edit > Undo <**`command name`**>**.

To Restore a Drawing Operation

After undoing the operation

- Click .
- Press `CTRL+Y`.
- Click **Edit > Redo <**`command name`**>**.

You can only undo and redo some of the top level operations. For example, you can undo or redo the entire show and erase operation, but not the individual show and erase operations while the dialog is open.

Note:

Undo and Redo do not preserve the revision number of the object. For example, if an associative part dimension is created in the drawing, and you perform undo on dimension creation, the part and drawing are still considered modified.

Any command that changes the active model (this can be done with, **File** > **New**, **File** > **Open**, **File** > **Close Window**, **File** > **Erase** > **Current**, **File** > **Delete** > **[current]**, **File** > **Exit**, **Window** > **Activate**, **Window** > **Close**) will clear the stack . However, these commands can still be invoked and not change the active model, therefore will not clear the stack in that case. Such an example would be if multiple windows of the same drawing sheet were open, and **Window** > **Activate** was used to activate one of these windows.

Any command that changes both the active window and the current drawing page number will clear the stack.

Previewing Drawings

About View-Only Mode

If you simply want to view or quickly check a drawing, you can reduce the amount of time it takes to retrieve it by opening it in *View-Only* mode.

In *View-Only* mode, the system does not retrieve any of the associated model files when it opens the drawing. However, since the solid models are not in session, the system temporarily freezes the drawings, so you cannot modify them.

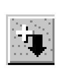
A prerequisite to using view only mode is that the configuration option `save_display` is set to `yes` the last time the file is saved in normal mode. This lets the drawing file "remember" the 3D dimension locations and values without having to access the 3D model files.

If you decide that you do need to modify a drawing in view only mode, you can use **File** > **Retrieve Models** to retrieve all of the models in mid-session.

Note:

- If the geometry display information of a view is missing, the system displays an empty view boundary.
- Since Pro/ENGINEER does not retrieve any of the associated solid models, plotting (for example, complex overlap checking) does not function the same way that it does in Drawing mode because much of the information is missing.
- If you store the display with snap lines in Drawing mode, the system plots them in View-Only mode.

To Open a Drawing in View-Only Mode

1. Click **File** > **Open**; then click  and check the **Retrieve Drawing as View Only** option from the list.
2. Open the drawing.

Note:

Since View only mode does not access the 3D models, you must save the existing display information in normal mode by

- Checking **Save Display** in the **Tools** > **Environment** dialog box, or,
- Setting the configuration file option `save_display` to `yes`.

To Modify a Drawing in View-Only Mode

1. Click **File** > **Retrieve Models**.
2. Click **Confirm** in the Menu Manager. The system enters Drawing mode and retrieves all models for the current drawing.
3. Modify the current drawing as necessary.

Setting Up and Configuring Drawings

About Setting Up a Drawing

You can customize your drawing environment and drawing behavior using a combination of:

- Drawing Setup File Options
- Configuration Options
- Templates
- Formats

For example, you may predetermine characteristics like the height of dimension and note text, text orientation, geometric tolerance standards, font properties, drafting standards, and arrow lengths.

Using True Type Fonts

You can use TrueType fonts with Pro/ENGINEER. Pro/ENGINEER provides 42 TrueType fonts. You can purchase additional TrueType fonts and place them in the Pro/ENGINEER loadpoint directory.

See the *Pro/ENGINEER Installation and Administration Guide* for information about the location of the font file in the load directory.

Note: TrueType fonts can take more time to repaint on the graphics window.

UNICODE Support

You can represent and manipulate out-of-locale text and symbols in drawings consistently in the Pro/ENGINEER graphics window using TrueType or OpenType fonts. To do so, ensure that you perform the following steps:

- Install the necessary TrueType or OpenType fonts on computers where drawings containing text and symbols from multiple locales are retrieved.
- Specify the location of these fonts using the `PRO_FONT_DIR` configuration option.

Drawing Setup File Options

About Drawing Setup File Options

While configuration file options control the design environment for parts and assemblies, drawing setup file options add additional controls to the detailing environment. The drawing setup file options determine characteristics like the height of dimension and note text, text orientation, geometric tolerance standards, font properties, drafting standards, and arrow lengths.

Default values are provided for the drawing setup file options, but you can customize and save various versions for use in other drawings. The file that you specify in the `drawing_setup_file` configuration option establishes the default drawing setup options for any drawing that you create during a Pro/ENGINEER session. If you do not set this option, Pro/ENGINEER uses the default drawing setup file option values. You can retrieve the following sample drawing setup files from the `<loadpoint/text>` directory with the following names:

- `iso.dtl` ISO (International Organization for Standardization)
- `jis.dtl` JIS (Japanese Institute of Standards)
- `din.dtl` DIN (Deutsches Institut für Normung / German Institute for Standardization)

Pro/ENGINEER saves the drawing setup file options settings with each individual drawing file. Pro/ENGINEER saves the values in a setup file named `<filename>.dtl`. The location of this file is determined by the `pro_dtl_setup_dir` configuration file option. You can specify the complete path to the directory that contains your drawing setup files. If you do not specify the pathname using this configuration file option, Pro/ENGINEER brings you into the default setup directory.

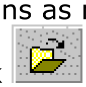
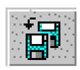
Note:

- If a drawing setup file option is set to `default`, and if a configuration file option exists with the same name the drawing setup file option refers to the configuration setting. If you set the drawing setup option to something other than default, the configuration option setting is overwritten within the drawing environment only.
- The environment settings supersede configuration file settings (**Tools > Environment**). Therefore, if you seldom use a particular command, such as the drawing grid, you could use the configuration file option to keep it cleared and then set the environment as needed.
- The Detailed Drawings Help lists all the drawing setup file options in alphabetical order. Each topic contains the following information:
  - Drawing setup file option name.

- Brief description and notes describing the drawing setup file option.
- Default and available variables or values. All default values are followed by an asterisk (*).

To Customize the Drawing Setup File Options

You can customize existing drawing setup files or you can create new ones based on your detailing requirements.

1. Click **File** > **Properties**. The **File Properties** menu appears on the menu manager.
2. Click **Drawing Options**. The **Options** dialog box opens.
3. Edit the drawing setup file options as necessary. If you are customizing an existing drawing setup file, click  and browse to the appropriate DTL file.
4. Click **Apply** after making your edits. The system updates the drawing using the new setup file option values.
5. To save your changes, click  and then type a name for the file.

Note: If the newly set options do not immediately update on your drawing, do one of the following:

- Click **View** > **Repaint**
- Click the **Redraw** button
- Click **View** > **Update** > **Current Sheet** or **All Sheets**, and then use the **GET SELECT** menu to select the view you want to update.

To Retrieve an Existing Drawing Setup File

You can retrieve and apply an existing drawing setup file to a drawing. Several standard drawing setup files are available from the <loadpoint/text directory>.

1. With the drawing file open, click **File** > **Properties**. The **File Properties** menu appears in the menu manager.
2. Click **Drawing Options**. The **Options** dialog box opens.
3. Click  and browse to the desired drawing setup file. Drawing setup file options have a DTL extension.
4. Click **Apply**. The drawing setup file options are applied to the current drawing.

2d_region_columns_fit_text

Determines whether each column in a two-dimensional repeat region automatically resizes to fit the longest piece of text in each column, and does not overlap adjacent columns or force large gaps in the table.

Columns with no text in them use the default column width for the region (the width of the template cell).

Default and Available Settings

- **yes**—Resizes each column in 2-D repeat regions to fit the longest piece of text.
- **no***—Column maintains the same size.

Note: Columns of tables that contain automatically resized two-dimensional repeat regions cannot be manually resized.

allow_3D_dimensions

Determines if dimensions are shown in isometric views.

Default and Available Settings

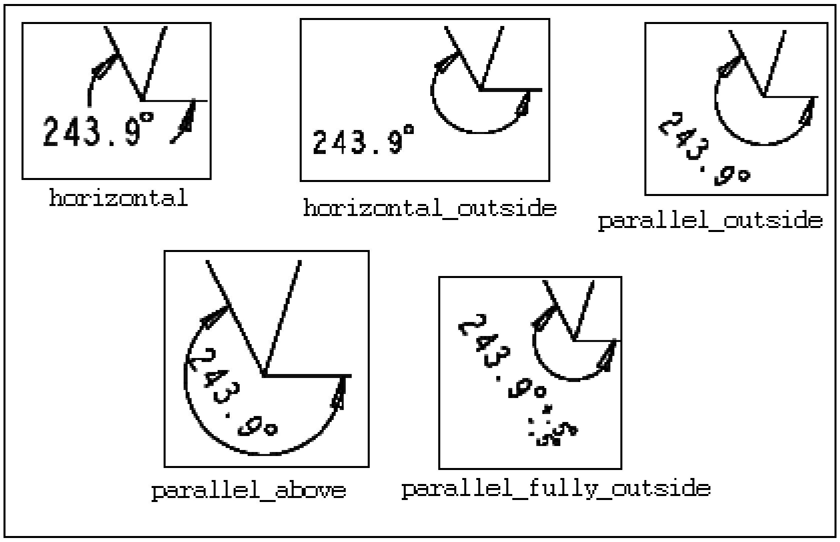
- **no***—Dimensions are not shown in isometric views.
- **yes**—Dimensions display in isometric views.

angdim_text_orientation

Controls the placement of angular dimensions within drawings.

Default and Available Settings

- **horizontal***—Displays text of angular dimensions horizontally at all times, centered between the leaders (equivalent to the value "horizontal" for the drawing setup file option "text_orientation").
- **parallel_outside**—Displays text parallel to the leader lines, regardless of their orientation (equivalent to the value "parallel" for the drawing setup file option "text_orientation").
- **horizontal_outside**—Displays text horizontally outside the dimension.
- **parallel_above**—Displays text parallel to the dimension arc, but above it.
- **parallel_fully_outside**—Displays text of angular dimensions (with a plus/minus tolerance) parallel to the leader lines.



asme_dtm_on_dia_dim_gtol

Controls the placement of a set datum attached to a diameter dimension.

Default and Available Settings

- **on_gtol***—Places the set datum on the gtol according to the ASME standard.
- **on_dim**—Attaches the set datum to the diameter dimension.

associative_dimensioning

Associates draft dimensions to draft entities. The system associates only dimensions that you create while you have this set to "yes".

Default and Available Settings

- **yes***—Associates draft dimensions to draft entities.
- **no**—Does not associate draft dimensions to draft entities.

aux_line_font

Enables the use of line fonts in multiple drawings. The line font <fontname>.lsl file must be in the working directory for aux_line_font to function.

This setup option is not visible in the Options dialog unless you:

- Create a new line font within a drawing (the option is automatically set).
- Manually type and set the option.

Default and Available Settings

- **# font name**—Where `#` represents a value between 1 and 10,000 and `font name` represents the auxiliary font you want in the drawing.

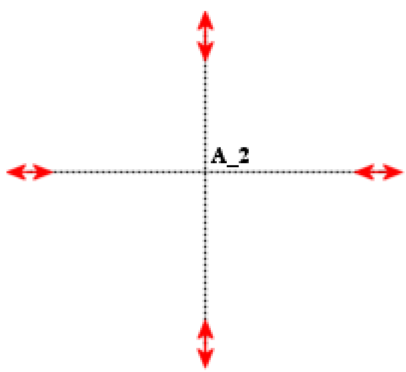
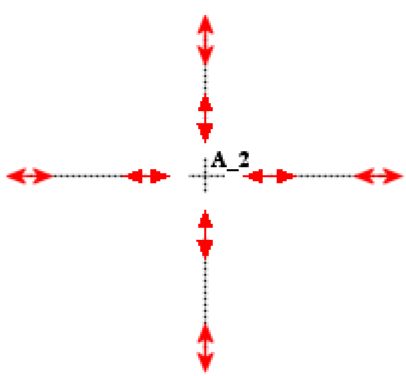
Note: You can not delete additional aux_font lines from a `DTL` file because auxiliary fonts might also be used within drawing notes. Instead, you should replace the old font name with a different font. The integer before the font name serves as a cross-reference between the geometry and the line font, enabling you to make blanket changes to fonted geometry. For example:

Replacing the value of `aux_line_font` from `100 dash-1` to `100 solidfont` changes all geometry that was originally `dash-1` to `solidfont`.

axis_interior_clipping

Determines the ability to clip (or drag) the interior ends of an axis.

Default and Available Settings

no*	Only allows clipping of external axes ends. 
yes	Enables clipping of axes from both internal and external ends. 

axis_line_offset

Sets the default distance that a linear axis extends beyond its associated feature.

Default and Available Settings

- **0.100000***

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `axis_line_offset` drawing setup file option.

blank_zero_tolerance

Controls the display of a plus or minus tolerance value.

Default and Available Settings

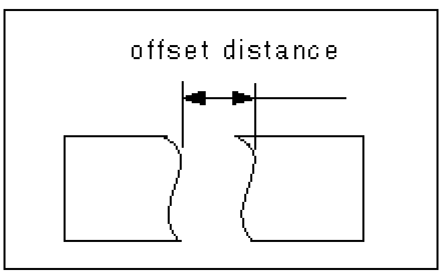
- **no***—Displays a plus or minus tolerance value if you set the tolerance value to zero.
- **yes**—Does not display a plus or minus tolerance value if you set the tolerance value to zero.

broken_view_offset

1.000000, value

Sets the offset distance between the two halves of a broken view.

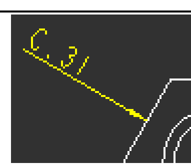
Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `broken_view_offset` drawing setup file option.

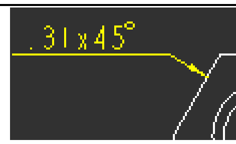


chamfer_45deg_dim_text

Controls the display of 45-degree chamfer dimensions in drawings. This only affects the text of newly created chamfer dimensions.

Default and Available Settings

Value	Display when value is applied
jis*	

Value	Display when value is applied
iso/din	

Note: This drawing setup file option does not affect the three-dimensional model.

chamfer_45deg_leader_style

Controls the leader type of a chamfer dimension without affecting the text.

Default and Available Settings

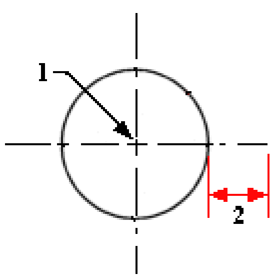
- **std_asme_ansi***—Applies American Society of Mechanical Engineers (ASME)/American National Standards Institute(ANSI)
- **std_din**—Applies Deutsches Institut für Normung (DIN) (German Institute for Standardization)
- **std_iso**—Applies International Organization for Standardization (ISO)
- **std_jis**—Applies Japanese Industrial Standard (JIS)

circle_axis_offset

1.100000

Sets the default distance that a circular cross-hair axis extends beyond the circular edge.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `circle_line_offset` drawing setup file option.



1. Circular cross-hair axis

2. The distance at which the circular cross-hair axis extends beyond the circular edge. This distance is specified by the circle_axis_offset drawing setup file option

clip_diam_dimensions

Automatically clips the diameter dimensions at the view border. Dimension endpoints outside of the view border are clipped to the view border. No clipping occurs when both endpoints are inside view border.

Default and Available Settings

- **yes***—Automatically clips the diameter dimensions at the view border.
- **no—**Does not automatically clips the diameter dimensions at the view border.

Note: The default setting is `no` for drawings created in earlier releases.

clip_dim_arrow_style

`double_arrow`, `arrowhead`, `dot`, `filled_dot`, `arrow`, `slash`, `integral`, `box`, `filled_box`, `target`, `none`

Controls the arrow style of clipped dimensions.

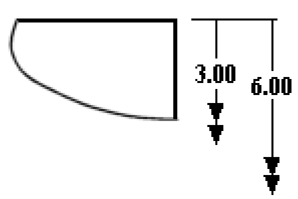
Note: The arrow style of the clipped dimensions changes only after you click **Apply** in the **Options** dialog box.

clip_dimensions

Controls the display of dimensions in a detailed view.

Default and Available Settings

- **yes***—Does not display dimensions completely outside of a detailed view boundary; shows dimensions that cross a detailed boundary with a double arrow.
- **no**—Displays all dimensions.

	When set to `yes`, dimensions that cross a detailed boundary display with a double arrow.

create_area_unfold_segmented

Makes the display of dimensions in area unfolded cross-sectional views similar to those in total unfolded cross-sectional views.

This option only affects new views. Pro/ENGINEER does not support detailed views of segmented area unfolded cross-sectional views or total unfolded cross-sectional views. However, it does support detailed views of non-segmented area unfolded cross-sectional views.

Default and Available Settings

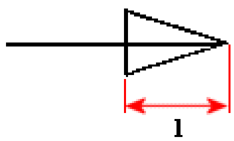
- **yes***—Displays the view in segments when creating a new view—one piece at a time—corresponding to the straight segments of the cross-sectional sketch.
- **no**—Does not display the view in segments when creating a new view.

Note: To draw view borders between view segments, set `show_total_unfold_seam` to `yes`.

crossec_arrow_length

0.187500

Sets the length of the arrowhead for cross-section cutting plane arrows.



1. The length of the arrowhead as specified by the `crossec_arrow_length` drawing setup file option

Note: The `drawing_units` drawing setup file option determines the measurement unit for the `crossec_arrow_length` drawing setup file option.

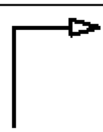
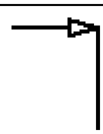
crossec_arrow_style

tail_online, head_online

Determines which end of the cross-section arrow touches the cross-section line.

`tail_online`—The tail of the cross-section arrow touches the cross-section line

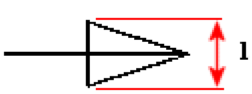
`head_online`—The head of the cross-section arrow touches the cross-section line

tail_online*	
head_online	

crossec_arrow_width

0.062500

Sets the width of the arrowhead on the cross-section cutting plane arrows.



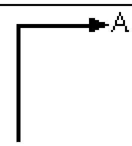
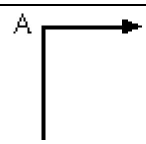
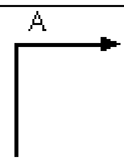
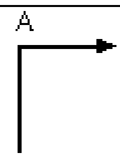
1. The width of the arrowhead as specified by the `crossec_arrow_width` drawing setup file option

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `crossec_arrow_width` drawing setup file option.

crossec_text_place

Sets the location of cross-section text.

Default and Available Settings

after_head*	
before_tail	
above_tail	
above_line	
no_text	Does not display any cross-section text.

crossec_type

Improves the ability to create complicated cross section views in drawings, and reduces or eliminates the number of occurrences when a cross section view cannot be created.

Default and Available Settings

- **old_style***—The system uses a cut to remove geometry to create the cross section view.
- **new_style**—The system uses a z-clipping plane to create the cross section view.

cutting_line

Controls the standard for the cutting line display.

Default and Available Settings

- **std_ansi***—Uses the ANSI standard for cutting lines.
- **std_din**—Displays the thickened portion of the cutting line in white, and displays its thin portion in gray.
- **std_iso**—Uses the ISO standard for cutting lines.
- **std_jis**—Uses the JIS standard for cutting lines.
- **std_ansi_dashed**—Displays the cutting line as dashed lines.
- **std_jis_alternate**—If set to "std_jis_alternate" and the drawing setup file option "cutting _line_segment" is set, displays view arrows as follows:
    - The thickened portion of the cutting line forms an angle and the system displays it in blue.
    - Displays the connecting portions of the cutting line segment between thickened segments in yellow.
    - Displays arrow portions in white.

  If you set the `cutting_line_segment` drawing setup file option to `0`, Pro/ENGINEER displays the entire cutting line as a dashed yellow line. If the length of a cutting line segment is too large, the entire cutting line displays in blue.

cutting_line_adapt

Controls display of line fonts used to show cross-sectional arrows.

Default and Available Settings

- **no***—All line fonts do not display adaptively.
- **yes**—All line fonts display adaptively, beginning in the middle of a complete line segment and ending in the middle of a complete line segment.

Note: When plotting, the section cutting line overlaps the dimension witness and leader lines.

cutting_line_segment

Specifies the length in drawing units of the thickened portion of a non-ANSI cutting line.

Default and Available Settings

- **0.000000***—The default length of the cutting line segment is 0. You can specify a different value, if required.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `cutting_line_segment` drawing setup file option.

dash_supp_dims_in_region

Controls display of dimension values in Pro/REPORT table repeat regions.

Default and Available Settings

- **no***—Displays the values in Pro/REPORT table repeat regions.
- **yes**—Suppresses the dimension and displays a dash instead.

datum_point_shape

cross, dot, circle, triangle, square

Determines the display of datum points.

datum_point_size

Controls size of model datum points and sketched two-dimensional points.

Default and Available Settings

- **default***—Reverts to the setting established in the drawing_units setup file option; which has a default setting of inches. You can type a value for the datum point size.

Note: The drawing_units drawing setup file option determines the unit of measurement for the datum_point_size drawing setup file option.

decimal_marker

comma_for_metric_dual, period, comma

Specifies the character to be used as the decimal point in secondary dimensions.

def_bom_balloon_leader_sym

Sets the default arrow (attach point) style for BOM balloons in reports.

Default and Available Settings

- **arrowhead***
- **dot**
- **filled_dot**
- **no_arrow**
- **slash**
- **integral**
- **box**

- **filled_box**

def_bom_balloons_attachment

Sets the default attachment method for BOM balloons.

Default and Available Settings

- **edge***—BOM balloons point to component edges.
- **surface**—BOM balloons point to component surfaces.

For records of components that represent bulk items and included item components, the default attachment method is set.

def_bom_balloons_edge_att_sym

Controls the default leader head when BOM balloons are attached to edges.

Default and Available Settings

- **arrowhead***
- **dot**
- **filled_dot**
- **no_arrow**
- **slash**
- **integral**
- **box**
- **filled_box**
- **target**

def_bom_balloons_snap_lines

Determines whether snap lines are created around the view when showing BOM balloons.

Default and Available Settings

- **no***
- **yes**

def_bom_balloons_stagger

Determines whether BOM balloons are displayed in a staggered manner by default.

Default and Available Settings

- **no***
- **yes**

def_bom_balloons_stagger_value

Controls the distance between consecutive offset lines when BOM balloons are displayed in a staggered manner.

Default and Available Settings

- **0.600000***

def_bom_balloons_surf_att_sym

Controls the default leader head when BOM balloons are attached to surfaces.

Default and Available Settings

- **integral***
- **arrowhead**
- **dot**
- **filled_dot**
- **no_arrow**
- **slash**
- **box**
- **filled_box**
- **target**

def_bom_balloons_view_offset

Controls the default offset distance from the view boundaries on which to show BOM balloons.

Default and Available Settings

- **0.800000***

def_view_text_height

0.000000

Sets the text height in view names used in view notes, and in arrows in cross-sectional and projection detail views.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `def_view_text_height` drawing setup file option.

def_view_text_thickness

0.000000

Sets default thickness for new text in view names used in view notes and in arrows in newly created cross-sectional and projection detail views.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `def_view_text_thickness` drawing setup file option.

default_dim_elbows

Controls display of dimension elbows.

Default and Available Settings

- **yes***—Dimensions display with elbows.
- **no**—Dimensions do not display with elbows.

default_font

Sets default text fonts as those fonts listed in the specified font index.

Default and Available Settings

- **font***—Takes the default font. You can specify a different font, if required.

Note: Do not include the `.ndx` extension. The font settings (`font` and `filled`) are located in the setup file.

default_pipe_bend_note

Controls display of pipe bend notes in drawings.

If set as text within quotation marks, uses that value when creating bend notes. Text may include parameters such as "&bend_name:att_pipe_bend" and "&bend_tol: att_pipe_bend".

If set as a directory path, references a previously created note saved as a file.

Default and Available Settings

- **no***—Pipe bend notes do not display in drawings.
- **yes**—Pipe bend notes display in drawings.

default_show_2d_section_xhatch

Controls the display of crosshatches for newly created 2D cross-sections for assemblies and parts.

Default and Available Settings

- **assembly_and_part***—Displays crosshatches for both assembly and part 2D cross-sections.
- **assembly_only**—Displays crosshatches only for assembly 2D cross-sections.
- **part_only**—Displays crosshatches only for part 2D cross-sections.
- **no**—Does not display crosshatches for assembly and part 2D cross-sections.

Note: This drawing setup file does not affect the display of existing 2D cross-sections.

default_show_3d_section_xhatch

Controls the display of crosshatches for 3D cross-sections.

Default and Available Settings

- **yes***—Crosshatches are displayed for selected 3D cross-sections.
- **no**—Crosshatches are not displayed for selected 3D cross-sections.

default_view_label_placement

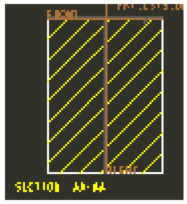
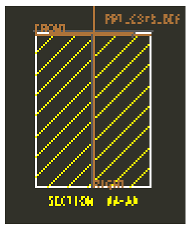
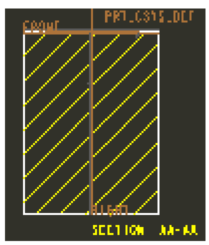
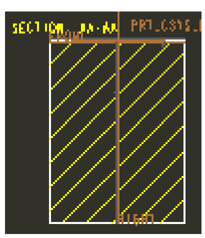
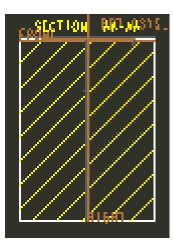
Enables you to specify the position and justification of the view labels. This drawing setup file option is applicable only for the following types of drawing views as they support automatic creation of view labels:

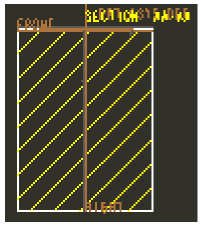
- Section
- Scaled
- Projection
- Detailed
- Copy and Align

This option is not applicable for the following types of drawing views as they do not support the automatic creation of view labels:

- General
- Auxiliary
- Revolved
- Partial
- Broken

Default and Available Settings

bottom_left*	The view label is placed at the bottom left of the view and the label text is right-aligned. 
bottom_center	The view label is placed at the bottom center of the view and the label text is center-aligned. 
bottom_right	The view label is placed at the bottom right of the view and the label text is left-aligned. 
top_left	The view label is placed at the top left of the view and the label text is right-aligned. 
top_center	The view label is placed at the top center of the view and the label text is center-aligned. 

top_right	The view label is placed at the top right of the view and the label text is left-aligned. 

Note: To display the view labels for projection views, set the `make_proj_view_notes` configuration option to `yes`. You can then use the `default_view_label_placement` drawing setup file option to specify the location of the labels for these views.

detail_circle_line_style

`solidfont`, `dotfont`, `ctrlfont`, `phantomfont`, `dashfont`, `ctrlfont_s_l`, `ctrlfont_l_l`, `ctrlfont_s_s`, `dashfont_s_s`, `phantomfont_s_s`, `cntrl_font_m_l`, `intmit_lww_hidden`, `pdfhidden_linestyle`

Sets the line font for circles indicating a detailed view in a drawing.

You can select any of the above line styles, or you can set any available system-defined or user-defined line font.

detail_circle_note_text

`DEFAULT`

Determines the text displayed in non-ASME-94 detail view reference notes.

Determines the default contents of the reference note for a detailed view. For example if the value is "See View" the note text reads "See View <viewname>"

detail_view_boundary_type

Determines default boundary type on the parent view of a detailed view.

Default and Available Settings

- **Circle***—Draws a circle in the parent view for the detailed view**.**
- **Ellipse**—Draws an ellipse in the parent view for the detailed view to closely match the spline, and prompts you to select an attachment point on the ellipse for the view note.
- **H/V Ellipse**—Draws an ellipse with a horizontal or vertical major axis and prompts you to select an attachment point on the ellipse for the view note.
- **Spline**—Displays the actual spline boundary on the parent view for the detailed view, and prompts you to select an attachment point on the spline for the view note.

- **ASME 94 Circ** —Displays an ASME standard compliant circle in the parent view as an arc with arrows and the detailed view name.

detail_view_circle

Sets display of a circle drawn about the portion of a model that is detailed by a detailed view.

Default and Available Settings

- **on***—Displays a circle about the detailed portion of a model.
- **off**—Does not display a circle about the detailed portion of a model.

detail_view_scale_factor

Determines default scale factor between a detail view and its parent view.

Default and Available Settings

- **2.000000***—Detail view scale is twice that of its parent view.

dim_dot_box_style

Controls the arrow style display of dots and boxes only for leaders of linear dimensions.

Default and Available Settings

- **default***—Uses the style established in the `draw_arrow_style` drawing option. BOM balloon arrows are controlled by the `def_bom_balloon_leader_sym` drawing setup file option, which defines the default arrow (attach point) style for BOM balloons in reports.
- **filled**—Fills dots and boxes for arrows of linear dimensions. Use "filled" to have new drawings appear with dots and boxes filled for dimension arrows.
- **hollow**—Dots and boxes for arrows of linear dimensions are not filled.

dim_fraction_format

Controls the display of fractional dimensions within drawings.

This drawing setup option supersedes the configuration file option of the same name (`dim_fraction_format`) unless the drawing setup option is set to default.

Default and Available Settings

- **default***—Displays fractional dimensions in drawings according to the setting of the configuration file option `dim_fraction_format`.

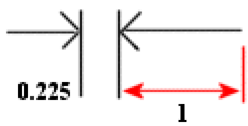
- **std**—Displays fractional dimensions in drawings in the standard Pro/ENGINEER format.
- **aisc—**Displays fractional dimensions in drawings in the AISC format. The AISC setting also displays architectural units according to AISC format for feet-inches dimensions.

Note: When you retrieve drawings created prior to Pro/ENGINEER Release 2000*i*, the `dim_fraction_format` and `use_major_units` drawing setup options combine to control the display of dimensions.

dim_leader_length

0.500000

Sets the length of the dimension leader lines when the leader arrows are outside the witness lines.



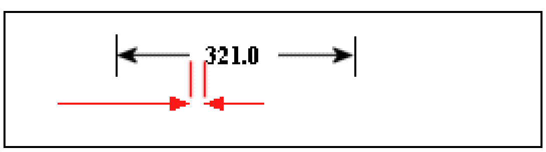
1. The length of the dimension leader line of the leader arrow that is outside the witness lines. The length is equal to the value specified by the `dim_leader length` drawing setup file option.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `dim_leader_length` drawing setup file option.

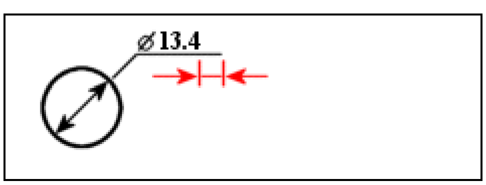
dim_text_gap

0.500000

Controls the distance between dimension text and dimension leader line and represents the ratio between gap size and text height.



For diameter dimensions, if `text_orientation` is set to `parallel_diam_horiz`, the `dim_text_gap` drawing setup option controls the extension of an elbow line beyond the dimension text.



Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `dim_text_gap` drawing setup file option.

dim_tol_lead_trail_zeros

Controls the appearance of leading and trailing zeroes of the dimension tolerance values that appear in any tolerance mode other than the limits and nominal format. This does not affect the appearance of angular dimension tolerance values.

Default and Available Settings

- **same_as_lead_trail_zeros***—The appearance of leading and trailing zeroes of the dimension tolerance values is same as that of dimension values set by the drawing setup file option `lead_trail_zeros`.
- **by_model_units**—The leading and trailing zeroes of the dimension tolerance values appear according to the units of the model. If the model has English units, the leading zero does not appear and trailing zeroes are added according to the number of decimal places you have set for the dimension value. If the model has Metric units, the leading zero appears and trailing zeroes do not appear.
- **trail_only**—Regardless of the units of the model, trailing zeroes appear, but the leading zero does not appear.
- **lead_only**—Regardless of the units of the model, only the leading zero appears, while the trailing zeroes do not appear.
- **both**—Regardless of the units of the model, both the leading zero and trailing zeroes appear.

dim_trail_zero_max_places

same_as_dim

Sets the maximum number of decimal places for dimensions and dimension tolerances when trailing zeros are used to attain the maximum number of decimal places defined by the `default_dec_places` configuration option. Specify a value that is less than the specified number of decimal places for dimensions to control the number of trailing zeros.

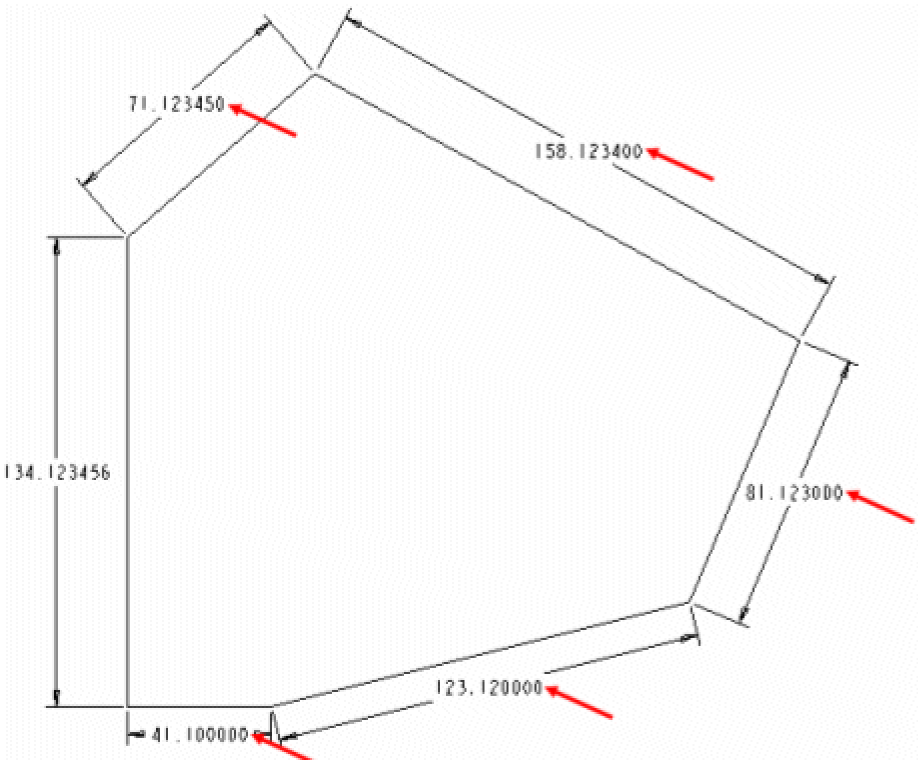
Note:

- This drawing setup file option does not affect decimal places if the value specified for the `lead_trail_zeros` drawing setup file option prevents trailing zeros from being displayed.
- This drawing setup file option affects the display of all dimensions in a drawing. You cannot individually control the number of decimal places for dimensions with trailing zeros.
- This drawing setup file option does not affect angular dimensions displayed in the degree, minute, and second format.

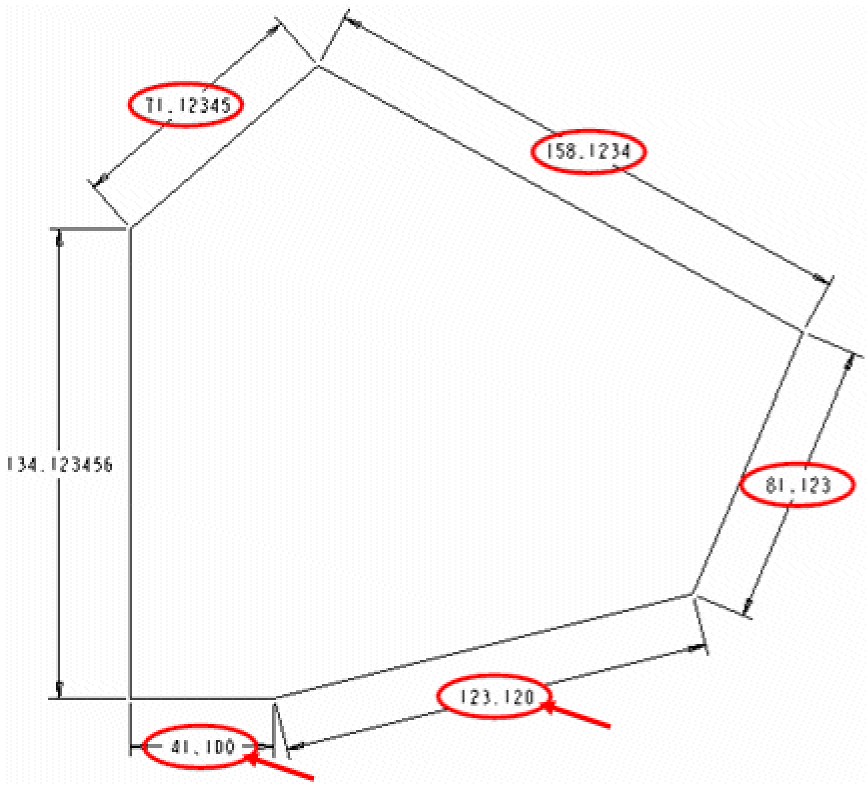
Consider the following example:

1. Specify the value `2` for the `default_dec_places` configuration option.
2. Specify the value `same_as_dim` for `dim_trail_zero_max_places`.

3. Specify the value `both` for `lead_trail_zeros`.
4. Click **Format** > **Decimal Places** and type the value `6`. Trailing zeros are added for dimensions whose values do not extend upto 6 decimal places. Such dimensions are identified by an arrow as shown in the following figure:



In this example, if you change the value of `dim_trail_zero_max_places` to `3`, trailing zeros are added for all dimensions with values less than 3 decimal places. Such dimensions are identified by an arrow as shown in the following figure:



draft_scale

1.000000

Controls the scale of draft dimensions relative to the actual length of draft entity in drawing.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `draft_scale` drawing setup file option.

draw_ang_unit_trail_zeros

Controls display of angular dimensions.

Default and Available Settings

- **yes***—Removes trailing zeros (in adherence to ANSI standards) when showing angular dimensions in degrees/minutes/seconds format.
- **no**—Does not display trailing zeros in angular dimensions or tolerances.

draw_ang_units

Sets display of angular dimensions in a drawing.

Default and Available Settings

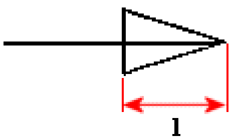
- **ang_deg***—Creates decimal degrees

- **ang_min**—Creates degrees and decimal minutes
- **ang_sec**—Creates degrees, minutes, and decimal seconds

draw_arrow_length

0.187500

Sets the length of leader line arrowheads.



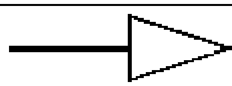
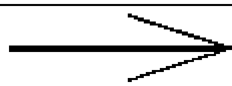
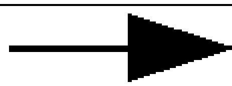
1. The length of the leader line arrowhead as specified by the `draw_arrow_length` drawing setup file option

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `draw_arrow_length` drawing setup file option.

draw_arrow_style

Controls arrow style for all detail items involving arrows, including leaders of dimensions, notes, 3-D notes, geometric tolerances, symbols, and balloons.

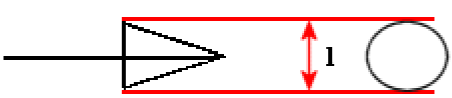
Default and Available Settings

closed*—	
opened—	
filled—	

draw_arrow_width

0.062500

Sets the width of the leader line arrowheads.



1. The width of the leader line arrowhead as specified by the `draw_arrow_width` drawing setup file option

Note:

- The `drawing_units` drawing setup file option determines the unit of measurement for the `draw_arrow_width` drawing setup file option.
- The value specified by the `draw_arrow_width` drawing setup file option determines the values of the `draw_attach_sym_height`, `draw_attach_sym_width` and `draw_dot_diameter` drawing setup file options.

draw_attach_sym_height

Sets height of leader line slashes, integral signs, and boxes.

Default and Available Settings

- **default***—Uses value set for `draw_arrow_width`
- **value**—Type a value for the height of leader line slashes, integral signs, and boxes.

Note: The measurement is controlled by the `drawing_units` setup file option.

draw_attach_sym_width

Sets width of leader line slashes, integral signs, and boxes.

Default and Available Settings

- **default***—Uses value set for `draw_arrow_width`
- **value**—Type a value for the width of leader line slashes, integral signs, and boxes.

Note: The measurement is controlled by the `drawing_units` setup file option.

draw_cosms_in_area_xsec

Controls display of cosmetic sketches and datum curve features that lie in the cutting plane in planar area cross-sectional views.

Default and Available Settings

- **no***—Does not show all cosmetic sketches and datum curve features that lie in the cutting plane.
- **yes**—Shows all cosmetic sketches and datum curve features that lie in the cutting plane.

draw_dot_diameter

Sets diameter of leader line dots.

Default and Available Settings

- **default***—Uses the value set in the `draw_arrow_width` drawing setup file option.
- **value**—Type a value for the diameter of leader line dots.

Note: The measurement is controlled by the `drawing_units` setup file option.

draw_layer_overrides_model

Directs drawing layer display setting to determine the setting of drawing model layers with the same name.

Default and Available Settings

- **no***—Ignores nondrawing layers when the display status of layers is set in the drawing model.
- **yes**—Implicitly includes drawing model layers in drawing layers with the same name for purposes of setting the display.

drawing_text_height

Sets default text height for all text within the drawing.

Default and Available Settings

- **0.156250***—The default value is 0.156250. You can specify a value for the height of all text within drawings.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `drawing_text_height` drawing setup file option.

drawing_units

`inch`, `foot`, `mm`, `cm`, `m`

Specifies the unit for all drawing parameters.

dual_digits_diff

`-1`

Controls the number of digits to the right of the decimal of the secondary dimension as compared to the primary dimension.

For example, if you specify `-1` as the value for secondary dimensions when primary units are in inches and secondary units are in millimeters, the result is as follows:

`10.235 [259.96].`

dual_dimension_brackets

Controls display of brackets with dimension units.

Default and Available Settings

- **yes***—Displays dimension units that occur second in brackets;
- **no**—Does not display brackets.

Note: This drawing setup file option only works when you also set the `dual_dimensioning` drawing setup option.

dual_dimensioning

Controls the formatting of dimension display; whether or not dual dimensions are used and also how they display.

Default and Available Settings

- **no***—Displays a single value for dimensions.
- **primary [secondary]**—Displays dimensions with primary units (established by the model) and secondary units.
- **secondary**—Only displays the secondary dimensions of the drawing, as if they were primary.

Note: The following drawing setup file options can be used in conjunction with this option to modify the display of dual dimensions:

- `dual_secondary_units` specifies the secondary set of units used by the drawing.
- `dual_digits_diff` specifies the number of decimal places that secondary units contain compared with the primary units used.
- `decimal_marker` specifies the character to use to mark the decimal point in secondary dimensions.
- `dual_dimension_brackets` specifies whether one of the units of dimensions appears within brackets.

dual_metric_dim_show_fractions

Determines whether the metric portion of dual dimension will display fractions when the primary/model units are fractions.

Defaults and Available Settings

- **no***—Dual dimensions do not display as fractions when the primary/model units are fractions..
- **yes**—Dual dimensions display as fractions when the primary/model units are fractions.

dual_secondary_units

mm, inch, foot, cm, m

Specifies the unit for the display of secondary dimensions in dual dimension schemes.

gtol_datum_placement_default

Determines whether the set datum is attached above or below the geometric tolerance control frame.

Defaults and Available Settings

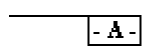
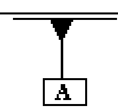
- **on_bottom***—Places set datum on the bottom of the control frame.
- **on_top**—Places set datum on the top of the control frame.

Note: To be able to attach set datum to geometric tolerances; the gtol_datums drawing setup file option should be set to either std_iso or std_iso_jis or std_jis. The default setting is std_ansi.

gtol_datums

Sets the drafting standard for displaying reference datums within drawings. This affects the display of axes, datum planes and reference part datums.

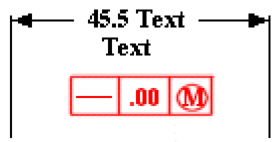
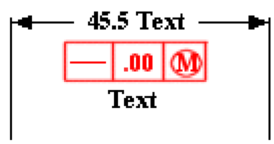
Default and Available Settings

std_ansi*	
std_ansi_mm	
std_iso	
std_jis	
std_din	
std_iso_jis	
std_ansi_dashed	
std_asme	

gtol_dim_placement

Determines the location of a feature control frame when a geometric tolerance is attached to a dimension symbol that contains additional text.

Default and Available Settings

on_bottom*	Places the geometric tolerance at the bottom of the dimension symbol, beneath any additional lines of text. 
under_value	Places the geometric tolerance immediately below the dimension value and before any additional lines of text. 

gtol_lead_trail_zeroes

Controls the display of leading and trailing zeroes in gtols.

Default and Available Settings

The display for the `same_as_lead_trail_zeros`, `by_model_units`, `trail_only(english)`, `lead_only(metric)`, and `both` settings is valid when the `dual_dimensioning` drawing setup file option is set to `no`.

- **same_as_lead_trail_zeros***—The display of a single, leading zero and trailing zeros in gtol tolerance values is determined by the current value of the `lead_trail_zeros` drawing setup file option.
- **by_model_units**—The display of the leading zero and trailing zeros is determined by the model units.
- **trail_only(english)**—The leading zero is not displayed and trailing zeros are displayed as specified while inserting the gtol.
- **lead_only(metric)**—A single, leading zero is displayed and trailing zeros are not displayed.
- **both**—A single, leading zero is displayed and trailing zeros are displayed as specified while inserting the gtol.

  Note:

  - When `by_model_units` is the value for the `gtol_lead_trail_zeros` drawing setup file option and `English` is the unit, the leading zero is not displayed and trailing zeros are displayed as specified while inserting the gtol.
  - When `by_model_units` is the value for the `gtol_lead_trail_zeros` drawing setup file option and `Metric` is the unit, a single, leading zero is displayed and trailing zeros are not displayed.

- **by_model_units [by_secondary_units]**—Primary display of the gtols values is based upon model units and secondary display is based upon the secondary units.
- **by_model_units [trail_only(english)]**—Primary display of the gtol values is based upon model units. Secondary units do not display a leading zero and trailing zeros are displayed as specified while inserting the gtol.
- **by_model_units [lead_only(metric)]**—Primary display of gtol values is based upon model units. Secondary units display a single, leading zero and trailing zeros are not displayed
- **by_model_units [both]**—Primary display of gtol values is based upon model units. Secondary units display a single, leading zero and trailing zeros are displayed as specified while inserting the gtol.
- **trail_only [lead_only]**—Primary units do not display a leading zero and trailing zeros are displayed as entered. Secondary units display a single, leading zero and trailing zeros are not displayed.
- **trail_only(english) [trail_only(english)]**—Both primary units and secondary units do not display a leading zero. Both display trailing zeros as specified while inserting the gtol.
- **trail_only(english) [both]**—Primary units do not display a leading zero and trailing zeros are displayed as entered. Secondary units display a single, leading zero and trailing zeros are displayed as specified while inserting the gtol.
- **lead_only(metric) [trail_only(english)]**—Primary units display a single, leading zero and trailing zeros are not displayed. Secondary units do not display a leading zero and trailing zeros are displayed as specified while inserting the gtol.
- **lead_only(metric) [lead_only(metric)]**—Both primary and secondary units display a single, leading zero and trailing zeros are not displayed.
- **lead_only(metric) [both]**—Primary units display a single, leading zero and trailing zeros are not displayed. Secondary units display a single, leading zero and trailing zeros are displayed as specified while inserting the gtol.
- **both [both]**—Both primary and secondary unit display a single, leading zero and trailing zeros are displayed as specified while inserting the gtol.
- **both [trail_only(english)]**—Primary units display a single, leading zero and trailing zeros are displayed as entered by you. Secondary units do not display a leading zero and trailing zeros are displayed as specified while inserting the gtol.
- **both [lead_only(metric)]**—Primary units display a single, leading zero and trailing zeros are displayed as entered by you. Secondary units display a single, leading zero and trailing zeros are not displayed.

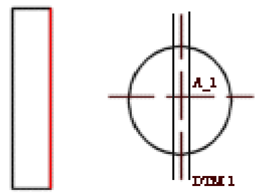
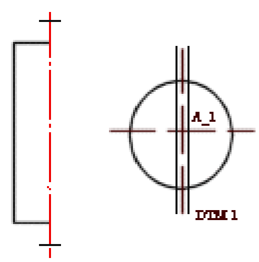
  Note: The display for the above settings is valid when the `dual_dimensioning` drawing setup file option is set to `primary[secondary]` or `secondary[primary]`.

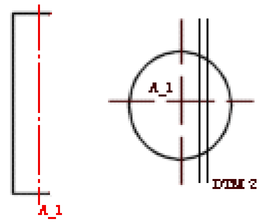
Note: You can specify secondary units using the `dual_secondary_units` drawing setup file option.

half_view_line

Formats the line that designates a half view.

Default and Available Settings

solid*	Draws solid lines where material is present. 
symmetry	Draws a centerline extending beyond the part and acting as a break line. 
symmetry_iso	Displays a half view symmetry line according to ISO standard 128:1982 5.5— half view symmetry lines are displayed in yellow with thin linestyle, and the "hash" marks at the ends of the symmetry line are also yellow with thin linestyle.
symmetry_asme	Displays a half view symmetry line according to ASME standard ASME Y14.2M-1992—half view symmetry lines are displayed in yellow with thin linestyle, and the "hash" marks are displayed in white with thick linestyle.
none	Draws the object a small distance past the symmetry line.

	 **Note**: When set to none, it is good practice to cut the view on a datum plane that is offset past the actual center line so the datum axis can be shown and the location of the part centerline will be clear. The cut will always be made on the datum plane selected.

harn_tang_line_display

Controls the display of all the internal segment portions of cables when you display thick cables.

Default and Available Settings

- **no***—Does not display all the internal segments for thick cables.
- **yes**—Displays all the internal segments for thick cables.
- **default**—Reverts to the setting established with the `harn_tang_line_display` configuration file option.

Note: Drawing setup file options that have configuration options of the same name only override the configuration setting within drawings. The configuration setting remains the same within the model.

hidden_tangent_edges

Controls the display of hidden tangent edges in drawing views.

Default and Available Settings

- **default***—Reverts to the environment display setting for tangent edges.
- **dimmed**—Plots hidden tangent edges in a view using Pen 7. Lines appear dashed in same color as dimmed visible tangent edges. However, you must select **Hidden Line** or **No Hidden Line** from the **Display Style** list in the Pro/ENGINEER Environment dialog box.
- **erased**—Removes all hidden tangent edges automatically from screen and plot.

hlr_for_datum_curves

Specify whether datum curves should be included for hidden line calculations.

Defaults and Available Settings

- **yes***—Datum curves are included when calculating the display of hidden lines. Blanking and unblanking datum curves on a drawing will cause hidden lines in views to be recalculated.
- **no**—Datum curves are not included when calculating the display of hidden lines. Blanking and unblanking datum curves on a drawing will not cause hidden lines in views to be recalculated. When using this option, datum curves will be displayed in wireframe.

Note: Hidden lines are initially calculated for all views when this configuration option is changed. This ensures that the display of the drawing remains stable when saved and retrieved. Subsequent blanking and unblanking of layers containing only datum curves will not force hidden lines to be recalculated.

hlr_for_pipe_solid_cl

Controls the display of pipe centerlines. This drawing setup option only operates on pipes created in Pro/PIPING, not on pipe features in a part.

Default and Available Settings

- **no***—Hidden line removal does not affect pipe centerlines.
- **yes**—Hidden line removal affects pipe centerlines.

hlr_for_threads

Controls the display of threads in a drawing depending on whether it complies with the ISO, ANSI, or JIS standard set by the `thread_standard` drawing setup option.

Default and Available Settings

- **no***—Displays thread edges as surfaces as they would be in Part mode.
- **yes**—Thread edges meet ANSI, ISO, or JIS standard for Hidden Line display.

Note: If you change the setting for the `hlr_for_threads` drawing setup file option, then the drawing reflects the changes only after you update the drawing by clicking **View** > **Update** > **All Sheets**.

ignore_model_layer_status

Controls whether the model layer status is considered in drawings.

Default and Available Settings

- **yes***—Ignores changes to all layer status in the models of the drawing made in another mode.
- **no**—Considers model layer status within drawings. You cannot use the layer status changes stored with an applied combination state when creating drawing

views, as all drawing views within a drawing can have only one layer status. You cannot apply each layer status of each combination state to each drawing view.

iso_ordinate_delta

Improves display of offset between an ISO-ordinate dimension line and witness line, referred to as the witness line delta.

Default and Available Settings

- **no***—Does not display offset exactly in accordance with the specified value (it is "off" by about 2 millimeters).
- **yes**—Displays offset correctly, according to value specified for the drawing setup file option `witness_line_delta`.

lead_trail_zeros

Controls the display of leading and trailing zeros in dimensions.

Default and Available Settings

- **std_default***—Displays only trailing zeros. No leading zeros are displayed.
- **std_metric**—Displays only leading zeros. Trailing zeros are not displayed.
- **std_english**—Displays only trailing zeros. Leading zeros are not displayed.
- **both**—Both leading and trailing zeros are displayed.

Note:

- If the `lead_trail_zeros_scope` drawing setup option is set to `all`, the `lead_trail_zeros` drawing setup file option controls the display of leading and trailing zeros in dimensions, hole parameters within hole notes, and all floating point parameters on a drawing, including parametric notes, view scale notes, tables, symbols, and cosmetic thread notes.
- In case of dual dimensioning, the `lead_trail_zeros` drawing setup file option controls the use of leading and trailing zeros in both `std_english` and `std_metric` standards independently.
  - If the units in the `dual_dimensioning` drawing setup file option are `primary[secondary]`, `std_english[std_metric]` shows the primary units values with trailing zeros, while the secondary units show values with leading zeros.
  - If the units in the `dual_dimensioning` drawing setup file option are `secondary[primary]`, `std_english [std_metric]` secondary units show values with trailing zeros, while the primary units show values with leading zeros.
- Hole parameters within the hole notes appear with 3 decimals, regardless of the value set for the `default_dec_places` configuration option. If you want to

change the number of decimal places for the hole parameter, type `[.n]` after the parameter in the **Text** tab of the **Note Properties** dialog box. Here, `n` is the number of decimal places. You can edit the hole parameter values only through the **Note Properties** dialog box that opens when you select the hole note and click **Edit** > **Value**. Alternatively, to edit hole parameter values through the **Note Properties** dialog box, select the hole note, right-click, and click **Edit Value** on the shortcut menu.

lead_trail_zeros_scope

Controls whether only dimensions are affected by the setting of the drawing setup option `lead_trail_zeros`.

Default and Available Settings

- **dims***—The drawing setup option `lead_trail_zeros` controls only dimensions.
- **all—**The drawing setup option `lead_trail_zeros` controls dimensions, hole parameters within hole notes and also all floating point parameters on a drawing, including parametric notes, view scale notes, tables, symbols, and cosmetic thread notes.

leader_elbow_length

0.250000

Determines the length of the leader elbow for model and draft datums.

Specify a value to this option to:

- Set the default elbow length for new set datums
- Reset the elbow length for the existing set datums if you have not explicitly modified the elbow length of the existing set datums.

Note:

- The elbow length of a set datum is not controlled by `leader_elbow_length` drawing setup file option, once you have explicitly modified the elbow length of a set datum.
- The `drawing_units` drawing setup file option determines the unit of measurement for the `leader_elbow_length` drawing setup file option.

leader_extension_font

Sets the font for leader extension lines for notes, surface finish symbols, and symbols. Leader extension lines are created when you drag a draft entity off its attachment.

Note: The option does not apply to curved leader extensions.

Default and Available Settings

- **SOLIDFONT***
- **DOTFONT**
- **CTRLFONT**
- **PHANTOMFONT**
- **DASHFONT**
- **CTRLFONT_S_L**
- **CTRLFONT_L_L**
- **CTRLFONT_S_S**
- **DASHFONT_S_S**
- **PHANTOMFONT_S_S**
- **CTRLFONT_MID_L**
- **INTMIT_LWW_HIDDEN**
- **PDFHIDDEN_LINESTYLE**

line_style_length

Sets the font length for two-dimension sketched entities

You must add this option to the drawing setup file whenever you want to modify the length. You must also set the drawing setup file option `axis_interior_clipping` to `no`.

The length measurement is controlled by the `drawing_units` setup file option.

Default and Available Settings

- **font_name default***—Type the font name and then a desired value for the font length in system units. The "default" setting indicates default length values.
- **font_name value**—Type the font name and then a desired value for the font length in system units.

Note: After you add `line_style_length` to the drawing setup file, you cannot delete it by deleting the row from the file or by retrieving a different `DTL` file into the drawing. You must change the value of this option to default to eliminate the option from the drawing setup file. Use the following format:

```
line_style_length font_name value/default
```

where `font_name` is the name of the font that you want to modify, `value` is the desired value for the font length in system units, and `default` tells the system to use the default length value.

line_style_standard

Controls text color in drawings.

Unless you set this option to `std_ansi`, all drawing text displays in blue, and the boundaries of detailed views display in yellow.

Default and Available Settings

- **std_ansi***—Applies American National Standards Institute (ANSI)
- **std_iso**—Applies International Organization for Standardization (ISO)
- **std_jis**—Applies Japanese Industrial Standard (JIS)
- **std_din**—Applies Deutsches Institut für Normung (DIN) (German Institute for Standardization)

location_radius

Modifies radius of nodes indicating location, improving their visibility, particularly when printing drawings.

Default and Available Settings

- **default (2.)***—Sets radius as 2 drawing units.
- **0.0**—Displays location nodes, but does not print them. There is no maximum value for this setting. You can type a value for the radius of nodes indicating location.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `location_radius` drawing setup file option.

max_balloon_radius

Sets the maximum allowable balloon radius.

Default and Available Settings

- **0.000000***—Balloon radius depends only on text size. You can type a value for the maximum allowable balloon radius.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `max_balloon_radius` drawing setup file option.

mesh_surface_lines

Controls display of blue surface mesh lines.

Default and Available Settings

- **on***—Blue surface mesh lines display.

- **off**—Blue surface mesh lines do not display.

min_balloon_radius

Sets minimum allowable balloon radius.

Default and Available Settings

- **0.000000***—Balloon radius depends only on text size. You can type a value for the minimum allowable balloon radius.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `min_balloon_radius` drawing setup file option.

min_dist_between_bom_balloons

Controls the default minimum distance between BOM balloons.

Default and Available Settings

- **0.800000***

model_digits_in_region

Controls display of number of digits in two-dimensional repeat regions.

Default and Available Settings

- **yes***—Two-dimensional repeat regions reflect the number of digits of part and assembly model dimensions.
- **no**—Two-dimensional repeat region digits are independent of the part and assembly model dimension settings.

model_display_for_new_views

Specifies the display style for drawing views.

Default and Available Settings

- **default***—Reverts to the environment setting. The environment is determined by the setting in the **Display Style** box in the **Environment** dialog box that opens when you click **Tools** > **Environment**.

  Note: This is the default model display only for views imported from Pro/ENGINEER Wildfire 2.0 and earlier. You can set this configuration option to the default setting or to the other settings supported for the current release. As a result, the drawings become valid for the current release.

- **follow_environment***—Always follow the current setting in the environment at the time of view creation.

Note: This is the default setting for drawing views created in Pro/ENGINEER Wildfire 3.0.

- **wireframe**—Displays the view in wireframe.
- **hidden_line**—Displays the view with hidden lines.
- **no_hidden**—Displays the view without showing hidden lines.
- **shading**—Displays the view in shading, taking the colors from the model.
- **save_environment**—Saves the current environment setting for newly created views within the drawing. If the environment setting changes, the view is not updated.

Note: If you set the `enable_shaded_view_in_drawing` configuration option to `no` and the `model_display_for_new_views` drawing setup file option to `shading`, then the actual display of the drawing view is in wireframe.

model_grid_balloon_display

Determines whether a circle will be drawn around the model grid text.

Defaults and Available Settings

- **yes***—Draws a circle around the model grid text.
- **no**—Does not draw a circle around the model grid text.

model_grid_balloon_size

0.200000

Specifies the default radius of balloons shown with the model grid in a drawing.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `model_grid_balloon_size` drawing setup file option.

model_grid_neg_prefix

Controls prefix of negative values shown in model grid balloons.

Default and Available Settings

- **- ***

model_grid_num_dig_display

Controls number of digits displayed in grid coordinates that appear in grid balloons.

Default and Available Settings

- **0***—The system default (0) to display coordinates as integers. You can type a value for the number of digits displayed in grid coordinates.

model_grid_offset

Controls offset of new model grid balloons from the drawing view.

Default and Available Settings

- **default***—Offsets new model grid balloons from the drawing view by twice the current model grid spacing. You can type a value for the offset of model grid balloons from the drawing.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `model_grid_offset` drawing setup file option.

model_grid_text_orientation

Determines whether the model grid text orientation will be parallel to the grid lines or always horizontal.

Defaults and Available Settings

- **horizontal***—Always displays the model grid text horizontal to the grid lines.
- **parallel**—Displays the model grid text parallel to the grid lines.

model_grid_text_position

Determines whether the model grid text will appear above, below, or centered about the grid line.

Defaults and Available Settings

- **centered***—Centers the model grid text about the grid line.
- **above—**Places the model grid text above the grid line.
- **below—**Places the model grid text below the grid line.

Note: If you set the `model_grid_text_orientation` drawing setup file option to horizontal, `model_grid_text_position` is ignored.

new_iso_set_datums

Controls display of set datums according to ISO standards.

Default and Available Settings

- **yes***—Displays set draft datums according to ISO standards.
- **no**—Does not display set draft datums according to ISO standards.

node_radius

Controls display of nodes in symbols.

Default and Available Settings

- **default***—Specifies the default radius of the nodes. You can type a value for the radius of nodes within symbols. If the setting is so small that the nodes do not appear, the system uses the default setting. There is no maximum value for this setting.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `node_radius` drawing setup file option.

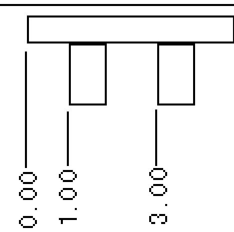
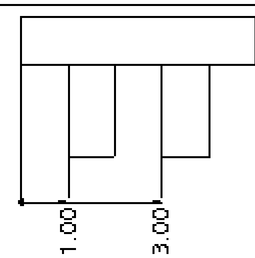
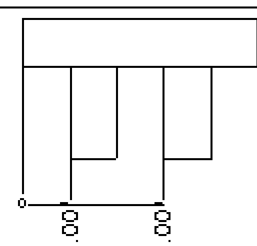
ord_dim_standard

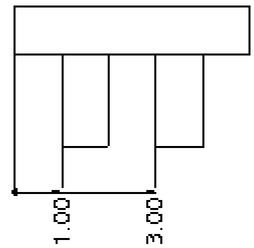
Controls the display standard for ordinate dimensions in your drawing.

The `ord_dim_standard` option works in conjunction with the following drawing setup file options:

- `draw_arrow_style`—Sets the arrow style to closed, open, or filled.
- `draw_dot_diameter`—Sets the diameter for the leader line dots (the circle on the baseline).

Default and Available Settings

`std_ansi*`	Places related ordinate dimensions without a connecting line.	
`std_iso`	Displays ordinate dimensions in the International Organization for Standardization (ISO) format.	
`std_jis`	Places related ordinate dimensions along the connecting line perpendicular to the baseline and starts with an open circle. Each segment of the connecting line ends with an arrow.	

std_din	Displays ordinate dimensions in the Deutsches Institut für Normung (DIN) (German Institute for Standardization) format.	
same_as_3d	Displays ordinate dimensions of a 3D model in the same style as they appear in the drawing.	

Note:

- When `ord_dim_standard` is set to `std_jis`, the end circle will always be an open circle with a fixed size regardless of the values of `draw_dot_diameter` and `draw_arrow_style`.
- When witness lines are interconnected, moving any of the related dimensions moves all the dimensions.

orddim_text_orientation

Controls ordinate dimension text orientation.

Default and Available Settings

- **parallel***—Displays dimension text parallel to the leader lines.
- **horizontal**—Displays it horizontally, parallel to the bottom of the drawing sheet.

parallel_dim_placement

Determines whether dimension value displays above or below leader line when you set the "text_orientation" option to "parallel."

Default and Available Settings

- **above***—Displays dimensions value above the leader line.
- **below**—Displays dimensions value below the leader line.

Note: The `parallel_dim_placement` drawing setup option does not apply to dual dimensions.

pipe_insulation_solid_xsec

Determines whether the insulation in the pipe cross section is displayed as a solid (filled) region.

Default and Available Settings

- **no***—Pipe insulation cross sections are not displayed as solid regions.
- **yes**—Pipe insulation cross sections are displayed as solid regions.

Note: This option does not apply to offset cross sections.

pipe_pt_line_style

Controls the line style of a theoretical bend intersection point that appears within the shape defined by the pipe_pt_shape drawing setup option in a piping drawing.

Default and Available Settings

- **default***—The bend intersection point appears with a default line style.
- **phantom**—The bend intersection point appears with a phantom line style.
- **solid**—The bend intersection point appears with a solid line.

pipe_pt_shape

Controls shape of theoretical bend intersection points in a piping drawing.

Default and Available Settings

- **cross***
- **dot**
- **circle**
- **triangle**
- **square**

pipe_pt_size

Controls size of theoretical bend intersection points in a piping drawing.

Default and Available Settings

- **default***—Specifies the default size of theoretical bend intersection points. You can type a value for the size of theoretical bend intersection points in piping drawings.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `pipe_pt_size` drawing setup file option.

pos_loc_format

Controls the appearance of `&pos_loc` text both in notes and report tables. The character pairs %% are used in the following ways:

- `%s` indicates single sheet
- `%x` and `%y` indicate the horizontal and vertical positions
- `%r` indicates the end of a repeatable substring

Default and Available settings

Type the desired character pairs to determine the appearance of `&pos_loc` text, such as `%s%x%y, %r`

projection_type

Determines method for creating projection views.

Default and Available Settings

- **third_angle***
- **first_angle**

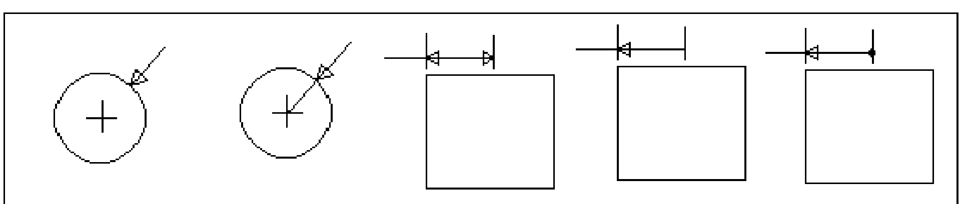
radial_dimension_display

Allows display of radial dimensions in ASME, ISO or JIS standard formats, except when the `text_orientation` drawing setup option is set to `horizontal`, which forces the display of dimensions to be in the ASME format.

Note: Use this drawing setup option with the `text_orientation` drawing setup option for setting the display of radial dimensions.

Default and Available Settings

- **std_asme**—Displays radial dimensions in the American Society of Mechanical Engineers (ASME) format
- **std_iso**—Displays radial dimensions in the International Organization for Standardization (ISO) format
- **std_jis**—Displays radial dimensions in the Japanese Industrial Standard (JIS) format

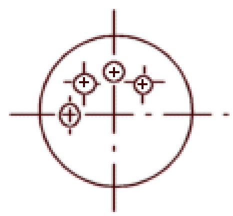
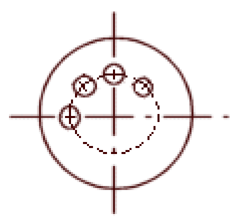


std_asme	✔		✔		
std_iso		✔		✔	
std_jis		✔			✔

radial_pattern_axis_circle

Sets display mode for axes of rotation that are perpendicular to the screen in radial pattern features.

Default and Available Settings

no*	Displays the individual axis lines. 
yes	A circular shared axis appears, and axis lines pass through the center of a rotational pattern. 

ref_des_display

Controls display of reference designators in a drawing of a cabling assembly.

Default and Available Settings

- **no***—Does not display reference designators in drawings.
- **yes**—Displays reference designators in drawings.
- **default**—Automatically selects the **Reference Designators** environment setting.

reference_bom_balloon_text

Controls reference balloons text identifier for BOM balloons.

Default and Available Settings

- **"DEFAULT"***—Text string REF appears next to the balloon for simple balloons. For quantity balloons, the string REF appears instead of the quantity value.

remove_cosms_from_xsecs

Controls display of datum curves, threads, cosmetic feature entities, and cosmetic cross-hatching in a full cross-sectional view.

Default and Available Settings

- **total***—Removes features located entirely in front of the cutting plane from the cross-sectional view. Displays these features in full only if they intersect the cutting plane. This setting is applicable only for full cross-sectional views and does not apply to view that has combination of cross section types.
- **all**—Removes datums and cosmetics from all types of cross-sectional views.
- **none**—Displays all datum quilts and cosmetic features.

set_datum_leader_length

Determines the default length of the leader for a draft set datum and model set datum.

Default and Available Settings

`0.375000`

Note:

- The `drawing_units` drawing setup file option determines the unit of measurement for the `set_datum_leader_length` drawing setup file option.
- For drawings created in releases prior to Pro/ENGINEER Wildfire 4.0, the default length of the leader for draft set datum and model set datum is 1.5 times the value of the `leader_elbow_length` drawing setup file option.

set_datum_triangle_display

Controls the display of the set datum triangle as an empty triangle or a filled triangle, in both draft and model set datum labels. Changing the value of the configuration option results in the change of display of all set datum triangles in the current drawing. You cannot change the display of individual datum triangles.

Default and Available Settings

- **filled***—The set datum triangle is filled.
- **open**—The set datum triangle is not filled.

Note: This drawing setup file option does not affect the three-dimensional model.

show_cbl_term_in_region

Allows use of the report symbols `&asm.mbr.name` and `&asm.mbr.type` to show terminators in Pro/REPORT tables for cable assemblies having connectors with terminator parameters.

Default and Available Settings

- **no***—For existing drawings, the default value is `no`.
- **yes**—Shows terminators. You must set the attribute **Cable Info** for the repeat region. When creating new drawings, the default value is `yes`.

show_dim_sign_in_tables

Controls the display of negative values in a drawing table for drawing items with negative values in a family table.

Default and Available Settings

- **yes***—Displays negative values in a drawing table.
- **no**—Does not display negative values in a drawing table.

Note: `yes` is the default setting only for newly created drawings. For existing drawings, the default setting is `no`. Set the value of `show_dim_sign_in_tables` to `yes` to display negative signs for existing drawings.

show_pipe_theor_cl_pts

Controls display of centerlines and theoretical intersection points in piping drawings.

Default and Available Settings

- **bend_cl***—Shows centerlines with bends only.
- **theor_cl**—Shows only centerlines with theoretical bend intersection points.
- **both**—Shows both bends and theoretical intersection points.

show_quilts_in_total_xsecs

Determines if surface geometry is included in a drawing cross section and whether the surface will be cut by the cross section cutting plane.

Surface geometry (surfaces and surface quilts) will only be cut by the cross-section cutting plane in a drawing view when the cross-section is created in part or assembly mode. If the cross-section is created as a Model (default), the surface geometry will not be cut by the cutting plane.

Default and Available Settings

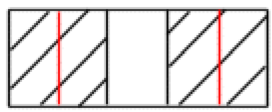
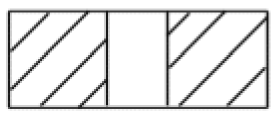
- **no***—Exclude surface geometry.

- **yes**—Include surface geometry.

show_total_unfold_seam

Controls display of seams (the edges of the cutting plane) in total unfolded cross-sectional views.

Default and Available Settings

yes*	Displays the seams (the edges of the cutting plane). 
no	Blanks the seams (the edges of the cutting plane). 

shrinkage_value_display

Displays dimension shrinkage in percentages or as final values.

Default and Available Settings

- **percent_shrink***
- **final_value**

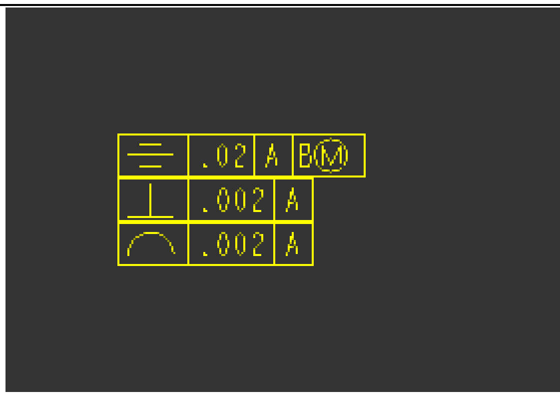
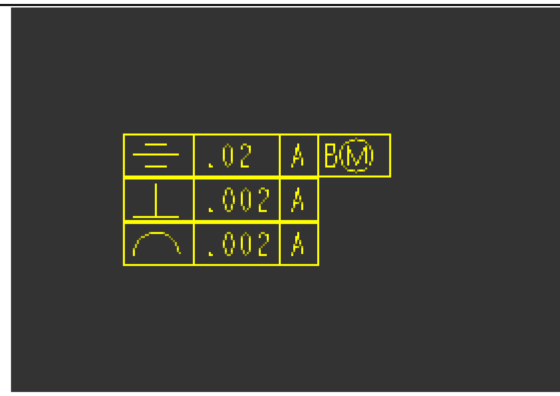
stacked_gtol_align

Controls the alignment of the control frame for stacked geometric tolerances.

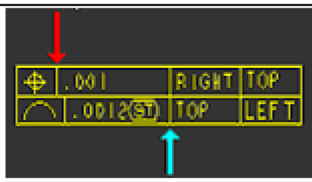
The geometric tolerance stack control frame is always centered on the attached leader line.

Defaults and Available Settings

- **yes***—Geometric tolerances comply with the JIS standard; alignment on both ends of the control frame.
- **no**—The controls frames do not align.

<table>
<tr><th>Unaligned Control Frames</th><th>Aligned Control Frames</th></tr>
<tr><td>
</td><td>
</td></tr>
</table>

Note: If the geometric symbols are set to varying sizes, alignment of the remaining control frame compartments starts at the right margin of the tolerance value compartment.



sym_flip_rotated_text

Flips any text in a **Rotate Text** symbol that is upside down, making it right-side up.

Default and Available Settings

- **no***—Does not flip the text.
- **yes**—If the symbol orientation is +/- 90 degrees, flips the text, rotating it along with the symbol.

tan_edge_display_for_new_views

Specifies the tangent edge display.

Default and Available Settings

- **default***—Displays tangent edges according to the setting in the Environment dialog box (**Tools** > **Environment**).
- **tan_solid—**Displays selected tangent edges regardless of the environment setting for tangent edges.
- **no_disp_tan**—Turns off the display of tangent edges.
- **tan_ctrln**—Displays tangent edges in centerline font.
- **tan_phantom**—Displays tangent edges in phantom font.

- **tan_dimmed**—Displays tangent edges in dimmed color.
- **save_environment**—Saves and uses the environment setting for newly created views within the drawing.

text_orientation

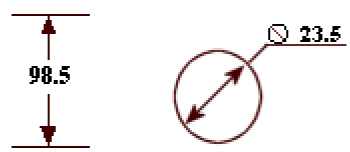
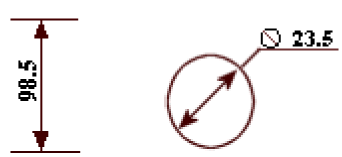
Controls orientation of dimension text in the drawing.

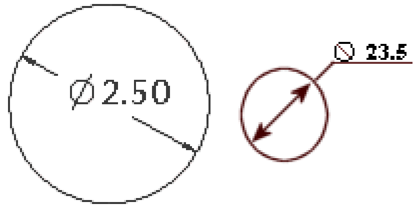
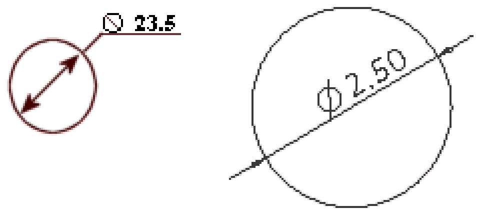
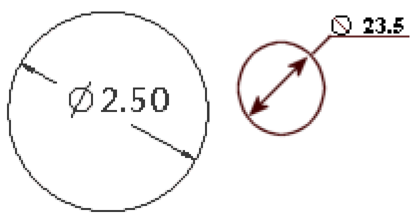
Note: The drawing setup file option `angdim_text_orientation` controls the display of angular dimensions (not text orientation).

Default and Available Settings

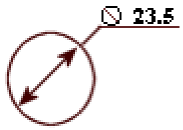
You can flip through the various arrow directions for each orientation setting using the **Flip Arrows** command on the shortcut menu or in the **Dimension Properties** dialog box (**Edit** > **Properties**). While moving a dimension, you can simultaneously right-click and cycle through the displays for the designated text orientation setting.

The following examples have the `default_dim_elbows` drawing setup file option set to `yes`.

horizontal*	Displays all dimension text horizontally. 
parallel	Displays text parallel to a dimension leader line. 
parallel_diam_horiz	Displays all dimensions except diameter dimensions parallel to their leaders; displays only diameter dimensions horizontally.

<table>
<tr><td></td><td>
</td></tr>
<tr><td>ISO _parallel</td><td>Displays text parallel to a dimension witness line. This differs from the value <code>parallel</code> as this will also allow dimension tolerances to be displayed in ISO 406:1987 (E) format or British Standard Tolerance Format.

</td></tr>
<tr><td>ISO_parallel_diam_horiz</td><td>Displays all dimensions except diameter dimensions parallel to their leaders. Limit tolerances will be stacked in accordance with ISO standard.

</td></tr>
</table>

Note: For the following arrow display, the elbow of a diameter dimension always extends through to the end of the text. To control the elbow extension beyond the text, use the `dim_text_gap` drawing setup file option.



text_thickness

Sets default text thickness for new text after regeneration and existing text whose thickness has not been modified. Type the value in drawing units.

Default and Available Settings

- **0.000000***
- **0<value<.5**

text_width_factor

Sets default ratio between the text width and text height. The system maintains this ratio until you change the width using the **Text Width** command.

Default and Available Settings

- **0.800000***
- **25<#>0.8**

thread_standard

Controls the display of threaded holes with an axis perpendicular to the screen as an arc (ISO standard), or as a circle (ANSI standard), or the hidden lines inside the threaded holes (JIS Standard).

Default and Available Settings

- **std_ansi***—Displays the threaded holes in accordance with the ANSI standard.
- **std_ansi_imp**—Does not display hidden thread lines when you select **No Hidden Line** from the **Display Style** list in the Pro/ENGINEER **Environment** dialog box that opens when you click **Tools** > **Environment**. When you select **Hidden Line**, thread lines are dispalyed as leader lines (yellow).
- **std_ansi_imp_assy**—Displays threaded holes in accordance with the ANSI standard.
- **std_iso**—Displays threads in cross-sectional assembly drawings in accordance with the ISO 6410 standard.

- **std_iso_imp**—Does not display hidden thread lines when you select **No Hidden Line** from the **Display Style** list in the Pro/ENGINEER **Environment** dialog box that opens when you click **Tools** > **Environment**. When you select **Hidden Line**, thread lines are displayed as leader lines (yellow).
- **std_iso_imp_assy**—Displays threads in cross-sectional assembly drawings in accordance with the ISO 6410 standard.
- **std_jis**—Displays threads in the top and side views of drawings in accordance with the JIS standard.

Note: The `std_iso` and `std_ansi` values are valid for drawings created before Pro/ENGINEER Release 15.0.

tol_display

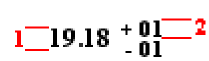
Controls display of dimension tolerances.

Default and Available Settings

- **no***—Does not display dimension tolerances.
- **yes—**Displays dimension tolerances.

Note: Drawing setup file options that have configuration options of the same name only override the configuration setting within drawings. The configuration setting remains the same within the model.

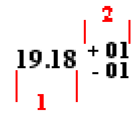
tol_text_height_factor

Sets the default ratio between the dimension text height and the tolerance text height when showing the tolerance in plus-minus and +- symmetric formats.	 1. Dimension text height 2. Tolerance text height

Default and Available Settings

- **standard***—Pro/ENGINEER uses 1 for the ANSI standard and .6 for the ISO standard. You can specify a value greater than zero.

tol_text_width_factor

Sets the default factor to maintain a proportion between the dimension text width and the tolerance text width when showing the tolerance in plus-minus and +- symmetric formats.	 1. Dimension text width 2. Tolerance text width

Default and Available Settings

- **standard***—Pro/ENGINEER uses .8 for the ANSI standard and .6 for the ISO standard. You can specify a value greater than zero.

use_major_units

Controls whether fractional dimensions are displayed in feet-inches within drawings.

This drawing setup option supersedes the configuration file option of the same name (`use_major_units`) unless the drawing setup option is set to default.

Default and Available Settings

- **default***—Displays fractional dimensions according to the setting of the configuration file option `use_major_units`.
- **yes—**Fractional dimensions are displayed in major units.
- **no—**Fractional dimensions are not displayed in major units**.**

Note: Drawing setup file options that have configuration options of the same name only override the configuration setting within drawings. The configuration setting remains the same within the model.

view_note

Determines the standard text requirements for view-related notes.

Default and Available Settings

- **std_ansi***, **std_iso**, and **std_jis—**Creates a view-related note with standard references, such as SEE DETAIL.
- **std_din—**Creates a view-related note with the words "SECTION," "DETAIL," and "SEE DETAIL" omitted.

Note: If you switch the setting from `std_ansi`, `std_iso`, or `std_jis` to `std_din`, the view related notes do not update.

view_scale_denominator

Determines denominator for the view scale before simplifying the fraction.

Default and Available Settings

- **0***—The default is 0. You can specify a different value, if required.

Note: If set to a positive integer and `view_scale_format` is a decimal, the view scale chosen for the first view of a model in the drawing is rounded to the nearest value with the specified denominator. If the view scale is so small that rounding would make the scale `0.0`, the value of `view_scale_denominator` is automatically changed by multiplying it by the smallest power of 10, which would give a positive value if rounding the scale down (although rounding up can happen). When you type a view scale value, you can round it to an allowable fraction.

The system does not round existing scale values after you have edited the setup file; they are approximate. Displays approximate scales preceded by a tilde (~), if you set the configuration file option `mark_approx_dims` to `yes`. If you set it to `0`, expresses the scale value in decimal format.

view_scale_format

Formats the view scale ratio.

Default and Available Settings

- **decimal***—Expresses the view scale as decimals.
- **fractional—**Expresses the view scale as fractions.
- **ratio_colon—**Displays the view scale values as a ratio. For example, instead of a view scale of 0.5, displays the view scale as 1:2. Since the ratio is just another way of displaying a fraction, make sure that you set the `view_scale_denominator` drawing setup option appropriately.

weld_solid_xsec

Determines if weld in cross section displays as solid (filled) region.

Default and Available Settings

- **no***—Weld cross sections do not display as solid regions.
- **yes—**Weld cross sections display as solid regions.

Note: This option does not apply to offset cross-sections.

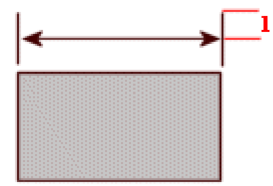
weld_symbol_standard

Controls the standard display for weld symbols in a drawings.

Default and Available Settings

- **std_ansi***—Applies the American National Standards Institute (ANSI).
- **std_iso—**Applies International Organization for Standardization (ISO).

witness_line_delta

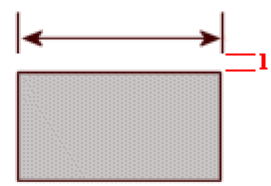
Sets the extension distance of the witness line (1) from the dimension leader arrow.	

Default and Available Settings

- **0.125000***—The default value is 0.125000. You can specify a different value for the distance between the witness line and dimension leader arrow.

Note: The `drawing_units` drawing setup file option determines the unit of measurement for the `witness_line_delta` drawing setup file option.

witness_line_offset

Sets offset between (1) a dimension line and object being dimensioned.	

The offset may only be visible when you plot a drawing. To see the effect, use the screen plot.

Default and Available Settings

- **0.062500***—The default is 0.62500. You can specify a different value for the offset between the dimension line and the object being dimensioned.

Note:

- The `drawing_units` drawing setup file option determines the unit of measurement for the `witness_line_offset` drawing setup file option.
- When you specify an offset value greater than the distance between the end of the intersection entity and the intersection, the length of the intersection witness line becomes zero as shown in the following figure:

yes_no_parameter_display

Controls display of `yes/no` parameters in drawing notes and tables.

Default and Available Settings

- **true_false***—Parameters can have a "true" or "false" value.
- **yes_no—**Parameters can have a "yes" or "no" value in drawing notes.

Drawing Configuration Options

About Drawing Related Configuration Options

Detailed Drawings configuration options enable you to customize your drawing environment.

Your drawing related configuration options, like all Pro/ENGINEER configuration options:

- Are set from the **Options** dialog box (**Tools** > **Options**).
- Are stored in a `config.pro` file.
- Use the default value unless you manually set the configuration option.

You can set and save multiple combinations of configuration options (`config.pro` file), with each file containing settings unique to certain design projects.

Detailed Drawings Help lists the configuration options unique to drawings. The options are arranged in alphabetical order. Each topic contains the following information:

- Configuration option name.
- Brief description and notes describing the configuration option.
- Default and available variables or values. All default values are followed by an asterisk (*).

allow_move_attach_in_dtl_move

Controls the working of the **Move** and **Move Attach** commands in conjunction with each other.

Default and Available Settings

- **no***—Move and Move Attach commands do not work together.
- **yes**—Move and Move Attach commands in drawing mode act together.

allow_move_view_with_move

Controls the movement of drawing views with the mouse.

Default and Available Settings

- **no***—Locks all views from mouse movement.

- **yes**—Allows the movement of views with the mouse.

auto_constr_offset_tolerance

Sets the auto constraint tolerance for creating offset dimensions. If distance is less than this tolerance multiplied by component size, offset is set as coincident.

Default and Available Settings

- **default**—Default value is 0.5. You can specify a different value if required.

allow_refs_to_geom_reps_in_drws (not in file)

Allows you to create drawing references to geometry representations (includes dimensions, notes, and leaders). These references may become invalid if the referenced geometry changes, resulting in geometry representations not updating in drawings.

Defaults and Available Options

- **yes**—Lets you create drawing references to geometry representations.

auto_regen_views

Note: When you regenerate a parent view, its child views do not automatically regenerate; you must individually select each view on the drawing, including detail views. Whenever you save changes to the model, Pro/ENGINEER displays them on the drawing the next time that you retrieve it, regardless of whether you regenerated the drawing views.

Default and Available Settings

- **yes***—Automatically repaints the display when changing from one window to another. Pro/ENGINEER automatically updates the drawing display by a repaint when you change from one window to another. For example, when you modify a model in a subwindow while you are working on a drawing in the main window. You can repaint or regenerate the drawing to reflect changes made to the model. When you regenerate it, Pro/ENGINEER updates the model to reflect the changes made in the drawing.
- **no**—Update the drawing, do not repaint the window. You can update only the drawing by choosing **Update** from the **View** menu, and then selecting **Drawing Views**, **Current Sheet**, or **All Sheets**. Neither the **Update** command in the **View** menu nor the **Regenerate** command in the **Edit** menu updates the drawing when you have this option set to `no`, even if you make the change to the model in Drawing mode (such as modifying a dimension value). You can select as many views as you want to regenerate at the same time.

  If you try to modify a view that you have not updated, Pro/ENGINEER displays an error message that it is not going to make changes to the drawing until you apply the **Update** or **Regenerate** command to the view.

Note: The `auto_regen_views` configuration option applies only to a view that is wireframe, hidden, or no-hidden. For a shaded view, the view is updated regardless of the setting for this configuration option.

auto_show_3d_detail_items

Determines whether or not 3D detail items are shown during drawing view creation.

Default and Available Settings

- **yes***—3D detail items with defined annotation planes are automatically shown in drawing views on view creation. However, annotation elements on these annotation planes must be parallel to the drawing view to be displayed automatically.
- **no**—3D detail items are not shown in drawings upon view creation.

bom_format

Sets the BOM format file to be used for a customized BOM. You must specify name and path.

Default and Available Settings

Enter name and path for BOM format file.

chamfer_45deg_dim_text

Controls the display of chamfer dimension text without affecting the leader. This only affects the text of newly created dimensions. ASME/ANSI is the default.

Default and Available Settings

- **asme/ansi***—
- **iso/din**—
- **jis**—

create_drawing_dims_only

Designates where to save new dimensions created within the drawing.

Default and Available Settings

- **yes**—Save all new dimensions created in the drawing inside the drawing as associative draft dimensions.
- **no***—Save all dimensions created in drawing mode in the part.

Note: If you set the `draw_model_read_only` configuration option to `cosmetic_only`, make sure that the `create_drawing_dims_only` configuration option is set to `no`.

def_layer

Specifies default layer names for different types of items. The first value string is the layer type. The second value string is the layer name. For example, `layer_dim_my_layer`.

Default and Available Settings

`<option name>_<layer name>`

The following settings are related to default layers for drawings:

- layer_all_detail_items
- layer_assem_member
- layer_assy_cut_feat
- layer_axis
- layer_chamfer_feat
- layer_comp_design_model
- layer_comp_fixture
- layer_comp_workpiece
- layer_copy_geom_feat
- layer_corn_chamf_feat
- layer_cosm_round_feat
- layer_cosm_sketch
- layer_csys
- layer_curve
- layer_curve_ent
- layer_cut_feat
- layer_datum
- layer_datum_plane
- layer_datum_point
- layer_detail_item
- layer_dgm_conn_comp
- layer_dgm_highway
- layer_dgm_rail
- layer_dgm_wire

- layer_dim
- layer_draft_constr
- layer_draft_dim
- layer_draft_dtm
- layer_draft_entity
- layer_draft_feat
- layer_draft_geom
- layer_draft_grp
- layer_draft_hidden
- layer_draft_others
- layer_draft_refdim
- layer_driven_dim
- layer_dwg_table
- layer_ext_copy_geom_feat
- layer_feature
- layer_geom_fea
- layer_gto
- layer_hole_feat
- layer_nogeom_feat
- layer_note
- layer_parameter_dim
- layer_part_refdim
- layer_point
- layer_protrusion_feat
- layer_quilt
- layer_refdim
- layer_ribbon_feat
- layer_rib_feat
- layer_round_feat
- layer_sfin
- layer_shell_feat

- layer_skeleton_model
- layer_slot_feat
- layer_snap_line
- layer_solid_geom
- layer_surface
- layer_symbol
- layer_thread_feat
- layer_trim_line_feat
- layer_weld_feat

default_ang_dec_places

Controls the default number of digits (0-14) for angular dimensions created in a drawing. This option also controls angular reference dimensions created in any mode.

Default and Available Settings

Type a value for the number of decimal places shown. The default is 1.

default_draw_scale

Sets the default drawing scale for views added with the No Scale command. The value must be greater than 0.

Default and Available Settings

- **-1.000000***—The default scale for views added with the No Scale command.
- **no**—The system does not set a default drawing scale.

default_font_kerning_in_drawing

Determines the initial setting of font kerning when 2D drawing annotations are created. Kerning reduces the space between certain pairs of characters, improving the appearance of the text string.

Default and Available Settings

- **no***—Font kerning is not available for new annotations in a drawing.
- **yes**—Font kerning is available for new annotations in a drawing.

  If `default_font_kerning_in_drawing` configuration option is set to `yes`, then by default the **Kerning** check box in the **Text Style** dialog box is selected for newly created annotations. The **Text Style** dialog box opens when you click **Format** >

Text Style. The `default_font_kerning_in_drawing` configuration option is not only applicable to new annotations created in the drawing, but also to the new text styles created from the **Text Style Gallery** dialog box that opens when you click **Format** > **Text Style Gallery**. Setting this option to `yes`, does not change the characteristics of already created notes when they are redefined.

disp_trimetric_dwg_mode_view

Displays the model in default orientation when placing a general view.

Default and Available Settings

- **yes***—Displays the model in default orientation when placing a general view on the drawing.
- **no**—Model does not appear until an orientation is chosen from the Orientation dialog box.

display_dwg_sketch_constraint

Enables the display of parametric sketching constraints when a drawing object is selected.

Default and Available Settings

- **yes**—Parametric sketching constraints, such as V for vertical, are displayed when a drawing object is selected.

display_dwg_tol_tags

Sets the display of tolerance tags in drawings. Does not control tolerances on dimensions.

Default and Available Settings

- **yes***—Displays tolerance tags in drawings.
- **no**—Does not display tolerance tags in drawings.

display_in_adding_view

Controls model display when `auto_regen_views` is `no`.

Default and Available Settings

- **Default**—Displays using the environment setting.
- **Wireframe**—Adds new views in wireframes, displays datums.
- **Minimal_wireframe**—Displays in wireframe, no datums, axes, or silhouette edges.

draw_models_read_only

Determines whether drawing changes results in design changes in the model.

Default and Available Settings

- **no***—Drawing model files are not read only; applicable drawing changes result in design changes in the model.
- **yes**—Drawing model files are read-only. You cannot add driven dimensions, geometric tolerances, or similar items to the views.
- **cosmetic_only**—Drawing model files are read-only. You can add driven dimensions, geometric tolerances, cross sections, and explode states, however, they do not cause design changes in the model.

Note: Make sure the `create_drawing_dims` configuration option is set to `no` when setting this configuration option to `cosmetic_only`.

The following commands will be allowed when `draw_models_read_only` is set to `cosmetic_only`. These commands cause iteration of the model and require the model to be saved:

- Creating and modifying driven dimensions (i.e. dimensions created in the drawing)
- Creation and modification of an assembly explode state
- Creation and regeneration of a cross-section. Note: The revision number of the design does increase after the model is saved.
- Modifying the attachment of a set datum. This applies to both driven and driving dimensions.

The following commands will be allowed when `draw_models_read_only` is set to `cosmetic_only`. These commands do not cause iteration of the model:

- Modifying layers and layer display
- Modifying the line style of a three dimensional solid model (**View** > **Drawing Display** > **Component Display**).

  Note: Any changes that are made to the part or assembly outside of drawing mode are not prevented by this configuration option.
- Moving the position of a set datum tag
- Extending or trimming a model axis
- Switching a detail item from one view to another
- Creating and modifying a geometric tolerance that is created in the drawing and only references the drawing
- Creating and modifying two dimensional notes in a drawing
- Creating and modifying two dimensional entities in a drawing

drawing_ole_image_dpi

0-600 dots per inch (DPI)

Controls the image resolution of object linking and embedding (OLE) objects when exporting or saving the drawings as picture files or read-only drawings. The file size increases with the increase in the DPI value.

`0`—OLE objects are not printed.

draw_points_in_model_units

Defines the current draft views coordinate values as model units rather than drawing units.

drawing_file_editor

Specifies the default text editor.

drawing_setup_file

Sets the default drawing setup file option values for your Pro/ENGINEER session, otherwise, the system uses the default drawing setup file option values.

drawing_shaded_view_dpi

0-600 dots per inch (DPI)

Controls the image resolution of shaded views when exporting or saving the drawings as picture files or read-only drawings. The file size of the image increases with the increase in the DPI value.

`0`—Shaded views are not printed.

drawing_view_origin_csys

The named coordinate system will be used as the origin of a newly created view.

Default and Available Settings

- **none**—The system will not use a previously specified coordinate system.
- **#string**—The system uses the named coordinate system.

enable_shaded_view_in_drawings

Enables the display of shaded views in drawings when the environment setting is Shading or when the view display style is set to Shading. If this configuration option is set to `no`, views are displayed in wireframe and the **Shading** option in the **Display Style** list in the **Drawing View** properties dialog box is not available for selection.

Default and Available Settings

- **yes***—Shaded views are allowed in the drawing.
- **no**—View display is in wireframe.

Note: If you set the `enable_shaded_view_in_drawing` configuration option to `no` and the `model_display_for_new_views` drawing setup file option to `shading`, then the actual display of the drawing view is in wireframe.

force_wireframe_in_drawings

Controls the display of drawing views.

Default and Available Settings

- **yes**—Displays all views in wireframe.
- **no***—Displays views according to display set for the session.

format_setup_file

Assigns a specified setup file to each drawing format. To assign the drawings parameter values to the format, you must retrieve the drawings setup file into the format.

general_undo_stack_limit

Limits the number of possible Undo/Redo operations stored in the stack (memory). If the limit is reached, the first operation that was added to the stack will be the first operation removed.

Default and Available Settings

- **50***—Limit the undo/redo mechanism to 50 operations. You can type a maximum storage value for the undo/redo operation.

harn_tang_line_display

Display the tangency lines between cables in thick cable display mode.

Default and Available Settings

- **yes***—Displays the tangency lines for thick cables.
- **no**—Does not display the tangency lines for thick cables.

highlight_erased_dwg_views

Controls the display of erased view outlines (environment option Highlight Erased Views.)

Default and Available Settings

- **no**—Turns off highlighting for erased views.

highlight_new_dims

In Drawing mode, highlights new dimensions in dark red until you repaint the screen.

Default and Available Settings

- **no***—Displays new dimensions in their default color.
- **yes**—Highlights new dimensions in dark red.

hlr_for_quilts

Controls quilt display in hidden line removal.

Default and Available Settings

- **yes**—Includes quilts in hidden line removal process.

hlr_for_xhatches

Controls the display of crosshatches in hidden line removal.

Default and Available Settings

- **yes**—Includes crosshatches in the hidden line removal process.

iges_in_dwg_line_font

Controls the import of user-defined line fonts. Pro/ENGINEER processes the IGES line font definition entity (type 304, form 2) when you set these options to `yes`.

Default and Available Settings

- **no***—Imports the user-defined line fonts as solid line fonts.
- **yes**—Gives default names to user-defined line styles without names in order (IGES_1, IGES_2 and so on.).

iges_out_dwg_line_font

Controls the export of user-defined line fonts through IGES. Pro/ENGINEER processes the IGES line font definition entity (type 304, form 2) when you set these options to `yes`.

Default and Available Settings

- **no***—Exports all geometry as solid fonts.

make_parameters_from_fmt_tables

Determines the handling of values entered in a format table when you replace one drawing format with another.

Default and Available Settings

- **yes**—Stores entered values for format table and copies them with the table.
- **no***—Prompts you to re-enter all the values for the format table.

make_proj_view_notes

Automatically adds view names to projection views in a specified format. The default format is `<VIEW view_name-view_name>`. You can modify the view name after adding the view to a drawing.

Default and Available Settings

- **no***—View names are not added to projection views automatically.
- **yes**—View names are added to projection views automatically.

open_draw_simp_rep_by_default

Set to `yes` to always open the **Open Rep** dialog box when opening a drawing.

Default and Available Settings

- **no***—Opens a drawing directly when you click **File** > **Open**, without opening the **Open Rep** dialog box.
- **yes**—Opens the **Open Rep** dialog box when you try to retrieve a drawing using **File** > **Open**.

pick_chain_tangent_only

Specifies the extent of draft entity chains selected.

Default and Available Settings

- **yes**—Only entities in the chain that are tangent will be selected.
- **no***—All entities connected end to end will be selected.

pro_dtl_setup_dir

Set the directory for the drawing setup files. If this option is not set, Pro/ENGINEER uses the default setup directory.

pro_format_dir

Sets the default directory for the drawing format library. Use the full pathname to avoid problems.

pro_note_dir

Specifies the directory from which to retrieve notes entered from a file. Use the full path name to avoid problems.

pro_palette_dir

Sets the default directory for drawing symbol palette files.

pro_symbol_dir

Set and automatically create the default directory for saving and retrieving user-defined symbols. If you do not specify a directory, the system uses the current working directory. Use full path name to avoid problems.

remember_last_get_point_option

Sets the default of the **Get Point** menu as the last option picked among **Pick Pnt**, **Vertex**, and **On Entity**.

Default and Available Settings

- **yes**—Sets the default of the **Get Point** menu as the last option picked among **Pick Pnt**, **Vertex**, and **On Entity**.

rename_drawings_with_object

Creates a copy of the associated drawing when you use **File > Save a Copy** to copy a model or an assembly. The new drawing adopts the new part filename or assembly filename.

Default and Available Settings

- none*
- part
- assembly
- both

restricted_gtol_dialog

Controls the restrictions in the Geometric Tolerance Dialog.

Default and Available Settings

- **yes***—The dialog will adhere to standards when picking certain GTOL types.
- **no**—The dialog drops all restrictions.

save_display

Determines whether to store view geometry and detail items such as solid dimensions in the View-Only mode.

Default and Available Settings

- **no***—Does not display geometry and detail items in View-Only mode.
- **yes**—Stores view geometry and detail items such as solid dimensions. These items are displayed when retrieving the drawing in View-Only mode.

Note: You can change this setting at run time: click **Utilities** > **Environment**, and then click or clear **Save Display** under **Default Actions** in the Environment dialog box.

save_drawing_picture_file

Default and Available Settings

- **no***—Does not embed or save the drawing as a picture file.
- **Embed**—Embeds a picture file inside a drawing for preview purposes.
- **Export**—Saves a drawing file as a picture file in the current working directory when saving a drawing.
- **Both**—Does both embed and export.

save_modified_draw_models_only

Determines whether the system saves the model after you have changed it.

Default and Available Settings

- **no**—Saves the model every time that you store the drawing.

show_preview_default

Determines the default behavior for preview in the Show/Erase dialog.

Default and Available Settings

- **remove***—Sets the default to be Sel to Remove.
- **keep**—Sets the default to be Sel to Keep.

select_hidden_edges_in_dwg

Controls the selection of hidden edges in drawings.

Default and Available Settings

- **yes***—Allows you to select hidden edges within drawings.
- **no**—Disallows selecting hidden edges by rejecting edges behind the first surface at the selection point.

selection_of_removed_entities

Default and Available Settings

- **yes**—Entities in front of cross section (planar or offset) can be selected, clipped (using Z-clipping) or erased using **EDGE DISP** menu.
- **no***—Geometry, datum points or curves, cosmetic features, threads, grooves, or coordinate systems cannot be erased.

switch_dims_for_notes

Controls the display of dimensions during drawing note creation.

Default and Available Settings

- **yes***—Dimensions are displayed in their symbolic format during drawing note creation.
- **no**—Dimensions use numeric values.

symbol_instance_palette_file

Specifies the location of the symbol instance palette.

symbol_palette_input

Controls the display of the special text symbol palette during note creation.

Default and Available Settings

- **yes***—**Text Symbol** palette is displayed during note creation.
- **no**—**Text Symbol** palette is not displayed during note creation.

text_symbol_palette_file

Sets the path of the palette layout file. Use the full path name. For example, if you have a file with the name `my_custom_palette.txt` located at `D:\my_proe_files\palette`, you must set the path as `D:\my_proe_files\palette\my_custom_palette.txt`.

Note: To use a different palette, you have to change the path in the configuration option to point to the palette you want, exit the Pro/ENGINEER session and restart it, for the changed symbol palette to be available.

todays_date_note_format

Controls the initial format of the date displayed in a drawing. The format for the setting is a string consisting of three portions: the year, the month, and the date. You can enter the portions in any order.

triangulate_filled_areas

Subdivides filled areas of drafted entities into triangles. Setting this option to `yes` may affect memory usage and size of the plot file.

Default and Available Settings

- **no***—Does not subdivide filled areas of drafted entities into triangles.
- **yes**—Subdivides filled areas of drafted entities into triangles.

variant_drawing_item_sizes

Controls the size and position of drawing items when they are moved or copied to a different location on the sheet or on the paper.

Default and Available Settings

- **no***—Drawing items moved or copied to a different sheet or located on a changed sheet keep same size and relative orientation on paper.
- **yes**—Some items scale or reposition to be the same on paper, and others scale and/or reposition to be the same on screen.

warn_if_ISO_tol_missing

Controls the display of a warning message in the **Invalid ISO Tolerance** dialog box. This dialog box is displayed when Pro/ENGINEER validates an ISO tolerance table and finds a missing tolerance value in the selected table.

Clicking **Yes** in the **Invalid ISO Tolerance** dialog box opens the **Dimension Properties** dialog box from where you can select a different ISO tolerance table. Clicking **No** applies the existing tolerances.

Default and Available Settings

- **yes***—Pro/ENGINEER displays a warning message if a tolerance value in the selected table is missing.
- **no**—Pro/ENGINEER does not display a warning message.

Note: When regenerating the model or the drawing, Pro/ENGINEER saves a warning message in a log file for each dimension that has no corresponding tolerance value in the ISO tolerance table. After regenerating the model or the drawing, you can access this log file by clicking **Info** > **Session Info** > **Message Log**. You can print this log file and use it to manually fix the dimensions.

Templates for Drawing Layout

About Drawing Templates

Drawing templates may be referenced when creating a new drawing. They automatically create the views, set the desired view display, create snap lines, and show model dimensions based on the template.

Drawing templates contain three basic types of information for creating new drawings. The first type is basic information that makes up a drawing but is not dependent on the drawing model, such as notes, symbols, and so forth. This information is copied from the template into the new drawing.

The second type is instructions used to configure drawing views and the actions that are performed on that view. The instructions are used to build a new drawing with a new drawing object (model).

The third type is parametric notes. Parametric notes are notes that update to new drawing model parameters and dimension values. The notes are re-parsed or updated when the template is instantiated.

Use the templates to:

- Define the layout of views
- Set view display
- Place notes
- Place symbols
- Define tables
- Create snap lines
- Show dimensions

You can also create customized drawing templates for the different types of drawings that you create. For example, you can create a template for a machined part versus a cast part. The machined part template could define the views that are typically placed for a drawing of a machined part, set the view display of each view (for example, show hidden lines), place company standard machining notes, and automatically create snap lines for placing dimensions. Creating a template allows you to create portions of drawings automatically, using the customizable template.

To Create a Drawing Template

1. Click **File** > **New**. The **New** dialog box opens.
2. Click **Drawing**, and then type the name of the template you are creating or accept the default.
3. Select the **Use default template** checkbox (selected by default), and then click **OK**. The **New Drawing** dialog box opens.
4. Click **Empty** or **Empty with format**.

   If you click **Empty with format:** Under **Format**, specify the format you want to use. Then click **OK**. Pro/ENGINEER creates a new drawing with the specified format.

   If you click **Empty:** Under **Orientation**, specify the template orientation by clicking **Portrait**, **Landscape**, or **Variable**. For **Portrait** or **Landscape**, choose a standard size under **Size**. For **Variable**, specify a size using the **Width** and **Height** boxes, and specify a unit by selecting **Inches** or **Millimeters**.
5. Click **OK** to create the template.
6. Click **Applications** > **Template** to enter drawing template mode
7. Click **Insert** > **Template View**. The **Template View Instructions** dialog box opens.
8. Type the view name or accept the default, and then specify the view orientation.
9. Specify view options and view values in the **View Options** and **View Values** areas.
10. Click **Place View** and select the location of the General view.

    Note: After you place the view, you now have the options to move the symbol, edit the view symbol, or to replace the view symbol.
11. To place additional views, click **New**, type the new view name, and orient the new view. Specify the view options and view values of the new view.
12. When you are done placing all of the desired views, click **OK**. Save the template.

To Create a Drawing Using a Drawing Template

1. Click **File** > **New**. The **New** dialog box opens.
2. Click **Drawing**, and then type the file name for the drawing you are creating or accept the default name.
3. Clear the **Use default template** checkbox, and then click **OK**. The **New Drawing** dialog box opens.
4. Select the model from which you want to create the drawing.

5. Specify the template by clicking **Use template**. Type the name of the template you want to use or select a template from the **Template** list. Click **OK**. The drawing is created.

Note: The views with the correct attributes in both the template and the model are created. If attributes that are defined in the template do not exist in the model, errors occur when the drawing is being created. The **Drawing Template Error Info** dialog box opens and lists the errors.

To access the **Drawing Template Error Info** dialog box, click **Info** > **Template Errors**.

About the Template View Instructions Dialog Box

You can customize drawing templates by using the **Template View Instructions** dialog box.

You can access the **Template View Instructions** dialog box from the **TMPLT DWG** menu (on the Menu Manager), that appears when you click **Applications** > **Template** in the Drawing mode. For more information about Drawing template mode, click To Create a Drawing Template in Pro/HELP.

Use the following options in the **Template View Instructions** dialog box to customize your drawing templates:

- **View Name**—Set the name of the drawing view that will be used as the view symbol label.
- **View Type**—Set the type of the drawing view.

View Options	View Values
View States—Specify the type of view. By default, the **View States** check box is selected and the view state is displayed when the **Template View Instructions** dialog box opens. The value of **Orientation** is `FRONT`, by default. If you select **Combination State**, the **Orientation**, **Simplified Rep**, **Explode** and **Cross Section** boxes display the text `Defined by Combination State`. The **Arrow Placement View** and **Show X-Hatching** check boxes also become available for selection after you specify the Combination State.	Combination State, Orientation, Simplified Rep, Explode, Cross Section, Arrow Placement View
Scale—Type a new value for the scale or use the default value.	View Scale
Process Step—Set the process step for the view. You can specify the step number and set the view of the tool by checking the **Tool View** check box under **Process Step**.	Step Number, Tool View

View Options	View Values
Model Display—Set the view display for the drawing view.	Follow Environment, Wireframe, Hidden Line, No Hidden, Shading
Tan Edge Display—Set the tangent edge display.	Tan Solid, No Disp Tan, Tan Ctrln, Tan Phantom, Tan Dimmed, Tan Default
Snap Lines—Set the number, spacing, and offset of the snap lines.	Number, Incremental Spacing, Initial Offset
Dimensions—Show dimensions on the view.	Create Snap Lines, Incremental Spacing, Initial Offset
Balloons—Show balloons on the view.	

- **Place View**—Places the view after you have set the appropriate options and values. Using **Place View** you can define a bounding box for the view by selecting a point on the drawing sheet and dragging. The location and the scale of the view is controlled by the bounding box. After you place the bounding box on the drawing sheet, the view symbol is automatically placed at the center of the bounding box. You can move the bounding box using the **Move Special** command on the shortcut menu.

  Note:

  - The bounding box is not available if you have defined a scale for the view in the **Template View Instructions** dialog box.
  - The bounding box is available only in the Template mode.
  - If you have placed a bounding box on the sheet and you try to set the scale, the bounding box disappears and the view reverts to a standard view.
  - The view boundary cannot be changed after the view is placed.

- **Edit View Symbol**—Allows you to edit the view symbol using the **Symbol Instance** dialog box.
- **Replace View Symbol**—Allows you to replace the view symbol using the **Symbol Instance** dialog box.

Creating and Editing Sheet Formats: Using Formats in a Drawing

About Using Formats in a Drawing

The format of a drawing refers to the boundary lines, referencing marks and graphic elements that appear on every sheet before any drawing elements are shown or added. These usually include items such as tables for the company name, detailers name, revision number and date.

When you start a new Pro/ENGINEER drawing, you are prompted for a format file (`.frm`) to associate with the drawing. This file carries all the format graphical information, and it can also carry some optional default attributes like text size and draft scale. For a multisheet drawing, you can have two default formats—one for the first sheet and another for the remaining sheets.

Pro/ENGINEER ships with several standard drawing formats for various sheet sizes. You can also create and save your own format files.

You can change the format on any sheet (including the first sheet) independently of the formats on other sheets; therefore, you could use a different format on each sheet of the drawing.

Globally Updating an Associated Format

To replace a format on all existing sheets of a drawing use the **Page Setup** dialog box, that opens when you click **File** > **Page Setup**. Alternately, you can edit the existing associated `.frm` file with the changes you want to apply. The changes will appear the next time the `.drw` file is opened.

The Format Setup File

Like a drawing file, a format file retrieves default values from a `.dtl` setup file. If you want your format `.dtl` file to have the same values as your drawing setup file, retrieve the drawing setup file into the format using the **File** > **Properties** command. Use the **Open** icon in the dialog box to retrieve the `.dtl` file. The system reads only those options valid for formats.

To Import a Format from a Legacy System

To reuse an existing format (that is, one that was created in another system), use the interface options (such as DXF, SET, IGES, TIFF, and so on) to import it into your format.

To Create a Format

1. From the Pro/ENGINEER menu bar, choose **File** > **New**.
2. In the **New** dialog box, click **Format** and type a format name in the **Name** box, or accept the default name. Click **OK**.
3. In the **New Format** dialog box, specify the format size by doing one of the following:

- Use the **Specify Template** field to attach a template to the format if desired.
- Under **Orientation**, click **Portrait**, **Landscape**, or **Variable**.
- If you selected **Portrait** or **Landscape**, select a size from the **Standard Size** list.
- If you selected **Variable**, you must define both the height and width dimensions. Select **Inches** or **Millimeters** and type values in the **Width** and **Height** boxes.

4. Click **OK**. The system displays the format as specified, and the FORMAT menu appears.

To Create or Modify Format Geometry

To create or edit format geometry in a .frm file, use the **Sketch** menu, or use any of the sketcher icons in the side menu.

To modify the values of symbol text, symbol height, and note text in a standard drawing format, use the same commands that are available during format creation.

To Change Formats in an Existing Drawing

1. Click **File** > **Page Setup**. The **Page Setup** dialog box opens.
2. Use the drop-down list to the right of each sheet number to assign a format style.

   To set a custom size, use the **Variable Size** setting in the list box. To load a format not listed, use the **Browse** setting.

   To show or hide the format for a specific page, use the **Show format** checkbox.

To Hide the Format

1. Click **File** > **Page Setup**.
2. In the **Page Setup** dialog box, clear the **Show format** checkbox.
3. Click **OK**.

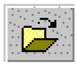
Tip: Sheet Outline and Plotting

The sheet outline is the border of the standard drawing format you selected. Because it is the actual border, it may not appear on pen plots unless you use a paper size larger than the drawing size. Everything within the sheet outline border is also plotted, but you should make an allowance for the plotter hold-down rollers.

To Match Setup Values in Drawings and Formats

Like a drawing file, a format file retrieves default values from a `.dtl` setup file. If you want your format .dtl file to have the same values as your drawing setup file, you can retrieve the drawing setup file into the format.

With the format file open:

1. Click **File** > **Properties**.
2. On the menu manager, click **Drawing Options.** The **Options** dialog box opens.
3. Click  to open the target `.dtl` file. The system reads in only those options valid for formats.
4. Click **OK**.

About Using Tables in a Format

Using the **Table** command in the FORMAT menu, you can create tables and add them to drawing formats. Then, when you add a format to a drawing, Pro/ENGINEER copies all of the tables in the format into the drawing. The copied tables are independent of the original format, and you can move, modify, or delete them.

When you add, replace or remove a format, (using **File** > **Page Setup**), the system highlights the table copied from the old format and asks if it should remove it. You can choose one of the following: (**Y**)es, (**N**)o, (**K**)eep all, (**R**)emove all.

If you add a table to a format, the drawings that already reference this format do not display the new table. The table must be part of the format when you initially copy it into the drawings. To use the modified format in the drawing, you must use the replace function.

When you add a format to a drawing that contains the format table, the system stores the values you specified for the table as drawing parameters, if you have set the configuration file option `make_parameters_from_fmt_tables` to `yes`. You can access the parameters using the **Tools > Parameters and Relations** command.

If you set this configuration option to `no`, when you add a format to a drawing, the system prompts you to retype all of the values every time, and it does not evaluate the values on subsequent sheets of the drawing in the format table. These values are nonparametric text. For the system to reprompt you for the parameter text, you must use the **Add or Replace** commands.

Note: You cannot use reserved model parameters. If you set `make_parameters_from_fmt_tables` to `yes` in the configuration file, Pro/ENGINEER does not prompt you to type a value for reserved model parameters because it cannot add them to the drawing.

If you change the format size of a drawing with a table, the system scales the table (in the same way that it scales any views and draft entities) to maintain its location relative to other items and the sheet boundaries. Since the drawing setup file option `drawing_text_height` controls the height of any text in the table, Pro/ENGINEER does not scale text that you include in the table. Therefore, it does not maintain a size that is proportional to the rest of the table. You must modify the text height manually.

About Notes in Formats and Tables

When you place a parametric note in a format, the note in the format acquires the appropriate value when you use it in a drawing. For example, if you create the note `&model_name` in a format, the system displays the actual model name in the note.

For the system to update these parameters when you add the format to a drawing, you can include in parametric notes only the drawing labels listed in the topic titled System Parameters for Drawings (except `&todays_date`). You must include all user-defined model and drawing parameters in format table cells in the form of `&param` in order for the system to update them in a drawing. Format mode interprets the following types of parametric notes:

- Notes with symbol instances
- Notes with standard system symbols
- Notes with drawing labels
- Notes with a default tolerance

The following rules apply to including parameters as labels in format tables:

- Pro/ENGINEER correctly evaluates parametric labels that you include in a format table only if you create the drawing first, add the model, and then add the format. It does not evaluate the labels correctly if you create the drawing, add the format, and then add the model.
- When you add a format to a drawing with more than one model present, parametric notes can reference only the active drawing model.
- When you add the format to a drawing, Pro/ENGINEER parses any and all that are present of the standard parametric symbols that it supports and displays the correct values in the table. It does this for every sheet of the drawing on which you use the format, so that the drawing name, model name, or any other standard parameter that you used appears on every sheet that uses that format.

To Add a Table to a Format

Using the **Table** command in the FORMAT menu, you can include tables in drawing formats. When you add a format to a drawing, Pro/ENGINEER copies all of the tables in the format into the drawing. After it copies them, the tables are independent of the original format, and you can move, modify, or delete them.

1. Make sure that the format file to which you want to add a table is the active file.
2. On the **FORMAT** menu, click **Table** > **Insert** > **Table**. The **TABLE CREATE** menu opens on the Menu Manager.
3. Click from the following options:
    - **Descending** or **Ascending**—Lets you choose the direction in which to create rows from the starting point.
    - **Rightward** or **Leftward**—Lets you choose the direction in which to add columns from the starting point.

  - **By Num Chars** or **By Length**—Lets you define the length of columns and rows using drawing units (such as inches or millimeters) or by specifying the number of characters as the unit.

4. Use the options on the **GET POINT** menu to locate the starting corner of the table on the format. The starting corner depends on the choices you made in the preceding step.
   - **Pick Pnt**—Lets you select a point on the format and specify it as the starting corner.
   - **Vertex**—Lets you select a vertex and specify it as the starting corner.
   - **On Entity**—Lets you select an entity on the format and specify it as the starting corner.
   - **Rel Coords**—Lets you specify relative coordinate values for placing the starting corner by entering relative values for the X- and Y-axes.
   - **Abs Coords**—Lets you specify absolute coordinate values for placing the starting corner by entering absolute values for the X- and Y-axes.
5. At the prompt, enter the width of the first column in drawing units, and then press ENTER.
6. Continue entering the width of additional columns until you have the number of columns you want. Then press ENTER again without typing another column width value.
7. At the prompt, enter the height of the first row in drawing units, and then press ENTER.
8. Continue entering values for the height of additional rows until you have the number of rows you want. Then press ENTER again without typing another row height value.

The table displays on the format sheet.

To Use Parameters in Labels in a Format Table

You can include parametric labels, such as the drawing name, model name, and sheet number, as text in a format table. For a multimodel drawing, you can type into the format table any parameters related to both the first model and the second model. To include the parametric labels in the table:

1. Double click the table cell where you want to place the label text. The **Note Properties** dialog box displays.
2. In the **Text** tab of the **Note Properties** dialog box, enter the appropriate label (such as `&dwg_name`). When you are done click **OK**. The label appears in the table exactly as you typed it until you add the format to a drawing.

To Set Up a Format Library

Use the configuration option `pro_format_dir` to set a path name (directory) where your company formats will be stored. When you use this configuration option, Pro/ENGINEER automatically searches the specified directory for company formats when adding or replacing formats in your drawing and layout. Pro/ENGINEER places modified formats in this directory when you save them.

1. Click **Tools** > **Options**. The **Options** dialog box opens.
2. Enter the configuration file option `pro_format_dir`. This option uses a path name (directory) as its value, so you can create a single set of formats that everyone on the system can use, and place them all in a single directory.
3. Under **Value**, enter the path name and directory where you want to store all formats.

To Retrieve a Format from the Format Library

Using the **File Open** dialog box, you can retrieve formats from a format library directory within Pro/ENGINEER. To retrieve a format:

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **System Formats** from the **Look In** list. Navigate the menu tree until you locate the format.
3. Click **Open**. The selected format opens.

Creating a Drawing

To Create a Drawing

When you start a new drawing, you specify a 3D model file for which to place drafting views.

1. Click **File** > **New**.
2. In the **New** dialog box, click **Drawing** and type a drawing name in the **Name** box (or use the default), and click **OK**. The **New Drawing** dialog box opens.
3. In the **Default Model** box, type the name of a model in the working directory. If you started the new file from an open 3D file, the 3D filename is entered by default. The selected model is set as the current drawing model.
4. Under **Specify Template**, do one of the following:
    - To use a Pro/ENGINEER drawing template, click **Use Template** and select a template from the list.
    - To create a drawing without a template but with an existing format, click **Empty with format**. Under **Format**, specify the format you want to use.
    - Specify the drawing size or retrieve a format. To specify the size, do one of the following:

Click **Portrait** or **Landscape** in the **Orientation** box and select a standard size from the **Standard Size** list. **Portrait** makes the height larger than the width, while **Landscape** makes the width larger than the height.

Alternatively,

Click **Variable** in the **Orientation** box to define both the height and width dimensions. Select **Inches** or **Millimeters** and type values in the **Width** and **Height** boxes.

To retrieve a format, select **Retrieve Format** and select a name from the **Name** list in the **Format** box. You can also type [?] or click **Browse** to select a name from the **Open** dialog box.

5. Click **OK**. Pro/ENGINEER displays the new drawing.

   Note:

   - If the part that you are using to create the drawing has part simplified representations, the **Open Rep** dialog box opens. Select the required representation and click **OK**. The new drawing is created with the selected representation set as the current representation of the drawing model.
   - The **Open Rep** dialog box does not open if you have used the default template to create the drawing. It opens only when you are creating a new empty drawing. If you are using the default template to create the drawing of a part that has part simplified representations, Pro/ENGINEER uses the default representation to create the drawing.
   - If a model has multiple instances defined with a family table, you are prompted to choose an instance from the **Select Instance** dialog box, before you select the representation using the **Open Rep** dialog box.

To Add Models to the Drawing

Before you can place a view of a 3D model, you must associate the 3D model with the drawing. This is called adding the model to the drawing.

When you start creating a new drawing file you are prompted in the **New File** dialog box for a 3D model to reference. Once the drawing is in progress, use the following procedure to add other models to the drawing file:

1. Click **File** > **Properties**. The **FILE PROPERTIES** menu appears.
2. Click **Drawings Models**. The **DWG MODELS** menu appears.
3. Click **Add Model**. The **Open** dialog box opens and displays all the files in your current working directory.
4. Select the model that you want to add as a drawing model to the current drawing, and click **Open**.

Note:

- If the new model that you are adding to the drawing contains part simplified representations, the **Open Rep** dialog box opens. Select the required representation and click **OK**.
- If the model that you are adding to the drawing has different instances of a family table, you are first prompted to select an instance from the **Select Instance** dialog box, and then the simplified representation from the **Open Rep** dialog box.

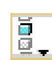
5. The model is added to the drawing and set as the current drawing model. If you have selected a representation in the **Open Rep** dialog box, the selected representation is set as the current representation of the drawing model.

Note: Adding a model to the drawing does not place a view of the model on the sheet, but lets the drawing reference the model so that you can place a view. You can insert drawing views of the newly added part as you would for any other drawing model.

To Set the Current Working Model

If a drawing has more than one model added to it, one of the models is always the current working model. Use this procedure to switch current status from one model to another.

1. Click **File** > **Properties**. The Menu Manager opens and displays the **FILE PROPERTIES** menu.
2. In the Menu Manager click **Drawing Models** > **Set Model**. The Draw Model menu opens.
3. Select the model from the namelist menu. It contains only model names that you have added to the drawing.

   Alternately, you can set the active model using the **Set Model** drop down list, found on the toolbar.

Note:

- To highlight the current active model, choose **File** > **Properties** > **Drawing Models** > **Model Disp** > **Hilite Cur**.
- To return the model display back to normal, choose **Model Disp** > **Normal**.

To Delete Models as Drawing Models from an Active Drawing

1. With a drawing open, click **File** > **Properties**. The **FILE PROPERTIES** menu appears.
2. Click **Drawing Models**. The **DWG MODELS** menu appears.
3. Click **Del Model**. The **DRAW MODELS** menu appears. This menu lists all the drawing models for the active drawing.

Note:

- The name of a drawing model in the **DRAW MODELS** menu is the same as its part or assembly name without the extension `.prt` or `.asm`. For example, if `MY_PART.prt` is a drawing model, it appears as **MY_PART** in the **DRAW MODELS** menu.
- Each drawing model has only one entry in the **DRAW MODELS** menu, regardless of whether it has simplified representations, and regardless of whether a simplified representation has been used in the drawing or has been set as the current representation.

4. Select the required drawing model from the **DRAW MODELS** menu to delete it from the active drawing. The drawing model and all its simplified representations are deleted from the active drawing.

Note: A model cannot be removed from the active drawing if the drawing has views that use the model or any of its simplified representations.

To Replace Models from Family Tables

If a model that you use in a drawing is a member of a family table, you can replace that model with other family table members. The system maintains dimensions, attached notes, and other drawing annotations when one model replaces another.

Use the **Select Instance** dialog box to select an instance with which to replace the model.

1. Click **File** > **Properties**. The Menu Manager opens.
2. In the Menu Manager click **FILE PROPERTIES** > **Drawing Models** > **Replace**.
3. Select the model that you want to replace as the active model. The **Select Instance** dialog box opens when you are replacing the model of one family table instance with another.
4. Select the name of another family member to replace it.
5. The **Select Instance** dialog box displays a list of all instances of the currently selected family table model. You can select one of these to use as the replacement model. You can select an instance by name when the **By Name** page is visible (the default), or click the **By Parameter** tab to select an instance by parameter.
6. After selecting an instance to be the replacement, click **Open**. The current model is replaced by the selected instance.

   Note: When you are using automatic replacement, if the system shows dimensions in the assembly drawing on a component that you have replaced with the new instance, it displays equivalent dimensions. However, if you replace the component using manual replacement, it does not preserve the dimensions.

If a model that you use in a drawing is a member of a family of parts or assemblies, you can replace that model with other family members. The system maintains dimensions, attached notes, and other drawing annotations when one model replaces another.

To Open a Model from a Drawing

To open a model from within Drawing mode, you can use **File** > **Open**, or you can use the Model Tree.

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Navigate to and select the model you want to open, and then click **Open**.

The model is retrieved into the current session in a separate Pro/ENGINEER window.

Using the Model Tree

1. In the Model Tree, select the model you want to retrieve, and then right-click to display the shortcut menu.
2. Click **Open** from the shortcut menu. The model is retrieved into the current session.

Sheets and Multiple Windows

You can:

- show multiple sheets in multiple windows
- show the same sheet in multiple windows
- create a new sheet in a new window

To create a new sheet in a new window:

1. Click **Window** > **New**.
2. At the prompt, type the sheet number to appear in the new window, and then press ENTER. You can also add a new sheet to the drawing. The new window appears with the specified sheet.

Assembly Drawings

About Assembly Drawings

In an assembly drawing, you can show and erase assembly and part dimensions. You can also create dimensions on individual components directly in an assembly view. However, keep in mind the following:

- You can display parameters for assembly features and all assembly components in assembly drawing notes, but you *cannot* type part parameters.
- The dimensions in an assembly drawing are visible for modification *only* if the assembly for which the drawing was created is in session.

Resuming Cosmetics

When you suppress parts or subassemblies in an assembly drawing, the system retains information about certain cosmetics of these views. When you resume them, you also resume these cosmetics:

- Dimension placement
- Surface finish symbol placement
- Parametric text in notes and symbol instances
- Cross-hatching cosmetics
- References to cross-sectional edges
- Detail axis cosmetics
- Member, edge, and set datum display information
- Draft geometric tolerances that reference part datums

About the Drawing View of a Manufacturing Process in an Assembly

If you have set the `auto_show_3d_detail_items` drawing setup file option to `yes` and create a drawing view of a manufacturing process, then the outer boundary of the tool path and the annotation elements attached to the step are automatically displayed within the drawing view, based on the placement of their references. Dimensions are displayed automatically for each step to represent the manufacturing area of the step.

The dimensions that are automatically displayed are for:

- Manufacturing location
- Manufacturing size

The dimensions are automatically created from the operation CSYS. The operation CSYS is the origin for the dimension. All outer boundary curve dimensions are displayed on the same plane as the curve, except depth dimension, which is displayed in the side view.

The outer boundary curve is displayed as an external contour of an envelope (including the tool diameter) of the tool path. The plane of the curve is defined as:

- Perpendicular to tool axis
- Passing through Z mini (bottom slice) of the tool path.

For holemaking steps, a curve circle is created in the start surface plane with the center on the hole axis. The diameter of circle depends on the following sequence types:

- For countersink sequence it is countersink diameter
- For other holemaking sequences it is tool diameter.

The dimensions show the distance between the origin of the Operation CSYS and the lowest tool path point. Four dimensions show the distance between the origin of the Operation CSYS and the maximum and minimum X and Y points of the curve outer boundary.

Note: The automatic display of dimensions is applicable only to NC Sequences such as face milling, hole making, and 2-axis trajectory milling.

You can create a drawing view of a manufacturing process in an assembly. Within the drawing view of the manufacturing process you can display information from the process manager step table into a Pro/REPORT table using the following manufacturing process parameters:

- Manufacturing criteria name
- User defined parameters from the step table
- Minimum and maximum X, Y, and Z values from the step table
- Machining time
- Step and operation CSYS name
- Status of the step
- Number of axes
- Template name
- Same behavior set name
- Parameters extracted from annotation elements

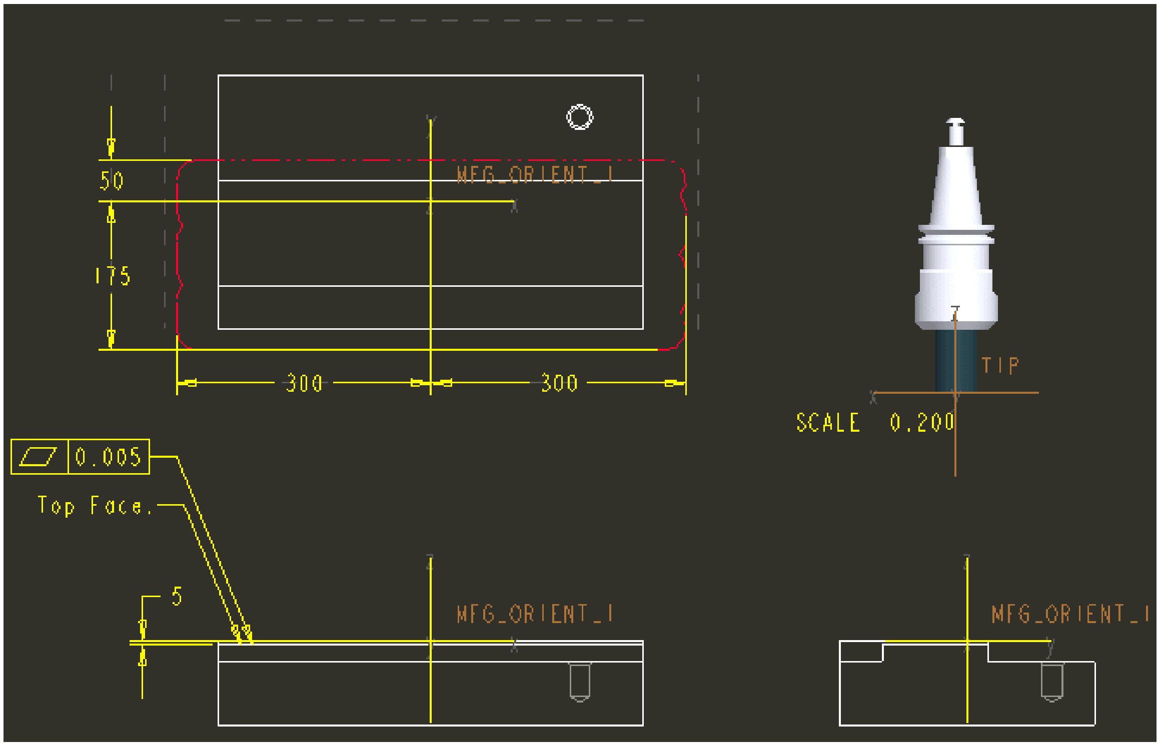
You can also create a drawing view for the current step tool used in the manufacturing process by clicking **Insert** > **Drawing View** > **Tool**. This creates a view that includes a representation of the tool. You can create a view of a solid tool only.

If you copy the drawing sheet that includes this view using **Edit** > **Move or Copy Sheet** > **Copy Process Sheet**, then the view is created in a new sheet and the tool related to the active step in the sheet is displayed. If the step in the drawing does not use a tool, then the view is left blank.

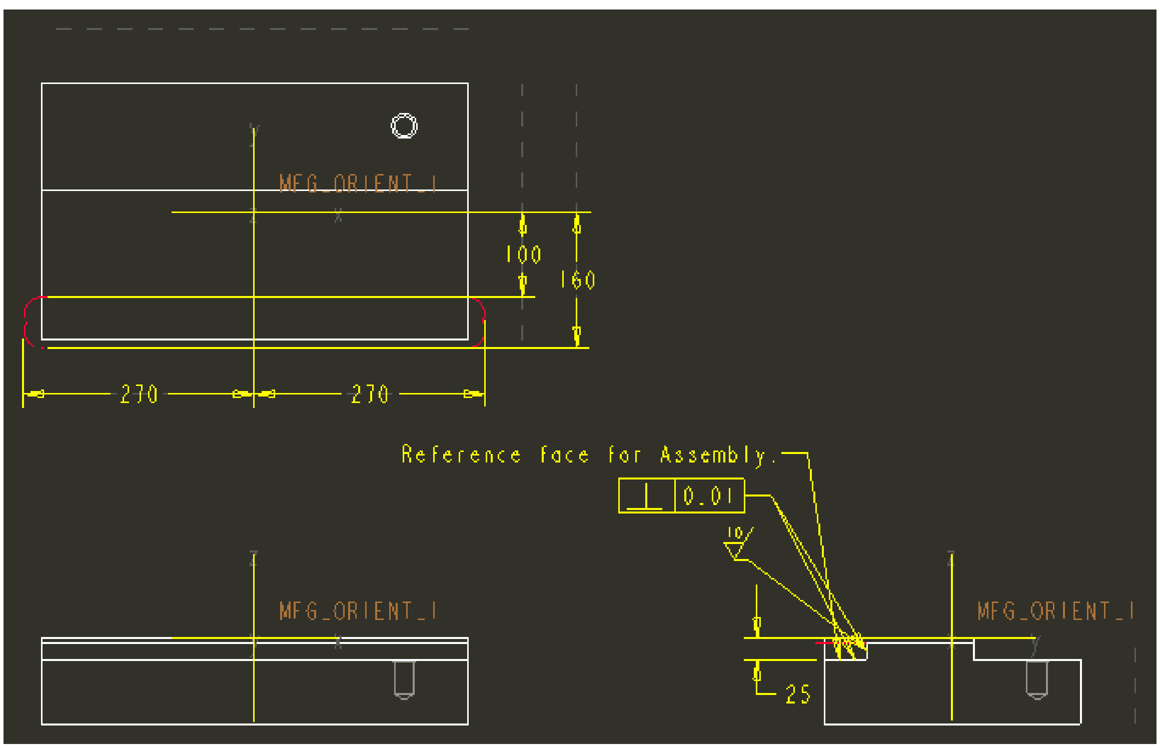
Note:

- You can show an annotation element only once in a drawing.
- You can change the color of the outer boundary curve.
- You can create a process sheet only for applied steps.

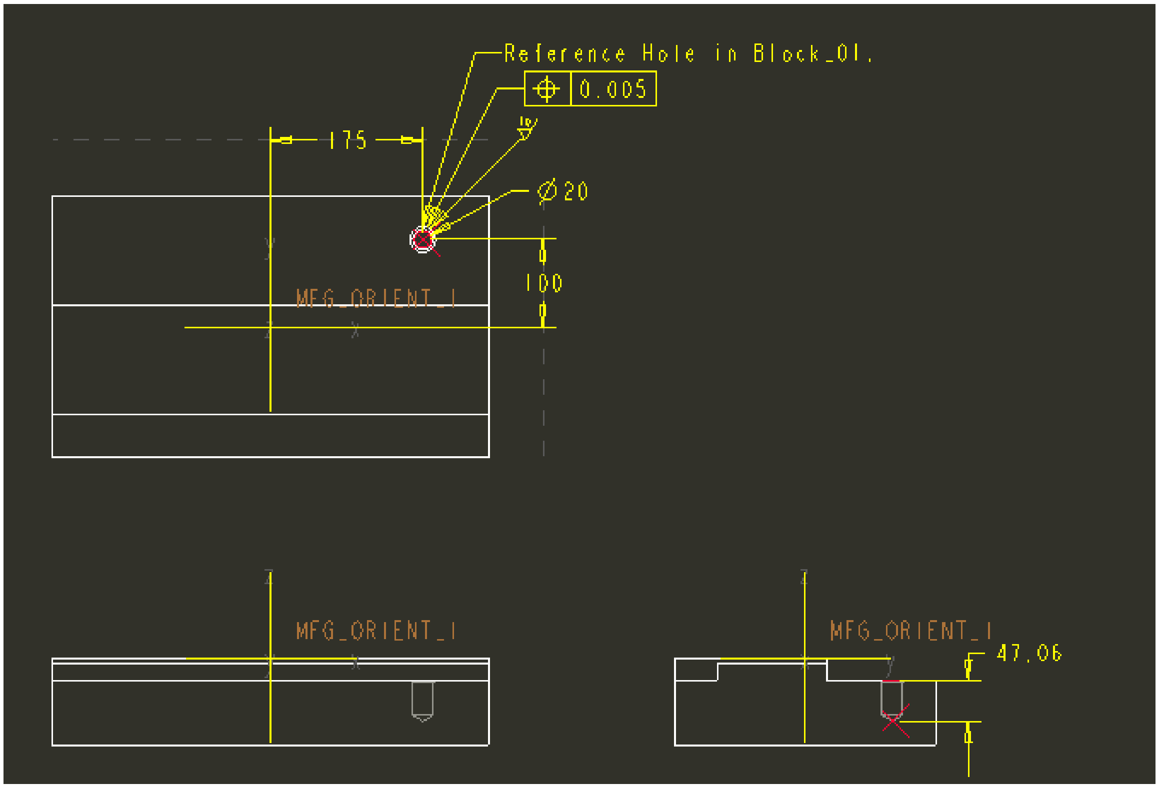
Example: Process Sheet for Face Milling Step



Example: Process Sheet for 2-Axis Trajectory Milling Step



Example: Process Sheet for Hole Making Step



Multisheet Drawings

About Multisheet Drawings

You can create multiple sheet drawings and move items from one sheet to another.

When working with multisheet drawings, keep in mind the following:

- You can switch a projection view to another sheet, but it will lose its association with its parent view. If you switch the projection view back to the same sheet as the parent, the association is restored.
- You can change drawing scales on each sheet independently.

To Add or Delete Sheets

To add a sheet, click **Insert** > **Sheets**. Pro/ENGINEER adds a new sheet to the end of the drawing.

To delete a sheet, Click **Edit** > **Remove** > **Sheet**.

Enter the sheet numbers of the sheets you want to remove. You can separate them by using commas or dashes and insert spaces anywhere, for example, [1,3-5,12,15-20]. The number to the left of a dash must always be less than the number to the right.

Press **Enter**.

To Reorder the Sheets in a Drawing

1. Click **Edit** > **Move Sheet**.
2. Use the dialog box to insert the current sheet within the sheet numbers.

   Note: If there is a process assembly in the drawing, the command is **Edit** > **Move or Copy Sheet**. There is an additional checkbox called **Copy process sheet**. When checked, the **Process State** dialog box opens and prompts you to select a step. Select **OK** to close the dialog box and insert the copied process sheet at the location specified in the list.

To Move Items to Another Sheet

1. Select the item or items to move.
2. Click **Edit** > **Move Item to Sheet** . You are prompted to enter the target sheet number.
3. Enter the target sheet number on the prompt line. Press Enter. The items are moved to the new sheet.

Maintaining View Size with Sheet Size

When moving items from one sheet to another in a drawing, use the configuration file option `variant_drawing_item_sizes` to control whether drawing items maintain their size and position on paper if you change the sheet size or drawing units.

- If you set this option to `yes`, some items scale and/or reposition to be the same size or in the same place when you plot them on paper, while other items scale and/or position themselves to be the same size or in the same place on screen.
- If you set it to `no`, items that you move or copy to a different sheet of the same or different 2-D model (or items on a sheet that changes size and/or units) retain the same size on paper and the same relative orientation as they did previously.

Controlling the Connection to the Model

About Drawing Changes and Model Revision

As you detail your drawing, some drawing entities and information are parametrically associated to the model that it documents. Therefore, changes made within the drawing could result in design changes within the model.

For example, if you set a dimension as basic within a drawing, the dimension becomes theoretically exact. Any dimensional or geometric tolerances are deleted from the drawing as well as from the 3D model.

The design changes in the model result from changes to drawing entities that are saved within the model; including all dimensions, tolerances, cross-sections, and model datums. Saving the model file with a version increase prevents loss of model data from changes made in the drawing file. After creating or modifying model geometry, such as shown model dimensions and their tolerances, or modifying the positioning of datums, the revision number of the model is incremented. Any other changes to drawing entities that are saved within the model are considered cosmetic and do not lead to a new model iteration. You can designate how to create drawing dimensions and how the model is affected using the `create_drawing_dims` and `drawing_model_read_only` configuration options.

The following drawing entities are considered cosmetic information and are saved within the drawing:

- All draft-only (non-associative) entities
- The view in which an entity appears
- The placement of an entity on the sheet
- Jogs and breaks in leaders and dimension lines
- The insertion of dimensions in notes
- The font, height, width, and slant angle of text

The following drawing entities are saved within the model and result in a new model iteration:

- Geometry changes, including modifications to shown model dimensions and their tolerances (dimension values, tolerance values, number of decimal places, and tolerance format).
- Datum position changes, including modification to the model dimensions that control the placement to model datums.

To Refresh the Connection to the 3D Model

Most drawing objects are updated dynamically as changes are made in the 3D associated model file. For a variety of reasons some objects may not refresh until you force a regeneration between the model and the drawing.

If you see objects in the drawing that should be updating but are not, use **View** > **Update** and one of the options below to refresh the model-drawing association:

- **Drawing View—**Updates a particular view with any recent model changes.
- **Current Sheet—**Updates all the views on the current sheet with any recent model changes.
- **All Sheets—**Updates all the views on all the sheets contained in the drawing, displaying any recent model changes.

Regenerating the Model from the Drawing

If graphics are not updating in the drawing, you may need to force a regeneration of the model. You can do this from the drawing, without opening the model file. There are two levels available for this process.

- **Edit** > **Regenerate** > **Draft**. Updates all values from the 3D model, but does not physically regenerate the 3D model, and does not refresh the geometry in the drawings. This can help performance in updating symbols or GTOLS without a complete refresh of the drawing. Drawing dimensions that have changed in the model but that have not been regenerated are displayed in a different color.

- **Edit** > **Regenerate** > **Model**. Completely refreshes the model-drawing association. Pro/ENGINEER first regenerates the model, then refreshes the geometry in the drawing. When you use this command, the **PRT TO REGN** menu opens on the Menu Manager. You have options concerning the extent of the regeneration you want to do. Use one of the options depending on the size of the file you are working with or the speed of your PCs performance:

  Select - Select a part or parts to regenerate. You can select multiple parts from the model tree, and click Done in the Menu Manager when you are finished.

  Automatic - Update all parts that have been edited in the 3D model since the last update.

To Open a 3D File from the Model Tree

You can quickly open any associated 3D file directly from the model tree.

1. Select the model in the model tree.
2. Right-click and click **Open** from the shortcut menu. The 3D Model is opened in a new window.

To Show a Referenced Part in the Model Tree

The drawing model tree displays one part and its features at one time. If more that one part is added to the drawing, use this procedure to change the part displayed in the model tree.

1. In the navigation window, click **Show** > **Use Model in Tree**. A selection prompt window opens.
2. Select a view representing the part you want to show in the model tree. Click OK in the selection window. The new part is displayed in the model tree.

To Turn Off Automatic Regeneration

By default, the display is updated with a repaint when you change from one window to another. Because model-to-drawing repaints and regenerations can be time consuming for larger files, you have the option to turn off the automatic regeneration function.

If you set `auto_regen_views` to `no`, you can selectively update the drawing by choosing **View** > **Update** from the menu. With this setting, you can only update the drawing to reflect changes in the 3D model, you cannot regenerate the model from the drawing as you can with the `yes` setting.

To Save Associated Models on Edit Only

By default, when you save a drawing, Pro/ENGINEER saves all associated models regardless of whether they were edited. This will update the file date, and may cause unwanted results in a file version-control situation.

If you want to save the associated model files *only when they are affected* by drawing changes, set the configuration file option `save_modified_draw_models_only` to `yes`.

To Disallow Changes Affecting the Model

To disallow changes made in Drawing mode that affect associated 3D models, set the configuration file option `draw_models_read_only` to `yes`. In this case, when you attempt to make a change that affects the model, Pro/ENGINEER issues a warning and does not make the modification.

Working with Model Views

About Working with Drawing Views

Once a model has been added to the drawing, you can place views of the model on a sheet. When a view is displayed on the drawing, you can determine how much of the model to show in a view, whether the view is of a single surface or shows cross sections, and how the view is scaled. You can then show the associative dimensions passed from the 3D model, or add reference dimensions as necessary.

Working with model views involves:

- Specifying the view types, such as general, projection, and detailed.
- Determining a location for the view on the drawing.
- Defining the drawing views; including determining visible areas, scaling, defining cross-sections, showing the model in various states, and aligning and setting the view origins.
- Moving and deleting views.

Inserting New Views

Inserting Drawing Views

Before you can place a view of a 3D model, you must associate the 3D model file with the drawing. This is called adding the model to the drawing.

When you start a new drawing file you are prompted in the **New File** dialog box for a 3D model file to reference. This will be the first model associated with the drawing. Once the drawing is in progress, you can add as many more models as you need to place views for.

If you have several models added to the drawing, you can only work on (place views and show dimensions for, etc.) one model at a time. This model is called the current active model.

Generally, placing a new view is a process of specifying the view type, the properties the particular type can have, and then choosing a location on the sheet for the view. Finally, the view is placed and you set the desired orientation for it.

Note: To move a view after it is placed, be sure the **Lock View Movement** check box is un-checked in the **Tools** > **Environment** dialog box. Or right-click on a blank section of the drawing and unlock the view using the shortcut menu.

To Insert a General View

A general view is usually the first view placed on a sheet. It is the most versatile view in that it can be scaled or rotated to any setting.

1. Click **Insert** > **Drawing View** > **General**. If user-defined presentation states exist in the model that the drawing is referring to, the **Select Presentation** dialog box opens.
2. Under **Presentation names**, a list of the user-defined combination states is displayed. Select a state or select **No Presentation**.

   Note: When you select a presentation state, the **Model View Names**, **Simplified representation**, **Presentation State**, **Assembly explode state**, and **2D cross-section** or **3D cross-section** boxes in the **Drawing View** dialog box are automatically updated based on what is stored in the selected presentation.
3. If you do not want to be prompted to select a combination state, click the **Do not prompt for Presentation** check box.
4. Click at a location where you want to place the general view. The general view appears with the orientation specified by the selected combination state and the **Drawing View** dialog box opens. By default, the **View Type** category is selected and the options for defining the view type and orientation are displayed.
5. If required, you can modify the view name in the **View name** box and change the view type from the available options in the **Type** list.
6. To change the current orientation, under **View orientation**, select from the following orientation methods:
   - **View names from the model**—Orient using saved views from the model.

     Select the appropriate model view from the **Model view names** list.

     Note: While creating the view, if you have selected a combination state, then the named orientation in the selected combination is retained in the **Model view names** list. If this named view is changed, then the combination state is no longer listed.

Define the x and y orientation by selecting the desired Default orientation. You can select either **Isometric**, **Trimetric**, or **User Defined**. For **User Defined** you must specify the custom angle values.

- **Geometry references**—Orient using geometry references from the previewed model in the drawing.

  Select the direction to orient the reference from the list next to the reference being defined. The list provides several options, including **Front**, **Back**, **Top**, and **Bottom**.

  Select the desired reference on the model previewed on the drawing. The model repositions according to the direction defined and the references selected.

You can change the orientation by selecting another direction from the direction list. You can change the selected reference by clicking the reference collector and selecting a new reference on the drawing model.

Note: To return the view to its original orientation, click **Default Orientation**.

- **Angles**—Orient using angles of selected references or custom angles.

  The **Reference Angle** table lists the references used to orient the view. By default, a new reference is added to the list and highlighted.

  Select the desired option from the **Rotation reference** box for the highlighted reference in the table:

  Normal—Rotate the model around an axis through the view origin and normal to the drawing sheet.

  Vertical—Rotate the model around an axis through the view origin and vertical to the drawing sheet.

  Horizontal—Rotate the model around an axis through the view origin and horizontal to the drawing sheet.

  Edge/Axis—Rotate the model around an axis through the view origin and according to the designated angle to the drawing sheet. Select an appropriate edge or axis reference on the previewed drawing view. The selected reference is highlighted and is listed in the **Reference Angle** table.

- Type an angle value for the reference in the **Angle Value** box.

Note: To create additional references, click and repeat the angle orientation process.

7. To continue defining other attributes of the drawing view, click **Apply** and then select the appropriate category. If you have completely defined the drawing view, click **OK**.

Note: If you delete or suppress geometry that Pro/ENGINEER is using to orient a view, Pro/ENGINEER changes that view and its children to the default orientation. You cannot recover the original view orientation if you delete the geometry.

Resuming a suppressed feature, however, restores the original orientation of the view.

To Insert a Projection View

A projection view is an orthographic projection of another view's geometry along a horizontal or vertical direction. Projection views are placed in projection channels, above, below or to the right or left of the parent view.

1. Click **Insert** > **Drawing View** > **Projection**.
2. Select the parent view that you wish to display in the projection. A box appears over the parent view, representing the projection.
3. Drag the box horizontally or vertically to the desired location. Left-click to place the view. To modify the properties of the projection, select and right-click the projection view. Click **Properties** on the shortcut menu.
4. To continue defining other attributes of the drawing view, click **Apply** and then select the appropriate category. If you have completely defined the drawing view, click **Ok**.

Note:

- You can also create a projection view by selecting and right-clicking the parent view. Click **Insert Projection View** from the shortcut menu.
- When a projection view is created it is given a default name based on the direction from which it is derived.
- To show the view name as a label on the view, make sure the `make_proj_view_notes` configuration option to `yes`. You must set the option before you create the view.
- Use the `default_view_label_placement` drawing setup file option to specify the location of the view label.

To Insert a Detailed View

A detailed view is a small portion of a model shown enlarged in another view. A reference note and border are included in the parent view as part of the detailed view setup. Once a detailed view is placed on the drawing sheet, you can modify the view, including its spline border, using the **Drawing View** dialog box.

1. Click **Insert** > **Drawing View** > **Detailed**. The **Select** dialog box opens.
2. Select a point in an existing drawing view that you want to enlarge in a detailed view. The drawing item is highlighted and you are prompted to sketch a spline around the point.

   Sketch a spline encompassing the area that you want to show in detail.

   Note: Do not initiate the spline sketch using the sketcher toolbar. Click in the drawing area to begin the sketching of the spline. If you access the sketcher toolbar to sketch the spline, the creation of the detailed view is aborted.

Do not worry about sketching a perfect shape because the spline is automatically corrected. Middle-click when the sketch is complete. The spline is displayed as a circle and a note with the detailed view name is created.

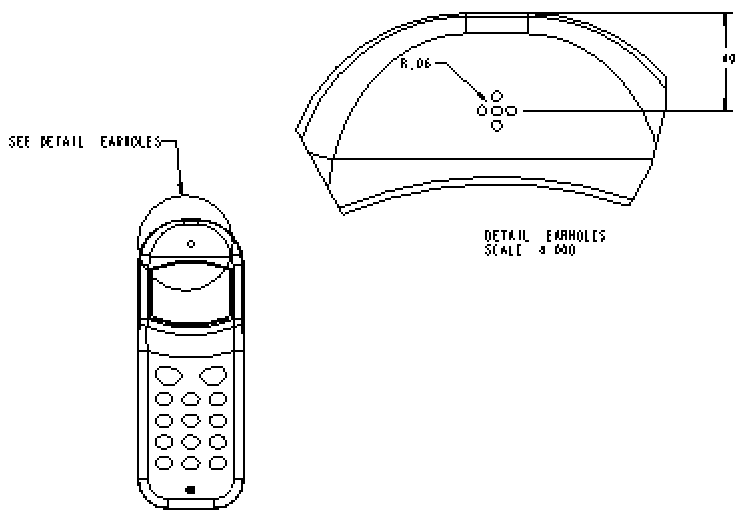
Note: You can define the shape of the sketch in the **View Type** category of the **Drawing View** dialog box. Select the required option from the **Boundary type on parent view** box:

- **Circle**—Draws a circle in the parent view for the detailed view.
- **Ellipse**—Draws an ellipse in the parent view for the detailed view to closely match the spline, and prompts you to select an attachment point on the ellipse for the view note.
- **H/V Ellipse**—Draws an ellipse with a horizontal or vertical major axis and prompts you to select an attachment point on the ellipse for the view note.
- **Spline**—Displays the actual spline boundary on the parent view for the detailed view, and prompts you to select an attachment point on the spline for the view note.
- **ASME 94 Circ** —Displays an ASME standard compliant circle in the parent view as an arc with arrows and the detailed view name.

3. Select where you want to place the detailed view in the drawing. The area of the parent view within the spline is displayed and labeled with the name and scale of the detailed view.
4. To continue defining other attributes of the drawing view, click **Apply** and select the appropriate category. If you have completely defined the drawing view, click **OK**.

Note:

- Use the `detail_circle_note_text` drawing setup option to set a default value for the note text. For example, if the value of the option is `See View <viewname>`, the note text reads as SEE DETAIL EARHOLES.



- Use the `default_view_label_placement` drawing setup file option to specify the location of the view label.

To Insert an Auxiliary View

An auxiliary view is a type of projection view that projects at right angles to a selected surface or axis. The direction of the selected surface determines the projection channel. The reference in the parent view must be perpendicular to the plane of the screen.

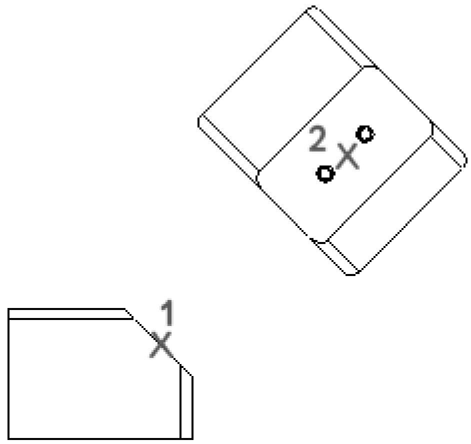
1. Click **Insert** > **Drawing View** > **Auxiliary**. The **Select** dialog box opens.
2. Select an edge, axis, datum plane, or surface to create the auxiliary view from. A box appears over the parent view, representing the auxiliary view.
3. Drag the box horizontally or vertically to the desired location. Left-click to place the view. The auxiliary view is displayed.

Note:

- To modify the properties of the auxiliary view, access the **Drawing View** dialog box by double-clicking the projection view or right-clicking the view and clicking **Properties** on the shortcut menu.

You can define other attributes of the drawing view by using the categories on the **Drawing View** dialog box. When you are done defining each category, click **Apply** and select the next appropriate category. When you have completely defined the drawing view, click **OK**.

Example



1. Select the plane to project
2. Select the center point for the new view

To Insert a Revolved View

A revolved view is a cross section of an existing view, revolved 90 degrees around a cutting plane projection. You can use a cross section created in the 3D model as the cutting plane, or you can create one on the fly while placing the view. The revolved view differs from a cross section view in that it includes a line noting the axis of revolution for the view.

1. Click **Insert** > **Drawing View** > **Revolved**. You are prompted to select the view you wish to cross-section.
2. Select the view to cross section. The view highlights.
3. Select a location on the drawing to display the revolved view, approximately along the cutting plane projection in the parent view. The View Dialog box opens. You can modify the view name, however, you can not change the view type.
4. Define where to revolve the view by selecting an existing cross-section from the **Cross-section** list or creating a new one. To create a new cross-section on the fly:

- Select **Create New** from the **Cross-section** list. The XSEC CREATE menu appears in the menu manager.
- Define the cross-section as **Planar** and select the appropriate attributes. Click **Done**. You are prompted to name the cross-section.
- Type the desired name and press `ENTER`. You are prompted to define the cross-section reference.
- Select and existing reference or create a new one. Only cross-sections parallel to the screen will be created.

If you selected or created a valid cross-section the revolved view is displayed on the drawing.

5. To continue defining other attributes of the drawing view, click Apply and then select the appropriate category. If you have completely defined the drawing view, click OK.

Note: Modify the centerline of symmetry, as necessary:

- Replace the butting plane extension as the centerline of symmetry by selecting a datum plane or axis in the revolved view that is parallel to the cutting plane projection in the parent view. The system centers the revolved view so that the selected plane (or axis) is co-linear with the cutting plane in the parent view. If you press the middle mouse button instead, the cutting plane passes through the center of the revolved view.
- Clip the centerline by choosing **Edit** > **Move** on the menu bar. Select an endpoint of the centerline and the new location; then press the middle mouse button.
- If you select a real axis (one that is normal to the screen) to center the view, display the axis by clicking **Axis** in the **Type** box of the **Show/Erase** dialog box (click **View** > **Show and Erase** on the menu bar).

To Insert a Copy and Align View

If you already have a partial or detailed view in the drawing you can create another aligned partial view as a copy of the original, but with a different boundary defined. A copy and align view is similar to a broken view, except a copy and align view enables you to sketch several partial views to define the portions of the whole view you want to show. Broken views remove geometry along a fixed horizontal or vertical line.

The edges of the portions are aligned to their "parent" edges in the original view.

Copy and align views are only available if a partial view, such as a detail view, is already displayed in the drawing.

1. Click **Insert** > **Drawing Views** > **Copy and Align**. You are prompted to select an existing partial view.
2. Select the partial view you wish copy and display in an alternative view. The selected view is highlighted.
3. Select where you want to place the detailed view on the drawing. The copied view is displayed on the drawing. The view temporarily appears in full. You are prompted select a center point for the area you wish to display.
4. Select a reference point on the new view to define the center of the partial portion. You are prompted to sketch a spline to define the outline.
5. Sketch a spline encompassing both the reference point and the area you want to show.

Note: Do not initiate the spline sketch using the sketcher toolbar. Click in the drawing area to begin the sketching of the spline. If you access the sketcher toolbar to sketch the spline, the creation of the 'copy and align' view is aborted.

On the newly created partial view, select a straight entity: edge, axis, datum curve, or cable segment. The current partial view aligns with the existing partial view along the selected entity.

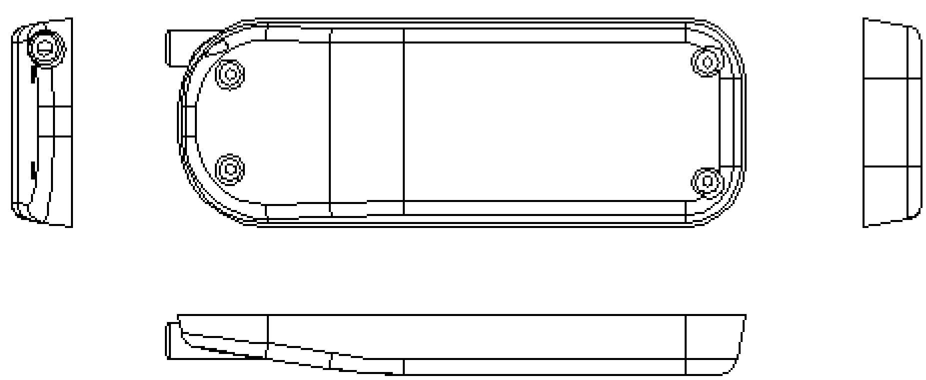
Note:

- To modify the properties of the copy and align view, access the Drawing View dialog box by double clicking the view or select and right-clicking the view and clicking **Properties** on the shortcut menu.
- You can define other attributes of the drawing view using the categories on the Drawing View dialog box. When you are done defining each category, click **Apply** and then select the next appropriate category. When you have completely defined the drawing view, click **Ok**.
- Use the `default_view_label_placement` drawing setup file option to specify the location of the view label.

Example: Basic View Types

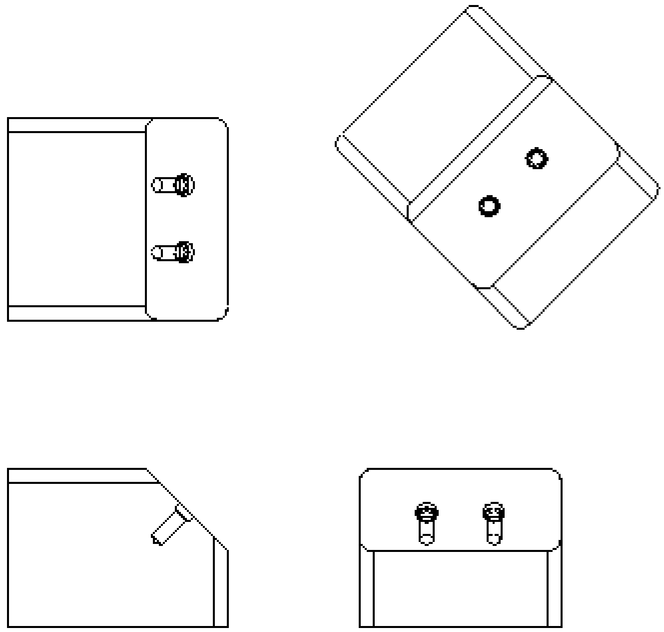
This topic describes the basic view types used by Pro/Engineer; general, projection, auxiliary, and detailed. Refer to the See Also topics at the bottom of the page for more specialized view types.

- A ***general*** view is usually the first view of a series to be placed, for example, it may serve as the parent for projection views or other views derived from it.
- A ***projection*** view is an orthographic projection of another view's geometry along a horizontal or vertical direction. Projection views are placed in *projection channels*, above, below or to the right or left of the parent view.



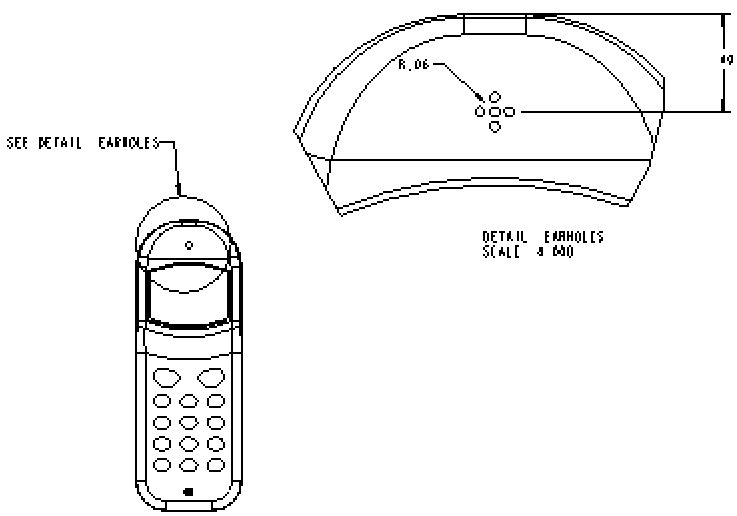
Above: General view (center) with projection views

- An ***auxiliary*** view is a type of projection view that projects at right angles to a selected surface or axis. The selected surface in the parent view must be perpendicular to the plane of the screen.



Above: An auxiliary view (upper right).

- A ***detailed*** view is a small portion of a model shown enlarged in another view. A reference note and border is included on the parent view as part of the detailed view setup.



Above: Detailed View

To Insert a Graph in a Drawing

1. Click **Insert** > **Graph**. If a user-defined graph exists in the model that the drawing is referring to, the **Graphs** dialog box opens.

2. Under **Select a datum graph**, select the required graph in the list of user-defined graphs and click **OK**.

   Note: You can change the scale of the graph from the **Scale** box.

3. Click at a location in the drawing where you want to place the graph. The graph appears in the drawing.

Defining Drawing Views

Determining Visible Area of Views

Determining Visible Area of the View

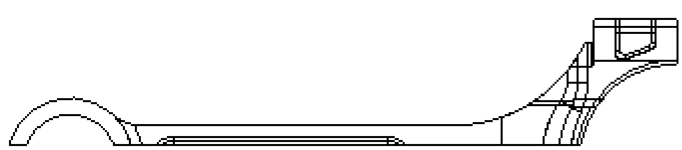
As you detail your model, certain portions of the model may be more relevant than others or may benefit from being displayed from a different view point. You can define the visible area of the view to determine which portion or portions to show or hide.

You can define the visible area of the view from the **Drawing View** dialog box which can be accessed:

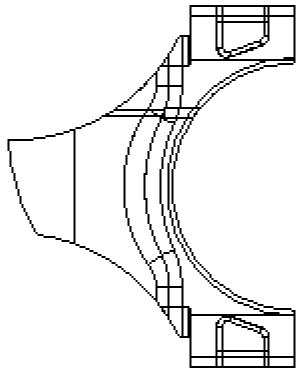
- When you insert a new view.
- Double click an existing view.
- Select and right-click an existing view, and the click **Properties** on the shortcut

You can define the visible area of the view in the following ways:

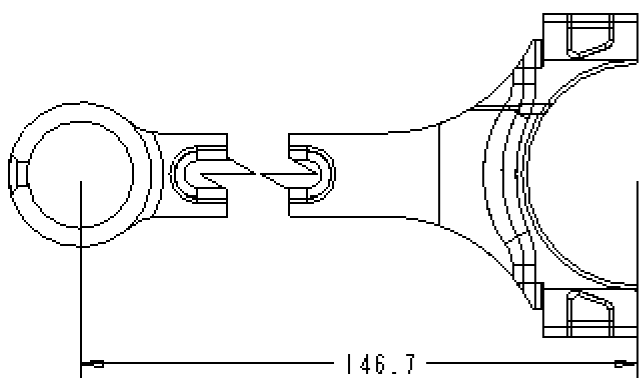
- **Half View**—Removes a portion of the model from the view on one side of a cutting plane.



- **Partial View**—Displays a portion of the model in a view within a closed boundary. The system displays the geometry appearing within the boundary, but removes the geometry outside of it.



- **Broken View**—Removes a portion of the model from between two or more selected points, and closes the remaining two portions together within a specified distance. You can break horizontally, vertically, or both, and use various graphic border styles for the breaks.



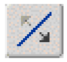
You can specify a plane parallel to the screen and exclude all graphics behind it. All geometry behind the defined Z-Clipping does not appear, but any geometry that the plane contains entirely does appear. The system clips geometry that intersects the plane. Only the portion in front of the plane appears.

To Insert a Half View

Half views cut the model at a plane, erasing one portion of it, and displaying the rest.

1. With the **Drawing View** dialog box open, click the **Visible Area** category. The **Visible area options** display in the dialog box.
2. Select **Half View** from the **View Visibility** list. The options for defining the view area display.
3. Select the reference that will divide the view. The cutting plane may be a planar surface or a datum, but it must be perpendicular to the screen in the new view.

The selected reference highlights and is listed in the **Half view reference plane** collector.

4. Define which half of the model to display by clicking the single red arrow, which displays from the reference plane pointing toward the side that will be displayed. The arrow will be displayed just after a planar surface or datum is selected or after the  is clicked. The view itself will not change until you click Apply or Ok. The arrow is no longer displayed.

5. Using the **Symmetry line standard list**, define how to indicate the split in the half view:

    - **No line**
    - **Solid line**
    - **Symmetry line**
    - **Symmetry line ISO**
    - **Symmetry line ASME**

6. You can specify a plane parallel to the screen and exclude all graphics behind it by clicking **Clip view in z-direction** and select an edge, surface, or datum plane clipping reference that is parallel to the view. When you perform Z-Clipping in a view, keep in mind the following:

    - If the system cannot regenerate the reference for the clipping plane, Z-Clipping does not take effect for the view (an error message appears).
    - The Z-Clipping of a detailed view is always the same as that of its parent. You cannot modify it individually.

7. To continue defining other attributes of the drawing view, click **Apply** and then select the appropriate category. If you have completely defined the drawing view, click **Ok**.

Note: Each half view can display its own symmetry line differently. You can format the half view line using the `half_view_line` drawing setup option. When changing a view to half view, the default symmetry line standard to use in the view will be decided by the above drawing setup file option value.

To Insert a Partial View

Partial views display geometry that exists inside of a sketched boundary.

1. With the **Drawing View** dialog box open, click the **Visible Area** category. The **Visible area options** display in the dialog box.

2. Select **Partial View** from the **View Visibility** list. The options for defining the view area display.

3. Select the geometry of the view near the center of the area you want to retain in the partial view. The selected item highlights.

4. Sketch a spline encompassing the area you want to show. **Note**: Do not initiate the spline sketch using the sketcher toolbar. Simply click on the drawing to begin the sketch. If you do access the toolbar, the partial view is cancelled and the spline is a two-dimensional draft entity. Middle-click when you done sketching the spline.
5. To display the boundary for the partial view contained within the spline, make sure Show spline boundary on view is selected. The boundary displays in geometry line font.
6. You can specify a plane parallel to the screen and exclude all graphics behind it by clicking **Clip view in z-direction** and select an edge, surface, or datum plane clipping reference that is parallel to the view. When you perform Z-Clipping in a view, keep in mind the following:

- If the system cannot regenerate the reference for the clipping plane, Z-Clipping does not take effect for the view (an error message appears).
- The Z-Clipping of a detailed view is always the same as that of its parent. You cannot modify it individually.

7. To continue defining other attributes of the drawing view, click **Apply** and then select the appropriate category. If you have completely defined the drawing view, click **OK**.

To Insert a Broken View

Broken views remove a portion of the model from between two or more selected points, and closes the remaining two portions together within a specified distance. You can break horizontally, vertically, or both, and use various graphic border styles for the breaks.

Broken views are only available for general and projection view types. Once a view is defined as broken, it can not be changed to another view type.

1. With the **Drawing View** dialog box open, click the **Visible Area** category. The **Visible area options** display in the dialog box.
2. Select **Broken View** from the **View Visibility** list. The options for defining the view area display.
3. Click  to add a break to the view. A row appears in the broken view table. Two lines define one break. The area between the lines will be removed. You can place both directions in the same session, including horizontal and vertical lines.
4. Sketch a horizontal or vertical break line by selecting a geometry reference and then dragging the mouse in the desired direction. Select the geometry reference carefully, since the first break line begins at the point selected. The break line reference is listed under 1st Break Line in the broken view table.
5. Select a point to define the placement of the second break line. The distance between the sketched line and selected point determines how much model geometry is removed from the view. The break line reference is listed under 2nd Break Line in the broken view table.

6. Define how the graphical representation of the break line by selecting a style from the **Break Line Style** list on broken view table. You may need to scroll or resize the table columns to access the list:
    - **Straight**
    - **Sketch**
    - **S curve on view outline**
    - **S curve on geometry**
    - **Heartbeat on view outline**
    - **Heartbeat on geometry**
7. If necessary, you can define additional breaks by clicking  and repeating steps 4 through 6.
8. You can specify a plane parallel to the screen and exclude all graphics behind it by clicking **Clip view in z-direction** and selecting an edge, surface, or datum plane clipping reference that is parallel to the screen. When you perform Z-Clipping in a view, keep in mind the following:
    - If the system cannot regenerate the reference for the clipping plane, Z-Clipping does not take effect for the view (an error message appears).
    - The Z-Clipping of a detailed view is always the same as that of its parent. You cannot modify it individually.
9. To continue defining other attributes of the drawing view, click **Apply** and then select the appropriate category. If you have completely defined the drawing view, click **Ok**.

Note:

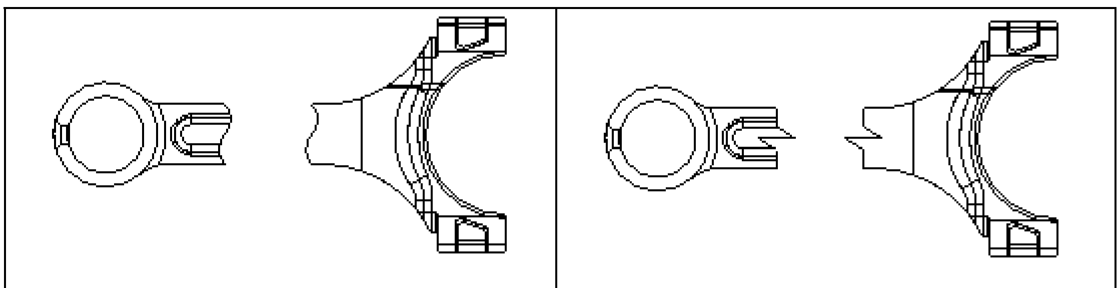
- You can control the offset distance when you first create a break by setting the `broken_view_offset` drawing setup file option. The default spacing is 1 drawing unit. To change the spacing, drag one of the subviews, or portions, of the broken view. The space between the sections will increase or decrease proportionally.
- To move the entire broken view to a different location on the drawing, select the upper-left sub-view.
- If the parent view is redefined to be a broken view, it will have no restrictions as to which direction the breaks can be made or the number of breaks, unless it has projected child views. If it at least one projected view in the horizontal direction and at least one projected view in the vertical direction, Broken View selection is not available in the View Visibility list.
- If the parent view is a full view and has at least one projected child view in the horizontal direction and no child view projected in the vertical direction, then the parent view will only have the ability to create break lines (number of breaks in unlimited) in the vertical direction.

- You can redefine a projected view to be broken if its parent view is not broken. However, it can broken according to the following limitations:
    - Horizontal projections can only use vertical breaks, while vertical projections can only use horizontal breaks.
    - If the parent view is broken horizontally and you want to redefine a horizontal projection view to broken, the projection view will automatically get the same number of horizontal breaks as its parent view. In fact, the projection parent view's horizontal breaks will be automatically inherited to the projection view.
    - If the projection parent view is broken but only has vertical breaks, the projection view can only have vertical breaks.

  Note: Similar restrictions are in place for top or bottom projection views if the projection parent has vertical breaks.

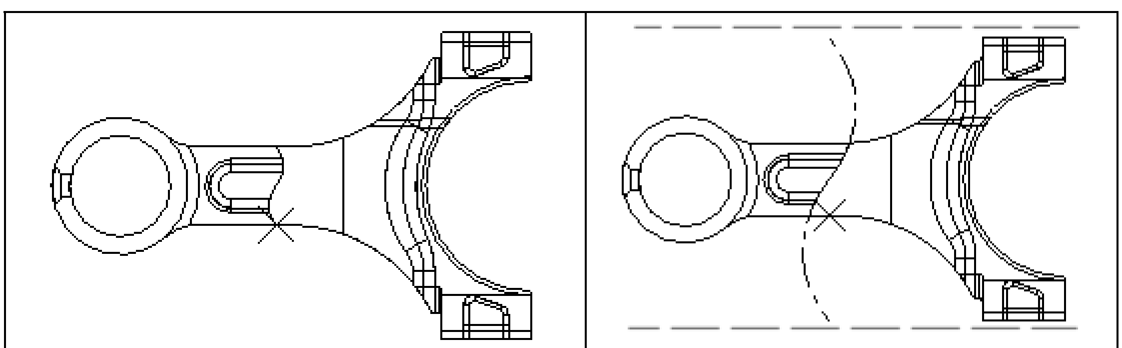
Example: Broken View Styles

You can draw the break line for a broken view in several ways. You can freehand sketch a spline for the break line, or use two standard shapes shown below.

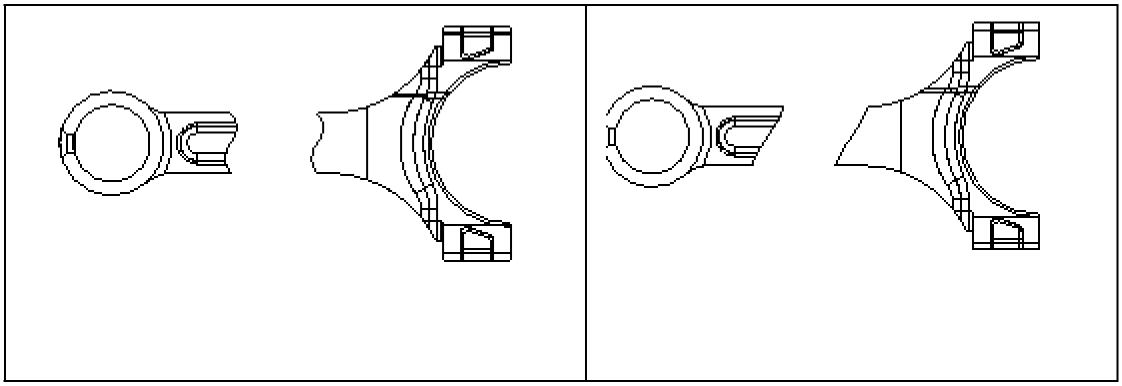


S-Curve | Heartbeat

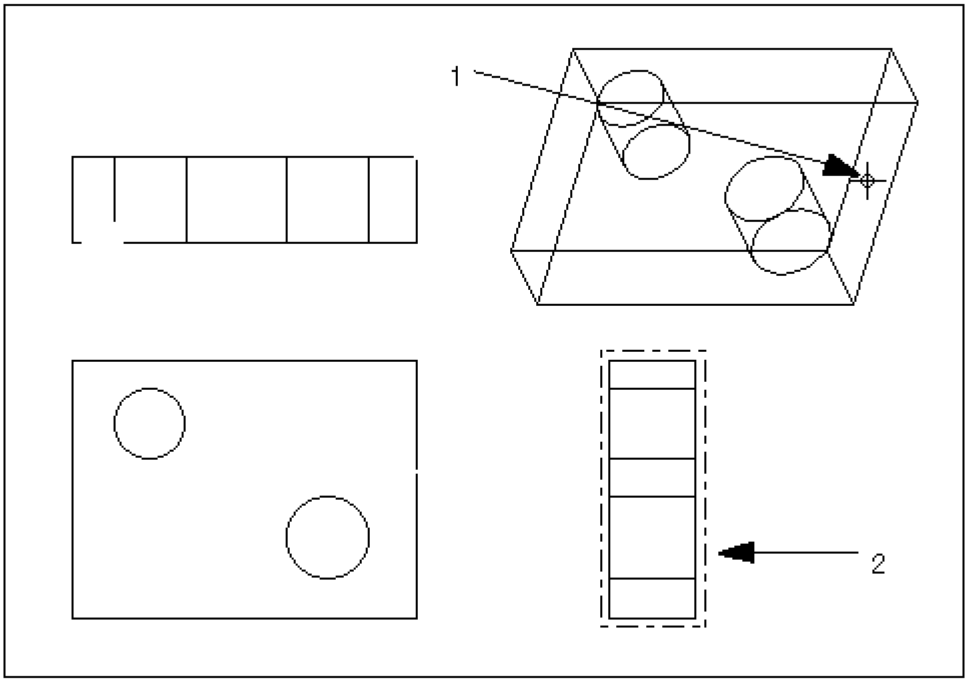
For each shape, you can choose to draw it to fit the extents of the geometry itself, or to fit the overall extents of the view. This is shown in the S-curve example below.

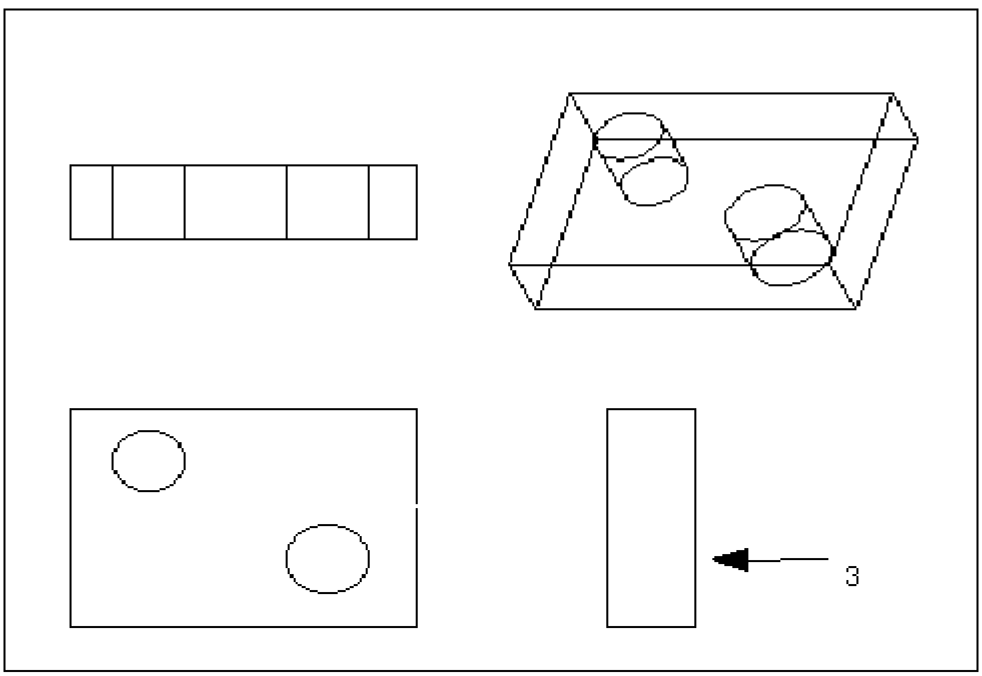


S-curve drawn to geometry | S-curve drawn to view extents



Example: Clipping view in z-direction





1. Select this surface as the reference point.
2. Select this view.
3. The geometry intersecting the plane is clipped.

Specifying the Scale of a View

Specifying the View Scale

Scaling your drawing and views enables you to show certain model elements in various sizes. For example, you may accent a portion of the model by depicting it at 2x its actual size. You can document the relationship between items on your drawings by specifying a scale, which informs readers of the drawing proportions.

You can use the following types of scales in your drawings:

- **Default scale for sheet**—Sizes the drawing views according to a default value. If you do not set a default value, Pro/ENGINEER determines a default scale for each sheet based on the sheet size and the model dimensions. The scale applies to all views that do not have a custom scale or perspective applied. The drawing sheet scale displays at the bottom of the drawing sheet.
- **Custom scale**—Sizes the view using a custom value typed in the **Drawing View** dialog box. When you modify the drawing sheet scale, custom views do not change, since the scale factor is independent. The custom scale appears below each view, shown in a note "Scale `<value>`."
- **Perspective**—Sizes the drawing using a combination of eye-point distance from model space and paper units, such as `mm`. This scale option is only available for general views.

A newly created view takes on the value of the drawing sheet scale, unless you specify a different scale value. You can change the scale of any existing view.

When you change the size of the drawing, the drawing sheet scale changes to keep the views in proportion to the size of the sheet. However, detailed and scaled views retain their original scale regardless of changes to the drawing size. You can change drawing scales on each sheet independently.

When you modify a drawing view scale, all related parent/child views (projected views, broken views, etc.) update accordingly.

You can use relational expressions to drive a custom or perspective view scale; however, you cannot use them to drive views using the drawing sheet scale. When you type a relational expression for the view scale, the system calculates the expression and retains the information. This functionality enables you to relate the view scale using dimensions within the model. The system stores the relation with the model and updates the view scale each time the dimension in the model changes, along with all items related to it.

Note:

- You can control the initial setting of the default scale by setting the `default_draw_scale` configuration file option; otherwise, the size is automatically calculated based on the size of the model. You can override your settings by modifying the scale.
- You can express drawing scales in a decimal or fractional format by setting the `view_scale_format` drawing setup file option (decimal format is the default).

- To establish the pattern of fractional scales, use the `view_scale_denominator` drawing setup file option.
- If you change the drawing scale, symbols with a fixed size will not change. To change the symbol size you should redefine the symbol attributes.
- To show or erase the view scale note, use the **Show/Erase** dialog box (click **View** > **Show and Erase)**.

To Specify the View Scale

You can specify the scale for individual views when you add the view to the drawing, or you can modify the scale at any time. You can set the individual view scales in the following ways:

- Using the drawing sheet scale
- Using a custom scale
- Using perspective

Using the Drawing Sheet Scale

By default, new views are set to display according to the drawing sheet scale. If you want to display the view at this scale simply place the view.

The drawing sheet scale displays at the bottom of the drawing sheet, or you can view the value in the Scale category on the Drawing View dialog box.

You can modify this value by:

- Setting the `default_draw_scale` configuration file option.
- Double-clicking on the drawing sheet scale at the lower-left corner of the screen and then typing the real value or relational expression.
- Selecting the drawing sheet scale at the lower-left corner of the screen and then right-clicking and clicking **Edit Value**.
- Selecting the drawing sheet scale at the lower-left corner of the screen and then clicking **Edit** > **Value**.

Using a Custom Scale

You can define a custom drawing view proportion for individual views.

1. With the **Drawing View** dialog box open, click the **Scale** category. The **Scale and perspective options** display in the dialog box.
2. Click **Custom Scale**. The scale box is enabled.
3. Type a real value or relational expression in the box. Click **Apply** to see the size of the view with the new scale. If scale is not acceptable, repeat the steps above by entering a different scale value. If the scale is appropriate and no changes needs to be made, select the next appropriate category to define or click **Close**.

Using Perspective

1. With the **Drawing View** dialog box open, click the **Scale** category. The **Scale and perspective options** display in the dialog box.
2. Click **Perspective**. The distance and diameter boxes are enabled. The unit of measurement displays next to each box.
3. Type the desired value for the **Eye-point distance**, which measures the distance in model space.
4. Type the desired value for the **View diameter**, which scales the view using paper units.

   Click **Apply** to see the size of the view with the new scale. If scale is not acceptable, repeat the steps above by entering a different scale value. If the scale is appropriate and no changes needs to be made, select the next appropriate category to define or click **Close**.

Note:

- When using a relational expression, type *only* the right-hand side of the expression. The system calculates the left-hand side as the view scale.
- To show or erase the view scale note, use the **Show/Erase** dialog box (click **View** > **Show and Erase**).

Displaying Sections of Views

To Create a 2D Cross Section in a Drawing

1. Create a drawing or open an existing drawing.
2. Double-click on a drawing view. The **Drawing View** dialog box opens.
3. Click **Sections**. The **Sections** category page is displayed in the **Drawing View** dialog box.
4. Click **2D cross-section**. The options for creating or displaying a 2D cross-section are displayed in the **Sections** category page. If a 2D cross-section does not exist in the drawing, you will need to create a new one.
5. Click  in the **Sections** page to create a new 2D cross-section.
6. Select **Create New** from the **Name** list in the **Sections** page. The **XSEC CREATE** menu appears.
7. Define the cross-section as either **Planar** or **Offset**. Select the appropriate attributes for **Planar** or **Offset** on the **XSEC CREATE** menu.
8. Click **Done**. You are prompted to type a name for the cross-section.
9. Type a name and press ENTER. The **SETUP PLANE** menu appears. You are prompted to define the cross-section reference.
10. Click **Plane** to select an existing reference or **Make Datum** to create a new one.

11. Middle-click when you have selected an existing plane or when you have created a new plane. The 2D cross-section is placed in the drawing and the **Drawing View** dialog box closes..

Note: You can only display existing 3D cross-sections in drawings. You cannot create a new 3D cross-section in a drawing.

Displaying Cross Sections in Drawings

A cross section is an imaginary cutting plane applied through a part, with a cross hatch pattern associated with it. How the cross section is shown in a particular view is a function of that view's view type- full, half, local, etc.

Before placing a cross section defined as planar in the view, you must orient the view so that the plane of the section is parallel to the plane of the screen. If the cross section is revolved, the plane of the section must be perpendicular to the screen.

You can create cross-sections in part and assembly mode, which can be used to display portions of the component within drawings. These three-dimensional cross-sections are similar to assembly cuts, but are only used for display purposes.

Note: If you want to redefine the side and direction of offset cross sections, you must do it in either Part or Assembly mode. You cannot redefine them in Drawing mode.

To Display a 2D Cross-Section View

1. With the **Drawing View** dialog box open, use the **View Type** category to define the view orientation, such that the view you want to cross-section is oriented properly to the screen.
2. Click the **Sections** category. The section options are displayed in the dialog box.
3. Click **2D cross-section.** The 2D cross-section properties table is enabled. If a 2D cross-section does not exist in the drawing, you must create a new one.
    - **Create 2D cross-section**—If a 2D cross-section does not exist in the drawing or if you want to display another cross-section, you can create one.

      Select **Create New** from the Name list, located in the 2D cross-section table. The **XSEC CREATE** menu appears on the Menu Manager.

      Define the cross-section as either **Planar** or **Offset** and select the appropriate attributes. Click **Done**. You are prompted to name the cross-section.

      Type the desired name and press ENTER. You are prompted to define the cross-section reference.

      Select an existing reference or create a new one. Only cross-sections parallel to the screen are created.
    - **Display existing 2D cross-section**—If a 2D cross-section already exists in the model, you can select it from the Name list in the 2D cross-section table. The existing cross-sections are listed in alphabetical order. Valid

cross-section are indicated by ✔; while ✘ indicates the cross-section is not parallel to the screen and cannot be placed.

Note: Crosshatches for newly created cross-sections are displayed based on the value that you have specified for the `default_show_2d_section_xhatch` drawing setup file option.

4. After you select a valid cross section name, define how to display the sectioned area. Select one of the visibility styles from the **Sectioned Area** list:
    - **Full**
    - **Half**
    - **Local**
    - **Full (Unfolded)**
    - **Full (Aligned)**

    Note: You can display **Full** and **Local** cross-section visibility by defining two cross-sections; one with **Full** visibility and the other with **Local** visibility. Only one section with Full visibility can exist at any time.

5. If you are displaying a **Half**, **Local**, or **Full (Aligned)** cross-section, you must define placement references. Click the yellow reference collector in the table and select the appropriate reference on the drawing:
    - **Half**—Select a datum plane reference. The reference name is listed in the table if the reference is perpendicular to the view..
    - **Local**—Select a point reference. Make sure the point reference is outside any other section breakout splines. The geometry name is listed in the table and is displayed on the drawing.
    - **Full (Aligned)**—Select an axis reference. All cutting planes of the offset section should contain the reference axis. The axis name is listed in the table and is displayed on the drawing.

6. If you are displaying a **Half** cross-section, define the side of the reference plane that should be sectioned by clicking the drawing on the side of the reference plane.

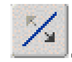
    If you are displaying a **Local** cross-section, sketch a spline encompassing the area you want to display the section in the view.

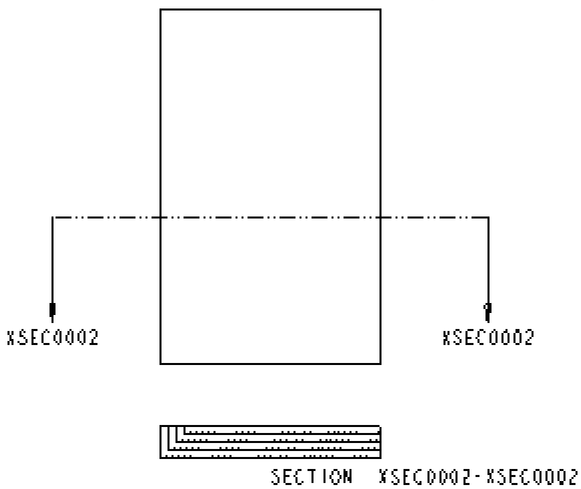
    Note: Do not initiate the spline sketch using the sketcher toolbar. Click on the drawing to begin the sketch. If you do access the toolbar, the cross-section view is cancelled and the spline becomes a two-dimensional draft entity.

    Make sure the spline does not intersect with other section breakout splines and that the end points meet. The section breakout should not be completely outside the outer border of the view, if there is one. Middle-click when the sketch is complete.

7. You can control the display of model edges by selecting **Model Edge Visibility**:

- **Total**—Display model edges behind the section planes as well as section edges.
- **Area**—Displays only section edges.

8. You can flip the direction of the cross-section by clicking . Flipping the cross-section does not reorient the model in the view. It only changes which side of cutting plane model material gets removed by the section. You should reorient the view to create a new view orientation for the cross-sectional view.
9. If desired, you can document the cross-section on a parent view by displaying arrows on that view. Click the highlighted **Arrow Display** collector in the table and select the view on the drawing. Click **Apply** to preview the arrows. Arrows are placed, including the name of the cross section at either end. (See illustration below.) You can remove the arrows by right-clicking the collector in the table and clicking remove, or deleting the arrows after they are placed.



10. You can add another 2D cross-section to the drawing by clicking and repeating steps 3 through 9.
11. To continue defining other attributes of the drawing view, click **Apply** and select the appropriate category. If you have completely defined the drawing view, click **OK**.

Note:

- When Pro/ENGINEER regenerates a cross section in a drawing, and you add or remove model geometry, it creates new edges in the drawing that you can reference. Because these edges are permanent additions, the model changes when those edges are first created. The model owns the cross-section.
- You can rename a cross-section by right-clicking the cross-section name on the 2D cross-section properties table and clicking **Rename**. Pro/ENGINEER updates the name of the cross-section in the current drawing, in the model that contains the cross-section, and in any drawing that was already using the cross-section.

To Display a 3D Cross-Section View

You can streamline the display of cross-sections within drawings by using 3-dimensional cross-sections created in the model. By default, 3D cross-sections, or zones, created in the model are available for display in the drawing views. You can display and control the cross-hatching of these 3D cross-sections within drawings.

1. Click **Insert** > **Drawing View** > **General**.
2. Click at the required location to place the general view. The general view is displayed and the **Drawing View** dialog box opens.
3. Click **Sections**. Under **Section options**, the 3D cross-section defined in the model, is displayed in the **3D cross-section** list. If more than one 3D cross-section is present in the model, they are available for selection in the **3D cross-section** list.
4. Select the required cross-section and click **Apply**. The drawing view displays the selected cross-section.
5. To display crosshatching in the view, click the **Show X-Hatching** check box.

   Note: The **Show X-Hatching** check box is selected by default. This check box does not appear as selected if you have set the value of the `default_show_3d_section_xhatch` drawing setup file option to `no`.
6. To continue defining other attributes of the drawing view, click **Apply** and select the appropriate category. If you have completed defining the view, click **OK**.

Note: To add another 3D cross-section to the drawing, you must complete the current view and insert a new general, projected, or detailed view.

Displaying 3D Cross-Sections in Child Views

The following rules apply to child views with 3D cross-sections:

- Detail views always display the same zone as their parent views. Auxiliary and projection views, are zone-dependent if you create them from parent views that display a 3D cross-section, or if you click **Same as Parent** in the **Drawing View** dialog box while modifying the auxiliary and projection views. If you change the 3D cross-section of the parent view, the change is reflected in the zone-dependent views. However, this change is not reflected in a **Copy and Align** view.
- For auxiliary and projection views whose parent view has the 3D cross-section defined, the **Same as Parent** value is displayed in the 3D cross-section list in the **Drawing View** dialog box.
- You cannot change the 3D cross-section of a detailed view.
- If you change the 3D cross-section of a child view from **Same as Parent** to another 3D cross-section, the associativity between the parent and child views is lost. To reestablish the associativity, select the child view and open the **Drawing View** dialog box. In the **3D cross-section** list, select **Same as Parent** and click **Apply**. The associativity is reestablished.

- You can change the 3D cross-section of a child view by associating it with a different parent view.
- If you change the **Section options** of a 3D cross-section parent view that has child views associated with it, to **No Section**, **2D Cross-section**, or **Single part surface**, a dialog box opens that warns you about the dependency of child views and asks you to confirm whether you want to retain the associativity or break it.

To Display a Single Part Surface View

You can create single part surface views out of a solid surface or a datum quilt. The single part surface view displays a selected surface in wireframe, however any other geometry is erased, including datums, cosmetic features, and coordinate systems. You can create regular projection views as well as single-surface projection views from single-surface views.

1. With the **Drawing View** dialog box open, click the **Sections** category. The **Section options** display in the dialog box.
2. Click **Single part surface**. The surface collector activates and you are prompted to select the surface to display.
3. Select the single part surface to display in the view. The surface highlights on the drawing and is listed in the collector. Click **Apply** to see the single part surface view. If the view is not acceptable, repeat the steps above. If the view is appropriate and no changes needs to be made, select the next appropriate category to define or click **Close**.
4. To continue defining other attributes of the drawing view, select the appropriate category. If you have completely defined the drawing view, click **OK**.

Restrictions on Aligned Cross-Sections

An aligned cross section displays an area cross-sectional view that is unfolded around an axis. Pro/ENGINEER revolves all cutting planes of an offset cross-section about the selected axis until they are oriented parallel to the screen (sheet).

Aligned cross sections create area-type cross sections. A full aligned cross-section shows an aligned total cross section of a general, projection, or auxiliary view. You can make a detailed view from an aligned cross-sectional view.

The following restrictions apply to both aligned cross-sectional views and total aligned cross-sectional views:

- You can align offset cross sections only.
- The axis around which the cutting planes revolve must be parallel to the screen.
- The axis must lie on all cutting planes.
- The axis must be coaxial with all cylinders in the section.
- If the view is to be general, orient the model so that the axis around which you are going to unfold is parallel to the screen.

- Pro/ENGINEER creates aligned and unfolded cross-sectional views as area cross sections.

Example: Cross-Section Views

You can create and save a cross section in Part or Assembly mode and show it in a drawing, or you can add a cross section to a view while you are inserting it.

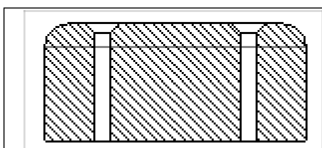
When you are creating cross sections, there are two basic methods you can define:

- A **Planar Cross Section** follows a selected datum plane or planar surface.
- An **Offset Cross Section** lets you draw a path offset from a reference plane through a solid using the Sketcher tool.

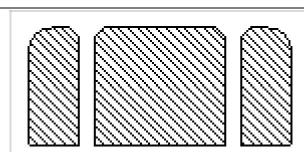
When you are inserting views, you can set each view type to use the cross section techniques shown below:

Total or Area Sections

- A *total* cross section shows not only the cross-sectioned area, but the edges of the model that become visible when a cross section is made.
- An *area* cross section displays only the cross section without the geometry.



Total Cross Section

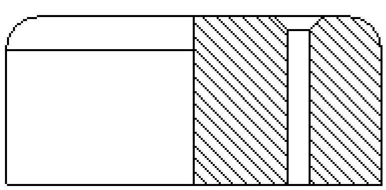


Area Cross Section

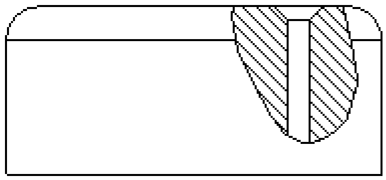
In planar area cross-sectional views, you can show all cosmetic sketches and datum curve features that lie in the cutting plane by setting the drawing setup file option `draw_cosms_in_area_xsec` to `yes`.

Full, Half, or Local Sections

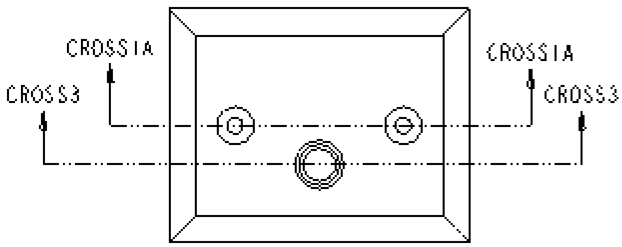
- A *full* cross section displays the cross section across the whole view, as shown in the two examples above.
- A *half* cross section shows a cross section of the model on one side of a selected plane, but not on the other side, as shown below.

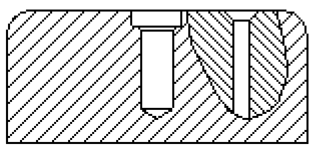


- A *local* cross section uses a breakout to see through an outer surface to a portion of an inner cross section.



- A *full & local* cross section shows a full cross-sectional view with local cross sections applied within it, as shown below.

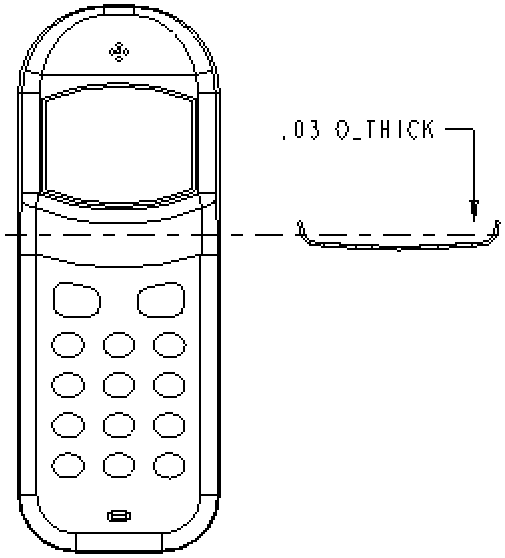




SECTION CROSS1A-CROSS1A
CROSS3-CROSS3

Revolved Cross Section

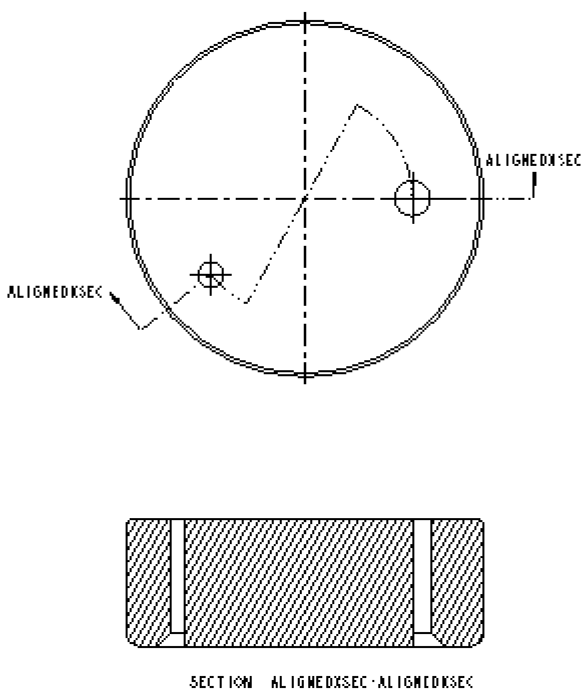
- A ***revolved*** view is a cross section of an existing view, revolved 90 degrees around a cutting plane projection. You can use a cross section created in the 3D model as the cutting plane, or you can create one on the fly while placing the view. The revolved view differs from a cross section view in that it includes a line noting the axis of revolution for the view.



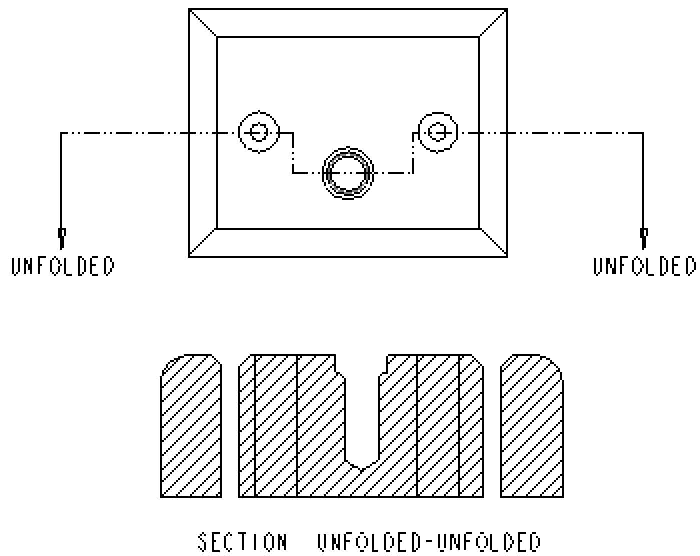
Above: Revolved View

Aligned or Unfolded

- An *aligned* cross section displays an area cross-sectional view that is unfolded around an axis, whereas a *total aligned* cross section shows an aligned cross section of a general, projection, auxiliary, or full view.



- An *unfolded* cross section shows a flattened area cross section of a general view, whereas a *total unfolded* cross section shows a total unfolded cross section of a general view.



Note: To control the display of datum curves, threads, cosmetic feature entities, and cosmetic crosshatch in a cross-sectional view, use the drawing setup file option `remove_cosms_from_xsecs`.

If you set it to `all`, you can remove datums and cosmetics from all types of cross-sectional views.

Modifying Cross-Sections

Modifying Cross-Sections

Using the **Sections** category on the **Drawing View** dialog box, you can:

- Add a section to view
- Remove a section from a view
- Replace a section used in the view
- Make the view to not show any section at al
- Change a full section view to a half or local section view.
- Change a total section view to an area section view, and vice versa.

You can also modify cross sections in the following ways:

- Change the reference points of a local cross section in a general or projection view
- Change the boundary of a local cross section in a general or projection view

- Modify a partial view with a local cross section
- Modify a broken view with a local cross section. Note: When you change the section properties of a broken view, the view temporarily displays as unbroken.
- Remove or replace a cross section
- Flip a full total cross-sectional view
- Add or remove section arrows for all sections used in the view

You can delete a cross section definition from a model without having to retrieve the model. You cannot delete a cross section when it is being used by a drawing in session. Deleting a cross section in the drawing also deletes it in other drawings. To delete a section not used in any drawing in the current Pro/ENGINEER session, select a section view and access the **Drawing View** dialog box by either double-clicking, right-clicking and selecting **Properties**, or by clicking **Edit** > **Properties**. Click on **Sections** category, in a section name list cell, select the name of the section you want to delete. Right-click and click **Delete from Model**.

Note: You can Include surfaces in drawing cross-sections by setting the `show_quilts_in_total_xsecs` drawing setup file option.

Displaying Cross-Section Datum Planes

Using the **Show/Erase** dialog box (**View** > **Show and Erase** on the menu bar), you can display set datum planes in area cross-sectional views. Set datums that you display this way behave identically to those in other views; you can perform all usual detail actions on them, including dimensioning.

Moving Cross-Section Arrows or Text

You can move cross-section arrows or cross-section text. When you select a cross-section arrow, you can move it independently of the other cross-section arrowhead. To move the arrows simultaneously, select the arrow line. To change the cross-section text, change the name of the cross section in either Part or Assembly mode.

1. Do one of the following:

- Select text to move independently of the arrows.
- Select an arrowhead to move the arrow and text together, independently of the other cross-section arrowhead.
- Select an arrow line to move the arrows simultaneously.

2. Click to place the arrow in the new location.

To Modify Cross-Section Text

1. On the menu bar, click **Format** > **Text Style**.
2. Select any of the cross section names.

3. In the **Text Style** dialog box, clear the **Default** check box for **Height**, and type a new value.
4. Clear the **Use Default** check for **Width Factor**, and type a new value.
5. Click **Apply**. The height and width of the selected item changes.
6. To reset the text to the old style, click **Reset Settings**, and then click **Apply**.

Note: To show or erase a cross-sectional view name, use the **Show/Erase** dialog box (**View** > **Show and Erase**).

Controlling the Cutting Line Display

You can control the display of the cutting line in a cross-sectional view by setting the following drawing setup file options.

1. Click **File** > **Properties.** The menu manager opens.
2. Click **Drawing Options**. The **Options** dialog box opens.
3. Set the following drawing setup file options:

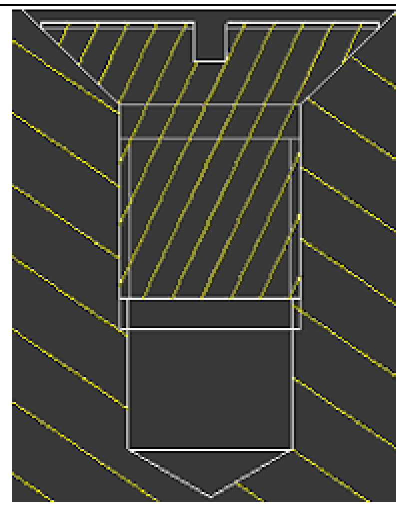
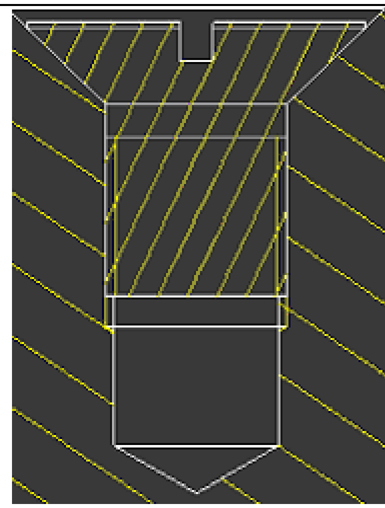
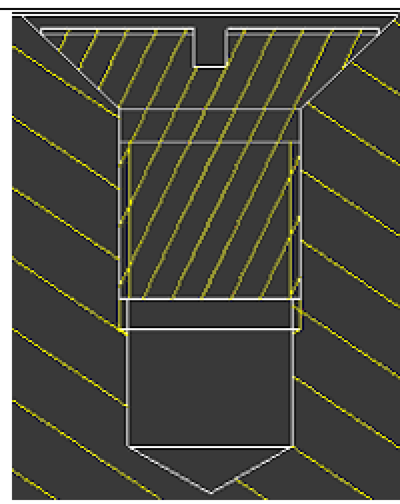
- `cutting_line`—Determines the standard (ANSI, ISO) for the cutting line display
- `cutting_line_adapt`—Sets the display of all line fonts used to display cross-sectional views adaptive so that they begin in the middle of a complete line segment and end in the middle of a complete line segment
- `cutting_line_segment`—Controls the length of the thickened portion of the cutting line

Showing Threads in the Top and Side Views in Drawings

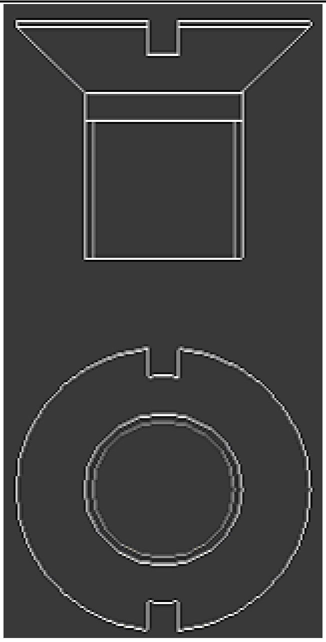
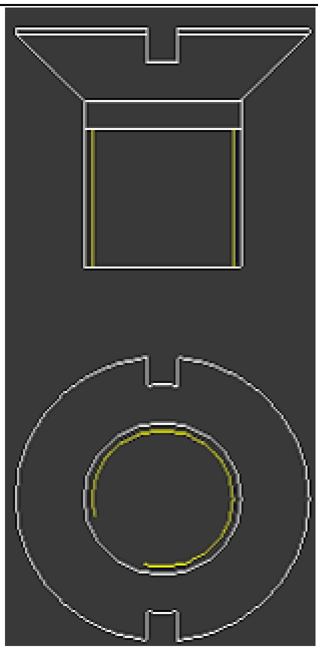
To display threads in drawings according to the ANSI, ISO or JIS standard, use the `thread_standard` drawing setup file option in conjunction with `hlr_for_threads`.

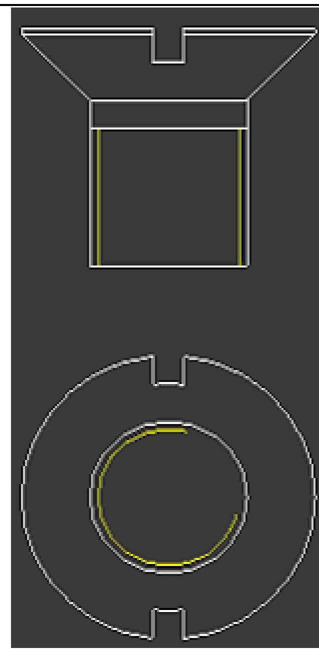
- To display the threads in accordance with the ANSI standard, set `thread_standard` drawing setup file option to `std_ansi`, `std_ansi_imp`, or `std_ansi_imp_assy`.
- To display the threads in accordance with the ISO 6410 standard, set `thread_standard` drawing setup file option to `std_iso`, `std_iso_imp`, or `std_iso_imp_assy`.
- To display the threads in accordance with the JIS standard, set `thread_standard` drawing setup file option to `std_jis`.

For assembled thread the display is as follows:

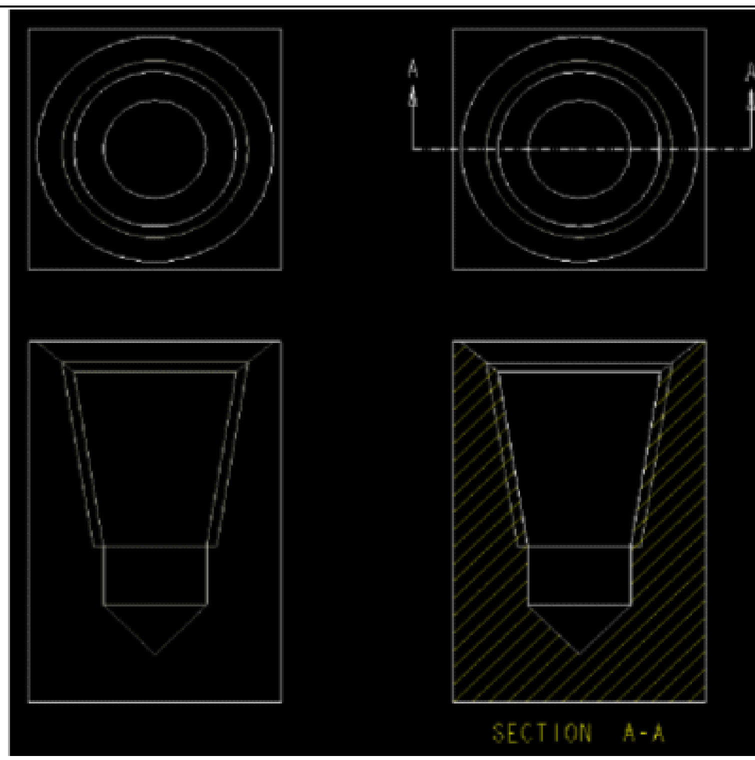
thread_standard	Display
std_ansi_imp_assy	
std_iso_imp_assy	
std_jis	

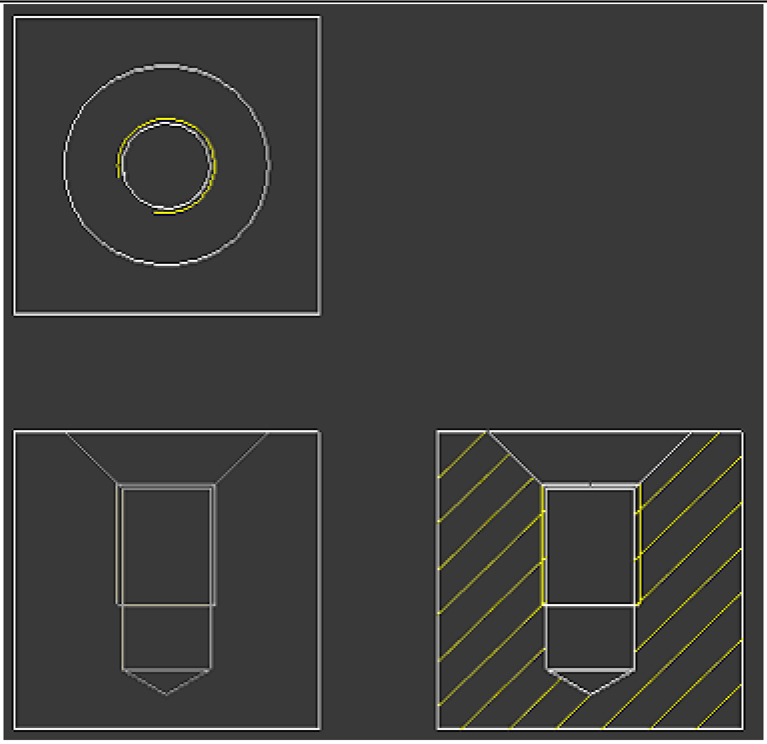
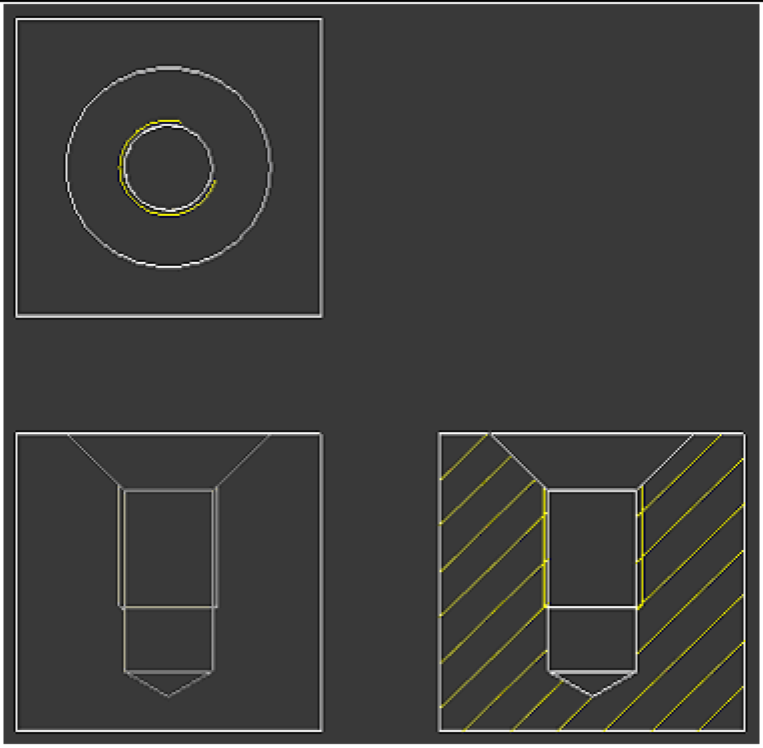
For visible thread the display is as follows:

thread_standard	**Display**
std_ansi_imp_assy	
std_iso_imp_assy	

thread_standard	**Display**
std_jis An incomplete thread is always displayed. In a view seen from one end of a thread, the minor diameter of the thread is represented by an incomplete circle almost equivalent to ¾ of its complete circumference. The open quadrant is at the upper right or at the lower left.	

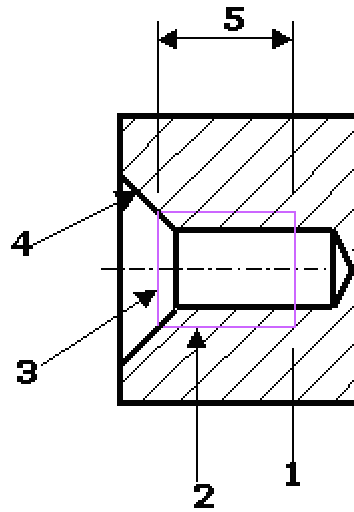
For hidden thread the display is as follows:

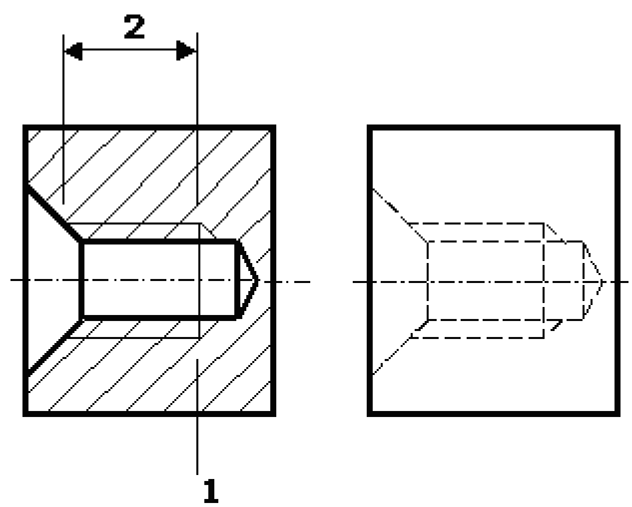
thread_standard	**Display**
std_ansi_imp_assy	

thread_standard	Display
std_iso_imp_assy	
std_jis An incomplete thread is always displayed. In a view seen from one end of a thread, the major diameter of the thread is represented by an incomplete circle almost equivalent to ¾ of its complete circumference.	

Note:

- While inserting the cosmetic thread, if you select **UpTo Surface** as the depth of a cosmetic thread, then in the drawing the useful length limit of the thread that is represented by the intersection between the cosmetic thread quilt and the reference surface is invisible when `hlr_for_threads` drawing setup file option is set to `yes`.

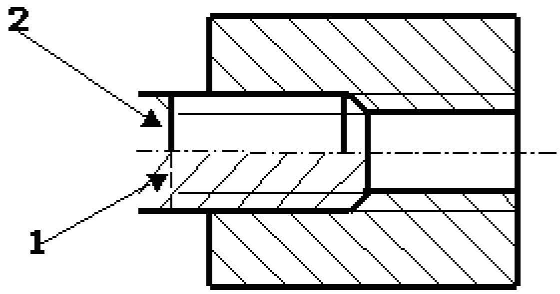
<table>
<tr><td><code>hlr_for_threads = no</code></td><td>

1. Start
2. Cosmetic Thread Quilt
3. Intersection between cosmetic thread quilt and reference surface
4. Reference surface
5. Depth</td></tr>
<tr><td><code>hlr_for_threads = yes</code></td><td>

1. Start
2. Depth</td></tr>
</table>

Note: If you change the setting for the `hlr_for_threads` drawing setup file option, then the drawing reflects the changes only after you update the drawing by clicking **View** > **Update** > **All Sheets**.

- While inserting the cosmetic thread, if you select any other option for defining the Depth, then in the drawing, the useful length limit is always displayed in thick solid line or thin dash line according to whether it is visible or hidden.



1. Hidden useful length limit

2. Visible useful length limit

- In the ISO-standard display, if the bolt diameter equals the hole thread diameter, and the bolt thread diameter equals the hole diameter, then the bolt covers the hole's crosshatch if one of these conditions exists:
    - The bolt and hole are coaxial.
    - Both axes are in the cross section.
    - The cross section itself is planar and parallel to the screen (normal to the viewing direction).
- In the case of an offset cross section, the portion of the cross section that intersects the bolt or hole pair must be planar and parallel to the screen.

To Set the View Visibility of an Unfolded Section

1. Create a drawing or open an existing drawing.
2. Double-click a drawing view. Alternatively, select the view, right-click and click **Properties**, or click **Edit** > **Properties**. The **Drawing View** dialog box opens.
3. Click **Sections** under **Categories**.
4. Under **Section options**, click **2D cross-section**.
5. Click  to add a 2D cross-section.
6. Select an existing 2D cross-section in the **Name** list. Alternatively, click **Create New** in the **Name** list to create a new 2D cross-section.
7. Select **Full(Unfold)** or **Full(Aligned)** in the **Sectioned Area** list.
8. Click **Apply**.
9. Click **Visible Area** in the **Categories** list and select one of the following options in the **View visibility** list.

- **Full View**—Sets full view visibility.

- **Half View**—Sets half view visibility. You must select a reference plane on the selected general drawing view to specify the half view reference plane. For a section with **Model edge visibility** option set to **Total**, select the reference plane within the section view in the graphics window.
- **Partial View**—Sets partial view visibility. You must click the reference point in the geometry collector and select a reference point on the selected general drawing view. Click the Spline boundary collector and define the boundary of the partial view.
- **Broken View**—Sets broken view visibility. You must click  to add breaks. Click the **1st Break Line** collector and create a break line. Additionally, click the **2nd Break Line** collector and create a break line.

    Note: Ensure that both break lines are either horizontal or perpendicular.

10. Click **OK**.

Note the following restrictions when creating an unfolded section with broken view visibility:

- The unfolding direction of the section must be parallel to one of the side edges of the view boundary or view frame.
- Break lines of breaks parallel to the unfolding direction must pass through all sections.
- Break lines of breaks perpendicular to the unfolding direction can pass through only one section.

About Switching the View Visibility of an Unfolded Section View

You can switch the visible area of an unfolded section from full to half, partial, broken, and vice versa. However, some attributes of the visible area may be lost during the switching due to missing references.

If a reference of the original visible area is missing, you are prompted to select a new reference.

If the reference related to a break line is missing, the break is removed and a message informs you about the deletion of the break that has a missing reference.

Annotations that have lost their references during the switching are erased. Additionally, annotations that are not valid for the switched visible area are erased. In both cases, a message informs you about the erased annotations. The switching of visible areas does not affect the orientation of annotations.

To Switch an Unfolded Section View to a Full Section View

1. Double-click an unfolded section view. Alternatively, select an unfolded section view, right-click and click **Properties**, or click **Edit** > **Properties**. The **Drawing View** dialog box opens.
2. Click **Sections** under **Categories**.

3. Select **Full** in the **Sectioned Area** list.
4. Click **OK**.

Note:

- For a full unfolded section with full view visibility, the orientation of the view is based on the full section and not on the unfolded section.
- For a full unfolded section with half view visibility, if the view reference plane is not normal to the view, this reference plane is not used to define the half view.

Working with Crosshatches

About Working With Crosshatches for 2D Sketches and Flat Surfaces

Use the **Edit** > **Hatch/Fill** command to hatch or fill flat surfaces and 2D sketches.

Remember the following points when working with crosshatches for flat surfaces:

- If you select more than one surface to create crosshatches, all these surfaces form a section and share the same properties.
- Crosshatches are created for flat surfaces in their respective views.
- Crosshatches move if you move their associated surfaces.
- Crosshatches are updated if you update the appearance of the associated surfaces in their respective views.
- If you convert flat surfaces to non-flat surfaces, crosshatches are automatically removed.
- If a surface becomes invisible, its associated crosshatches also becomes invisible.

About Modifying Crosshatches

Using the **Edit** > **Properties** command, you can modify crosshatches of detailed views and individual members of part and assembly cross-section views.

- **Assembly cross-sections**—When modifying an assembly cross-section, you can modify crosshatches for the entire section, or selected component sections, or selected area sections.
- **Part cross-sections**—When modifying a part cross-section, you can modify crosshatches for the entire section or selected area sections.
- **Flat Surfaces**—You can modify crosshatches associated with flat surfaces in their respective views. Crosshatches maintain a parametric relationship with their associated surfaces and are updated whenever you modify the surfaces.
- **Detailed views**—The crosshatch of a detailed view can follow that of its parent view, or you can make the detailed view independent of its parent. To modify the crosshatch of a detailed view, you must make the detailed view independent of its parent.

Note: If you have overlapping crosshatch patterns in your drawing (any combination of crosshatched cosmetic features, cross-sectional views, or draft cross sections), move the pointer over the crosshatch area, right-click, and select **Pick From List** to select the crosshatch to modify.

Using Smart Default Crosshatches

When you create an assembly cross-section, Pro/ENGINEER automatically applies a "smart" crosshatch pattern on parts that provide a superior visual representation of the cross-section. Smart default crosshatching applies crosshatch spacing appropriate to the model size and assigns different angles to different parts in the assembly. Smart crosshatching uses a randomized slant angle between adjacent components, making it easier to distinguish different parts in assembly drawings and also reducing the amount of time required to clean up drawings with cross-sections and crosshatching.

The following rules apply to smart default crosshatches:

- Smart default crosshatches affect newly created cross-sectional views only. When you retrieve previously saved drawings, smart crosshatch is not in effect.
- For assembly cross-sections, smart default crosshatches affect both spacing and angle. For part cross-sections, smart default crosshatches affect spacing only. By default, the angle is 45 degrees for parts.
- To override smart default crosshatches for newly created cross-sections, define the cross-section parameters `default_xhatch_spacing` and `default_xhatch_angle` prior to creating a new drawing view. Cross-sections of other surrounding parts that are not defined adjust accordingly.
- Smart default crosshatches are no longer valid after you modify the crosshatch properties of a view.
- Modifications that you make in the Drawing mode do *not* display in Part mode or Assembly mode.

Controlling the Spacing and Angle of Crosshatch Using Parameters

By setting parameters, you can control the default spacing and angle of a crosshatch in newly created planar and offset cross-sections. To control the display, set the number parameters using `default_xhatch_spacing` and `default_xhatch_angle`. These parameters only affect new cross-sections—they do not affect cross-sections previously created in a part or assembly.

To Create and Modify Crosshatches for Flat Surfaces

1. Open a drawing with one or more flat surfaces.
2. Select **Surface** from the drawing object filter.
3. Select one or more flat surfaces.
4. Click **Edit** > **Hatch/Fill** or right-click and click **Hatch/Fill** on the shortcut menu.

5. Type a name for the cross-section or accept the default name to open the **MOD XHATCH** menu on the Menu Manager.

   Note: Pro/ENGINEER assigns a default name to every section in the format `XSEC<nnnn>`, where `<nnnn>` is a number series unique to a drawing file.

6. A crosshatch is created by default. Click **Fill** to fill a solid color in the flat surface.
7. Click the other commands from the **MOD XHATCH** menu to modify the crosshatch. You can also retrieve a previously saved crosshatch pattern to use in the flat surface.
8. Click **Done** to apply the changes and close the **MOD XHATCH** menu.

Note: You can hatch or fill a flat surface only once. By default, all crosshatches use the Letter color, that is, the system default color. Use the **Line Style** command in the **MOD XHATCH** menu to change the color of the crosshatches. Use the **Color** command in the **MOD XHATCH** menu to change the color of the filled section.

To Create and Modify Crosshatches for 2D Sketches

1. Select 2D sketches in a drawing that form one or more closed loops.

   Note: Ensure that you do not select draft entities that overlap with each other.

2. Click **Edit** > **Hatch/Fill** or right-click and click **Hatch/Fill** on the shortcut menu.
3. Type a name for the cross-section or accept the default name to open the **MOD XHATCH** menu on the Menu Manager.

   Note: Pro/ENGINEER assigns a default name to every section in the format `XSEC<nnnn>`, where `<nnnn>` is a number series unique to a drawing file.

4. A crosshatch is created by default. Click **Fill** to fill a solid color in the 2D sketch.
5. Click the other commands on the **MOD XHATCH** menu to modify the crosshatches. You can also retrieve a previously saved crosshatch pattern to use in a 2D sketch.
6. Click **Done** to apply the changes and close the **MOD XHATCH** menu.

Note: By default, all crosshatches use the Letter color, that is, the system default color. Use the **Line Style** command in the **MOD XHATCH** menu to change the color of the crosshatches. Use the **Color** command in the **MOD XHATCH** menu to change the color of the filled section.

To Modify Section Crosshatches of an Assembly 2D Cross-Section

Use the following procedure to show or erase crosshatches associated with a section of an assembly 2D cross-section:

1. Select one or more cross-sections from an assembly drawing.
2. Click **Edit** > **Properties** or double-click or right-click and click **Properties** on the shortcut menu. The **MOD XHATCH** menu appears on the Menu Manager.

3. Click **X-Section** and **Pick** to select one or more cross-sections.
4. Click **Erase** to erase crosshatches of the selected cross-sections or click **Show** to display erased crosshatches for the selected cross-sections.
5. Click **Done** to close the **MOD XHATCH** menu.

Note: You can also use the **Erase** and **Show** commands on the shortcut menu to erase or show crosshatches, respectively, for the selected cross-sections.

To Modify Component Crosshatches of a 2D Cross-Section

Use the following procedure to show or erase crosshatches associated with components of an assembly or a part 2D cross-section:

1. Select one or more cross-sections of a 2D part or an assembly.
2. Click **Edit** > **Properties** or double-click or right-click and click **Properties** on the shortcut menu. The **MOD XHATCH** menu appears on the Menu Manager.
3. Click **X-Component** and **Pick** to select one or more components.
4. Click **Erase** to erase crosshatches of the selected components or click **Show** to display erased crosshatches for the erased components.
5. Click **Done** to close the **MOD XHATCH** menu.

Note: You can also use the **Erase** and **Show** commands on the shortcut menu to erase or show crosshatches, respectively, for the selected components.

To Modify Area Crosshatches of a 2D Cross-Section

Use the following procedure to show or erase crosshatches of selected areas associated with an assembly or a part 2D cross-section:

1. Select one or more cross-sections of a 2D part or an assembly.
2. Click **Edit** > **Properties** or double-click or right-click and click **Properties** on the shortcut menu. The **MOD XHATCH** menu appears in the Menu Manager.
3. Click **X-Area** and **Pick** to select one or more areas of a component.
4. Click **Erase** to erase crosshatches of the selected areas or click **Show** to display erased crosshatches for the selected areas.
5. Click **Done** to close the **MOD XHATCH** menu.

Note: You can also use the **Erase** and **Show** commands on the shortcut menu to erase or show crosshatches, respectively, for the selected cross-sections.

To Hide or Unhide Crosshatches

To hide crosshatches for 2D sketches or flat surfaces:

1. Select one or more crosshatches.

2. Click **View** > **Visibility** > **Hide** or right-click and click **Hide** on the shortcut menu to hide the selected crosshatches.

To unhide crosshatches for 2D sketches or flat surfaces:

1. Select one or more hidden crosshatches.
2. Click **View** > **Visibility** > **Unhide** to unhide the crosshatches.

Alternatively, click **View** > **Layer** to open the Layer Tree. Select one or more crosshatches from the hidden items layer. Right-click and click **Unhide** on the shortcut menu.

Click **View** > **Visibility** > **Unhide All** to unhide all the hidden crosshatches for 2D sketches or flat surfaces in an active drawing.

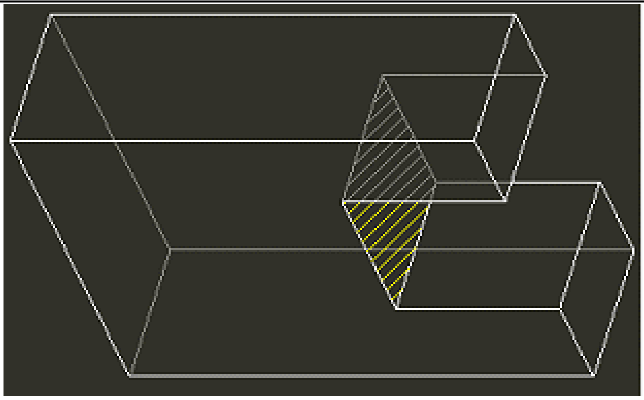
Note: If you have hidden crosshatches, you can unhide them immediately by selecting the hidden crosshatches, right-clicking, and clicking **Unhide** on the shortcut menu.

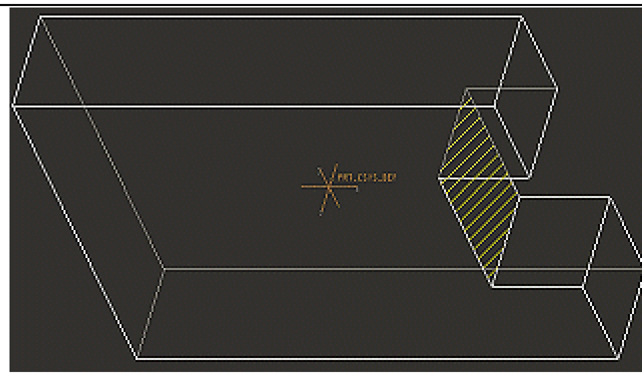
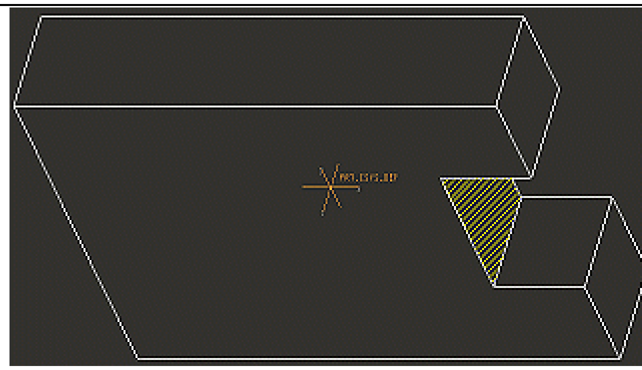
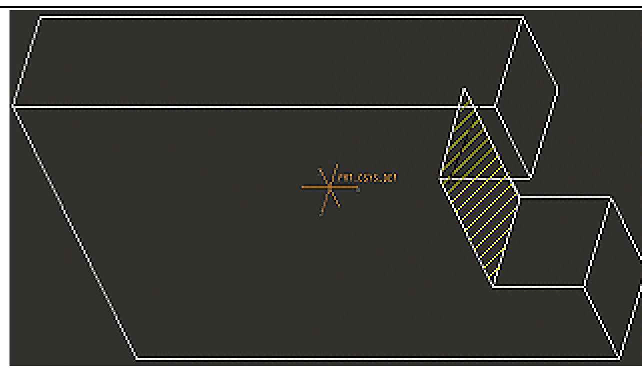
Example: Hidden Line Removal for Crosshatches

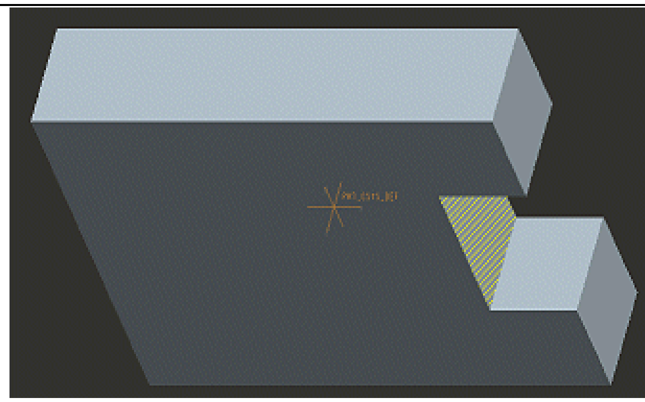
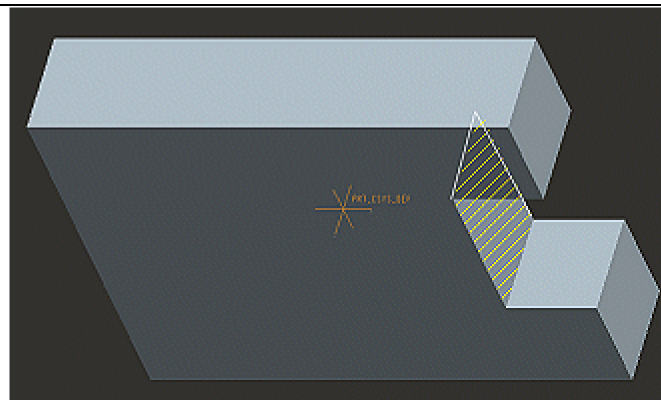
You can enable hidden line removal for crosshatches associated with:

- Flat surfaces with hidden line, no hidden, and shading display styles.
- 2D cross-sections with hidden line and no hidden display styles.

See the following table to enable or disable hidden line removal for flat surfaces and 2D cross-sections. Gray lines represent hidden crosshatches.

Display Style	HLR Enabled	Example
Hidden Line	Yes	

Display Style	**HLR Enabled**	**Example**
Hidden Line	No	
No Hidden	Yes	
No Hidden	No	

Display Style	HLR Enabled	Example
Shading	Yes	
Shading	No	

Determining Crosshatching Patterns

Crosshatching patterns may be based on the assigned material of the part. For example, if you create a cross-section that cuts through a defined material such as steel, the system looks for a crosshatching pattern that has the same name as the assigned material. If the system finds such a pattern, it automatically assigns it to the cross-section.

If the cross-section does not have a defined material, the system assigns the default crosshatching style.

Notes:

- If either of the two parameters `default_xhatch_angle` or `default_xhatch_spacing` are defined for the part containing the cross-section, these parameter settings supersede other definitions of the crosshatching style. That is, the existence of an assigned material name is no longer relevant to the assignment of the crosshatching style.
- To control the display of datum curves, threads, cosmetic feature entities, and cosmetic crosshatch in a full cross-sectional view, use the drawing setup file

option `remove_cosms_from_xsecs`. If you set it to `all`, you can remove datums and cosmetics from all types of cross-sectional views.

You can assign a name to a crosshatch pattern and save it for later retrieval by using the configuration file option `pro_crosshatch_dir` to specify a default directory for crosshatch patterns. Its value is the full path name of the default directory.

To Modify Crosshatch Patterns

1. Select the crosshatching to modify.
2. Click **Edit** > **Properties**.
3. Using the **MOD XHATCH** menu on the Menu Manager, perform one of the following:
   - Modify the spacing. To modify spacing, click **Spacing** and click **Half**, **Double**, or **Value** on the **MODIFY MODE** menu. Specify which lines to change by choosing **Individual** or **Overall**. Pro/ENGINEER automatically scales the crosshatch offset, as well as the spacing, according to the command you choose for spacing.
   - Modify the angle. To modify angle, click **Angle** and click **Individual** or **Overall** on the **MODIFY MODE** menu. Pro/ENGINEER changes only the first line of the pattern. Click an angle (**0**, **30, 45**, **60**, **90**, **120**, **135**, **150**) or **Value** to specify a different angle.
   - Modify the offset. To modify the offset, click **Offset** and type a value for the offset in drawing units.
   - Modify the line style. To modify the line style, click **Line Style** and use the Line Style dialog box to change the line style.
   - Modify the color of a filled area or a surface. To modify the color, click **Color** and select a color from the **Color** dialog box.

     Note: You can also modify the color of a filled area or a surface using the **Color** command on the shortcut menu.
   - Modify lines in the pattern. To modify lines, click **Next Line** and **Prev Line** to select a pattern line and click **Add Line** or **Delete Line**.

Note: You can also modify crosshatch characteristics associated with cosmetic features and closed datum curves.

To Create a Filled Cross-Sectional View

1. Select the existing cross hatched section.
2. Click **Edit** > **Properties** > **MOD XHATCH** > **Fill** > **Done**.
3. Once you have defined the cross-section as filled, you can use **Line Style** to set the appropriate color for the filled area, or **Save** to save it to disk with a `.xch` extension. Later, you can use **Retrieve** to read it in as a saved crosshatch pattern.

To Save a Crosshatch Pattern

1. Select the view containing the crosshatch pattern that you want to save. For an assembly cross section, you may need to use Next Xsec or Prev Xsec in the MOD XHATCH menu to select the member.
2. Click **Edit > Properties**.
3. On the Menu Manager, click **Save**.
4. Type a name for the crosshatched pattern.
5. If you have not set the configuration file option pro_crosshatch_dir, the system saves the crosshatch pattern in the current directory, giving it the name that you specified and the extension .xch. If you want to save the pattern in a separate directory containing crosshatch patterns, you must save the pattern first, and then move it to its new location using operating system commands.

To Retrieve a Crosshatch Pattern

1. Select a cross-sectional view
2. Click **Edit** > **Properties**.
3. Click **MOD XHATCH** > **Retrieve**.

   Pro/ENGINEER searches through the current directory, the `pro_crosshatch_dir` configuration file option, and the system directory. It then displays a namelist menu that contains the names of any crosshatch patterns in the current directory, all crosshatch patterns in the default crosshatch directory, and the nine standard crosshatch patterns provided with Detailed Drawings.

   Note: The selected crosshatch pattern replaces the view's (or assembly member's) current crosshatch. When you change the crosshatch type in the drawing, the model does not reflect it; conversely, if you change it in a model, its drawings do not reflect it.

About Creating a Filled Area

You can display cross-sectional drawing views as filled areas (displayed in solid color) by changing the crosshatch to fill, but you must first create the cross-sectional view.

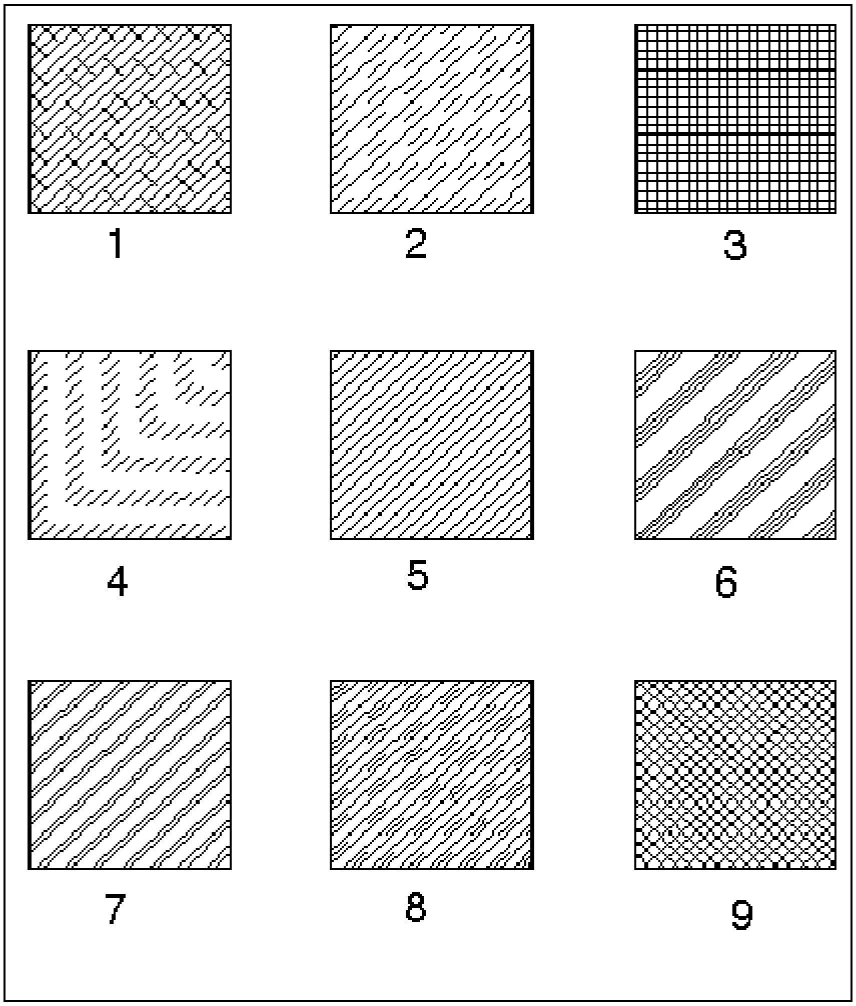
To switch between a hatched and filled cross section, double click the area and use the **Hatch** and **Fill** commands in the MOD XHATCH menu manager.

When creating filled cross-sectional drawing views, keep in mind the following rules:

- You can create filled cross sections only in views without clipping. If you attempt to change the crosshatch of a clipped view (such as a detailed view, broken view), the system issues an error message and does not allow you to proceed.
- If a cross-sectional view that you want to fill has detailed views, the system warns you that this view has dependent children views before allowing you to make the change. The detailed view appears without either crosshatch or filling; you can then set its crosshatch to be independent of its parent.

- You can modify only planar cross sections to be filled.

Examples: Crosshatch Patterns



1. Aluminum.
2. Copper.
3. Electric.
4. Glass.
5. Iron.
6. Plastic.
7. Steel.
8. Titanium.
9. Zinc.

Showing Various Model States

To Show Models in Various View States

While creating a new drawing or while modifying an existing drawing, you can show different view states in the drawing view using the **Drawing View** dialog box. You can create a drawing view for a combination state stored in a part or assembly model. This sets the drawing view to any state stored in a combination state in the model. However, drawing views do not recognize style states stored in an assembly combination state. You can add an exploded view of an assembly without exploding the assembly in Assembly mode. The Exploded state is an assembly view property that overrides any saved exploded views in the 3D assembly. You can display representations of assembly models in the drawing.

1. Do one of the following to open the **Drawing View** dialog box for an existing view:
   - Select the view and double-click.
   - Select the view and click **Edit** > **Properties**.
   - Select the view, right-click, and click **Properties** on the shortcut menu.
2. Under **Categories**, click **View States**.
3. Under **Presentation state**, select the required combination state. The presentation state lists all the available combination states. However, if a combination state has an orientation that is not parallel to the current drawing view orientation, then that combination state name is not available for selection. Additionally, for a part view, you cannot select a combination state with a part simplified representation that differs from the current part simplified representation. After you set the **Presentation state**, the **Assembly Explode State** and the **Simplified representation** boxes are updated accordingly to reflect the changes.

   Note: When you set the **Presentation state**, the **Sections** page is updated with the cross section information. If you change the cross section then, the presentation state changes. The same holds true for the view orientation of exploded states.
4. For an assembly view to display the exploded state, select **Explode Components in View** under the **View States** category and select the required explode state from **Assembly Explode State**. All saved explode states that are saved in 3D mode are available in the **Assembly Explode State** list. If you clear the **Explode Components in View** check box or change the **Assembly Explode State** to another state, then **Presentation state** changes to **No presentation**.

   Note: You can modify the explosion distances between components. However, this alters the drawing cosmetically and does not modify the actual exploded dimensions of the model.

5. Under **Simplified representation**, select the required simplified representation in an assembly. If you change **Simplified representation** to another state, then **Presentation state** changes to **No presentation**. You cannot switch part simplified representations through the **Simplified representation** box. You can change part simplified representations by selecting a new model.
6. Click **Apply**. The model appears in the drawing with the above states set.
7. Click **OK**. The **Drawing View** dialog box closes.

Note:

- Cross-section views do not support shaded display style. Therefore, if the default display style of a view is set to **Shaded** from the **Display Style** box in the **View Display** category of the **Drawing View** dialog box, and you try to place a combination state of the view that contains a cross-section, then the display style of the view changes from shaded to wireframe.
- You cannot change to a combination state that contains a different part simplified representation other than the part simplified representation that is used in the current view.
- Child views are generally dependent on parent views for most view state properties. Therefore, you cannot select a combination state for a child view. **Presentation state** is not available for child views.
- After a state is set, the view takes the settings of that state but is not parametrically linked to the state.

To Create an Exploded View

You can create and save multiple assembly explode states.

1. With the **Drawing View** dialog box open, click the **View States** category. The view state options display in the dialog box.
2. To separate the components of the model, click **Explode components in view**. The **Assembly explode state** list is enabled.
3. You can either select a predefined exploded view from the **Assembly explode state** or create a new state. If you select an existing explode state, click **Apply** to see the explode state view. If the view is not acceptable, repeat the steps above. If the view is appropriate and no changes needs to be made, select the next appropriate category to define or click **Close**.

   If you need to create a new explode state, click **Customize explode estate**. The **Explode Position** dialog box opens and you are prompted to select a component to move.

   a. Select the component to reposition on the drawing. The component highlights and the component name is listed under Select Component on the Explode Position dialog box.

   b. Under Motion Type, define how you want to move the component:

Translate

Copy Pos

Default Expld

Reset

c. Under **Motion Reference**, select a reference type from the list and then select a reference on the drawing. The reference name is displayed on the dialog box.

d. Under **Motion Increments**, define how to translate the component movement. Type or select a translation type from the Translation list.

e. Drag the component to its desired location on the drawing. The sheet coordinates for the component display under **Position**.

f. Repeat steps a. through e. for each component you need to reposition. When you have exploded the desired components, click **OK** in the **Explode Position** dialog box. Then click **Done/Return** on the **MOD EXPLODE** menu.

4. To continue defining other attributes of the drawing view, click **Apply** and then select the appropriate category. If you have completely defined the drawing view, click **OK**.

To Display Representations in Drawings

You can display representations of assembly models in the drawing:

1. With the **Drawing View** dialog box open, click the **View States** category. The view state options display in the dialog box.
2. Under **Simplified representation**, select the name of the simplified representation to use. Click **Apply**. The simplified representation is displayed on the drawing.

Note: If the Drawing View dialog box is open for a view whose model was simplified using the, now no longer supported, Represent functionality, **Restore to non-simplified state** appears in View States category. If you click **Restore to non-simplified state** and then click **Apply**, the view's model will be restored to a nonsimplified state, and normal view state properties like explode and assembly simplified representations can be accessed.

To Display Process Steps

If the manufacturing process was identified in the assembly, you can display manufacturing process steps in drawing views:

1. With the **Drawing View** dialog box open, click the **View States** category. The view state options display in the dialog box.
2. Under **Process Step**, select the name of the assembly step from the **Process assembly step** list. The view of the assembly step is displayed on the drawing.

Note:

- You can create a drawing view of a solid tool used in the manufacturing process by clicking **Insert** > **Drawing View** > **Tool**.
- If the **Drawing View** dialog box is open for a view whose model was simplified using the, now no longer supported, Represent functionality, **Restore to non-simplified state** appears in View States category. If you click **Restore to non-simplified state** and then click **Apply**, the view's model will be restored to a nonsimplified state, and normal view state properties like explode and assembly simplified representations can be accessed.

Simplified Representations

About Simplified Representations in Drawings

Using commands in the **File** > **Properties** > **Drawing Models** menu, you can create assembly and part views using simplified representations.

You must specify the simplified representation before adding a view. You can add multiple views of an assembly or a part, each of a different representation, to a drawing.

Assembly Simplified Representations

When working with assembly simplified representations in Drawing mode, you can use geometry representations—a type of advanced simplified representation. Geometry representations require less time to retrieve than the components of the assembly because Pro/ENGINEER does not retrieve any of the parametric information, only the geometry. You can use them to remove hidden lines, obtain measure information, and accurately calculate mass properties.

When working with geometry representations of assemblies, keep in mind the following:

By default, you cannot create drawing references to geometry representations (this includes dimensions, notes, and leaders). You can create references if you set the `allow_refs_to_geom_reps_in_drws` configuration option to `yes`. However, these references may become invalid if the referenced geometry changes. This option is for advanced users who are aware that some references to geometry representations may not be updated in drawings.

Note: For assemblies, Graphics representations and Symbolic representations are not available in the Drawing mode.

Part Simplified Representations

Part simplified representations in drawings are classified into two groups:

- **Representations being used**—Representations that have been added to the drawing and used by the drawing to define views.

- **Representations available**—Representations that have been added to the drawing but have not yet been used to define views.

  Note: For parts, the Geometry representations, Graphics representations, and Symbolic representations are not available in the Drawing mode.

Changing the Representation of an Assembly Model

After adding an assembly model to a drawing, you can change its representation by choosing **File** > **Properties** > **Drawing Models** > **Set/Add Rep.** Selecting a view sets the current representation to the representation shown in that view—you can select *only* views of the current drawing model.

You can use **Sel By Menu** to access a list of all representations that exist for the current assembly. The system makes the specified representation the active representation and applies it to subsequent views. If simplified representations have been made for the assembly model, it automatically retrieves the Master Representation.

When creating simplified representations in Drawing mode and making changes to them, you should keep in mind the following:

- In Assembly mode, simplified representations can contain substitute parts. In Drawing mode, you can only apply dimensions to these substitute parts by creating them. When creating the dimension you should reference the geometry of the simplified representation.
- When you make changes to a simplified representation in Assembly mode, you may lose information in the drawing. For example, when you change the status of a component by substituting a previously included component, all references to the substituted component are lost.
- When referencing an assembly or its components in a simplified representation, the system can find and use only those components that actually exist in the simplified representation. Specifically, when you orient a view, you cannot use datums belonging to or placed according to components that are not in the current simplified representation.
- You can make projected, auxiliary, revolved, and detailed views from a general view of the same simplified representation.

To Replace a View of a Simplified Representation

You can replace a view of a simplified representation (including the Master Representation) with another simplified representation.

1. Select the view. From the right mouse button shortcut menu, click **Properties**. The **Drawing View** dialog box opens.
2. Click the **View States** category.
3. Select a simplified representation from the list of available representations to use as a replacement. Click **Apply**. The view is replaced.

To continue defining other attributes of the drawing view, select the appropriate category. If you have completely defined the drawing view, click **OK**.

To Retrieve an Assembly Model Simplified Representation

1. Click **File** > **Properties**. The **FILE PROPERTIES** menu opens.
2. Click **Drawing Models** > **Set/Add Rep**.
3. Select an assembly name. The **SELECT REP** menu (in Assembly mode) appears when you retrieve the initial assembly and for all assembly models that you add subsequently. It lists the Master Representation and all simplified representations that exist for the current assembly. If none have been created for the assembly model, it does not appear, and the system automatically retrieves the Master Representation.
4. Select a representation. The system applies that representation to subsequent views. You can accept the Master Representation as the default, or select a different representation to retrieve.

About Creating Views of Part Simplified Representations

You can use part simplified representations as drawing models to create general views. Child views such as projection views, detail views, and so on use the general view of the same part simplified representation as the parent view.

When you display dimensions in the view, only dimensions of features included in the part simplified representation are displayed. Displaying annotations in the view of the part simplified representation is similar to displaying annotations in a view of a family table instance. For example, a view of a part simplified representation shows a note even if the parent feature of the note is excluded from the part simplified representation. If a note belongs to an annotation feature and the parent feature of the note is excluded from the part simplified representation, then the note is not shown in the view of the part simplified representation.

Note: Driven dimensions are stored with the part, and not with the part simplified representation.

You can set the part simplified representation as a drawing model for a new drawing by selecting the required representation from the **Open Rep** dialog box. This dialog box opens when you create an empty new drawing for a model that has part simplified representations that you can use to create a drawing.

You can set the part simplified representation as a drawing model for the active drawing by clicking **File** > **Properties** > **Drawing Models** and clicking **Set/Add Rep** on the **DWG MODELS** menu.

Note:

- You must convert the part simplified representations created in Pro/ENGINEER Wildfire 2.0 and earlier releases to Pro/ENGINEER Wildfire 3.0 part simplified representations to make them usable as models for creating drawings views. You can perform this conversion through the **View Manager** dialog box, that can be accessed, in the 3D mode.

- After you convert part simplified representations from the Pro/ENGINEER Wildfire 2.0 or earlier releases to Pro/ENGINEER Wildfire 3.0 part simplified representations, these conversions cannot be undone
- You can create drawing views of user-defined part simplified representations only.

To Create Drawing Views from Part Simplified Representations in an Active Drawing

When creating or editing drawing views of parts with part simplified representation, you can insert a drawing view for the part simplified representation within the same drawing sheet.

1. Click **File** > **Properties**. The **File Properties** menu appears.
2. Click **Drawing Models**. The **DWG MODELS** menu appears.
3. Click **Set/Add Rep**. The **SELECT REP** menu appears.
4. Select the required representation from the available list and click **Done/Return**.
5. Click **Insert** > **Drawing View** > **General**.
6. Click at a location where you want to place the general view of the part simplified representation. The **Drawing View** dialog box opens.
7. Click **View States**. The **View States** category page is displayed in the **Drawing View** dialog box. The **Simplified representation** box displays the current representation. You cannot modify this representation.
8. The rest of the options in the **Drawing View** dialog box are optional. If required, change these options and click **Apply**. The view of the part simplified representation is placed in the drawing.
9. Click **OK**. The **Drawing View** dialog box closes.

Note: You can create drawing views only of part simplified representations that are created in Pro/ENGINEER Wildfire 3.0 or later, or updated to Pro/ENGINEER Wildfire 3.0 or later.

To Create a New Drawing from Part Simplified Representations

1. Click **File** > **New**. The **New** dialog box opens.
2. Select **Drawing** to create an empty drawing for the current part.

   Note: The part should contain part simplified representations.
3. Clear the **Use Default Template** check box and click **OK**. The **New Drawing** dialog box opens. By default, the **Default Model** box displays the current part. The dialog box also displays the default values for the template, size, and orientation options. If required, you can change these values.
4. Click **OK**. The **Open Rep** dialog box opens.

5. Select the required representation and click **OK**. The drawing becomes active.
6. Click **Insert** > **Drawing View** > **General**.
7. Click at a location where you want to place the general view of the part simplified representation. The **Drawing View** dialog box opens.
8. Click **View States**. The **View States** category page is displayed in the **Drawing View** dialog box. The **Simplified representation** box displays the current representation. You cannot modify this representation.
9. The rest of the options in the **Drawing View** dialog box are optional. If required, change these options and click **Apply**. The view of the part simplified representation is placed in the drawing.
10. Click **OK**. The **Drawing View** dialog box closes.

Note: You can create drawing views only of part simplified representations that are created in Pro/ENGINEER Wildfire 3.0, or updated to Pro/ENGINEER Wildfire 3.0.

To Set a Part Simplified Representation as the Current Representation of the Drawing Model

1. Set a model as the current drawing model.
2. Click **File** > **Properties**. The **FILE PROPERTIES** menu appears.
3. Click **Drawing Models**. The **DWG MODELS** menu appears.
4. Click **Set/Add Rep**. The **SELECT REP** menu appears, which lists all the simplified representations of the current drawing model.
5. Select the required simplified representation that you want to set as the current representation to be used in by the drawing. The selected simplified representation is set as the current representation of the drawing model.

About Removing Simplified Representations from Session

Once you add a representation of an assembly or a part to a drawing, it remains in session until you explicitly remove it. If a drawing view refers to the particular simplified representation you choose to delete, Pro/ENGINEER does not allow you to delete it while the drawing is in session, and attempts to retrieve the master representation.

For assembly simplified representations, if you do remove a representation from session and delete it, then when you retrieve the drawing that has views that use the simplified representation that you deleted, Pro/ENGINEER prompts you to select an existing representation from the **Open Rep** menu, and assigns this representation to all views that previously referred to the deleted representation.

For a part simplified representation, if you remove a representation from a session and delete it, then you cannot retrieve the drawing that has views that use the deleted part simplified representations.

When removing representations from a session, if the removed representation is the current representation and is not the last representation of the drawing model in session, the drawing model stays as the current drawing model.

If the removed representation is the last representation of the drawing model in session, the drawing model is removed from the drawing. If the removed drawing model was the current drawing model, Pro/ENGINEER randomly selects a drawing model and sets it as the new current drawing model.

To Remove Part Simplified Representations from the Session

1. Set a part as the current drawing model.
2. Click **File** > **Properties**. The **FILE PROPERTIES** menu appears.
3. Click **Drawing Models**. The **DWG MODELS** menu appears.
4. Click **Remove Rep**. The **RMV REP** menu appears, which lists all the added simplified representations of the current drawing model.
5. Click a simplified representation to remove it from the session.

   Note: If the selected representation is used by the drawing, then it cannot be removed from the session. In this case Pro/ENGINEER displays a warning in the message area stating that it cannot remove the part simplified representation from the session as views in the drawing are using it.

Modifying View Display

About Shaded Views in Drawings

Shaded views inherit all the attributes of the 3D model, including the color, texture, reflection, opaqueness, material, and refraction. External rendering effects are not applied in shaded views. The shading display style also supports exploded states and process states of assemblies.

You can set the display style of the selected view to **Shading** by clicking **View Display** > **Display Style** > **Shading** in the **Drawing View** dialog box. The selected view appears shaded.

Colors of shaded views are associated with the model. If the model color is changed, the corresponding shaded views are also updated.

Note: If you set the display style to **Shading**, you cannot set the color designation options under **Colors come from**.

View Types That Support Shaded Display Style

The following view types support shaded display style:

- **General**
- **Projection**

- **Detail**—You can control the display of the detail shaded view independently or the view can have its parent view display style by selecting the **Use Parent View Style** check box in the **Drawing View** dialog box. However if the parent is an X-section, then the view is displayed in wireframe regardless of whether the **Use Parent View Style** check box is selected or not.
- **Auxiliary**—For the auxiliary view, the projection reference is shown over the shaded parent view when you are in the **View Type** category page of the **Drawing View** dialog box.
- **Broken**—All six break line styles are visible in shaded views.
- **Half**—The symmetry line display is retained in the shaded view.
- **Partial**—The partial view boundary is not visible in shaded views even if **Show spline boundary on view** in the **Visible Area** page of the **Drawing View** dialog box is selected.
- **Zone**—The display of X-hatching is controlled by the **Show X-Hatching** option in the **Section** page of the **Drawing View** dialog box.

View Types That Do Not Support Shaded Display Style

The following options are not available if you set the display style to Shading:

- **2D cross-section** under **Sections**
- **Revolved** under **View Type** (A revolved view can only have a wireframe display style.)

Note: The display of shaded views depends on certain capabilities of the graphics card. Some view types require flexible calculations that are not supported by certain graphics cards. Performance may also be affected if you are working with large assemblies and older graphics cards.

Plotting Options for Shaded Views

You can plot shaded views in the drawing mode. The following drivers support printing of shaded views:

- HPGL2

  Note: For the HPGL2 driver to print shaded views, the printer must support the HP RTL extension.
- PostScript
- Color PostScript
- MS Printer Manager
- Plot to Screen
- PDF
- DXF/DWG

Note: Shaded views cannot be plotted if you set the drawing to a non-supported file type.

Behavior of Shaded Views in Drawings

When two shaded views overlap, the newer view is displayed on top of the view that was created earlier. Non-shaded views always show through the shaded views when the two view types overlap. In a shaded view, surfaces and quilts also appear shaded.

Shaded views do not display tangent edges. The **Tangent Edge Display Style** option in the **View Display** page of the **Drawing View** dialog box is not available for selection for shaded views. Even if the tangent edges are not displayed in shaded views, you can select these edges for attachment of references.

Note:

- If you zoom into a shaded view in the **Preview** mode of the **File** > **Open** dialog box, the view may appear pixellated.
- To highlight the edges of the shaded view, set the `show_shaded_edges` configuration option to `yes`.
- You can control the display of datum curves in shaded views by using the `shade_with` configuration option.

Modifying View and Edge Display

You can define display settings that apply to all drawing views or you customize the display of individual views. For example, you may show one view of a model with hidden lines, and then depict a detailed view of the model with wireframe.

The various view display options can be applied to individual views, edges, or assembly members. If multiple items are selected, any display changes you make are applied to all the selected views.

Note:

- Once you have set the display mode for a specific view, it remains set regardless of the setting in the **Environment** dialog box, unless you choose **Default**
- The `hlr_for_quilts` configuration file option controls how the system displays quilts in the hidden line removal process.
- The `hlr_for_pipe_solid_cl` drawing setup file option controls how hidden line affects pipe centerlines. This option only controls pipes created in the Piping mode, not on pipe features in a part.

To Modify View Display

If you select multiple views, only the **View Display** category is displayed in the **Drawing View** dialog box. All display options are available in this page of the dialog box. Any changes you make are applied to all the selected views.

1. Select one or more views, right-click, and select **Properties**. The **Drawing View** dialog box opens.
2. Click the **View Display** category. The **View display** options are displayed in the dialog box. You can modify the view display in the following ways:

- Click **Use Parent View Style** to set the display of the detailed view to be similar to the parent view display. When cleared, the detailed view display is independent of its parent view and you can modify the display as necessary.
- Define how to show the model geometry by selecting a display style from the **Display style** box:
    - **Default**—When you import drawings from Pro/ENGINEER Wildfire 2.0 or earlier releases that were saved with the **Default** option, this option is retained for these drawings. Once you update these drawings in Pro/ENGINEER Wildfire 3.0, the **Default** option changes to **Follow Environment** and the drawings are considered as Pro/ENGINEER Wildfire 3.0 drawings.
    - **Follow Environment**—Uses the setting from **Tools** > **Environment** > **Display Style** or the view display style icon in the Pro/ENGINEER graphics window.
    - **Wireframe**—Shows all edges in wireframe style.
    - **Hidden**—Shows all edges in hidden line style.
    - **No Hidden**—Removes all hidden edge from view display.
    - **Shading**—Displays shaded views.

  Note:

    - If you have set the view display mode to **Wireframe**, Pro/ENGINEER does not erase or redisplay its edges until you change the view display mode to **Hidden** or **No Hidden**.
    - Drawing views saved with the **Default** option and imported from Pro/ENGINEER Wildfire 2.0 or earlier releases are retrieved in wireframe mode. These views are retrieved in wireframe even after they are saved in Pro/ENGINEER Wildfire 3.0, unless you specifically change the display to follow the environment or change it to shading. To update these views, change the display setting to any other display type and back to any of the Pro/ENGINEER Wildfire 3.0 settings and save the drawing. Once you update these drawings in Pro/ENGINEER Wildfire 3.0, the drawings are considered as Pro/ENGINEER Wildfire 3.0 drawings and the **Default** option in the **Display Style** list is no longer available for these drawings.
- Define how to show the tangent edges on the model by selecting a tangent edge display style from the **Tangent edges display style** box:
    - **Default**—Uses the setting from **Tools** > **Environment** > **Tangent Edges**.

  - **None**—Turns off the display of tangent edges.
  - **Solid**—Displays tangent edges.
  - **Dimmed**—Displays tangent edges in dimmed color.
  - **Centerline**—Displays tangent edges in centerline font.
  - **Phantom**—Displays tangent edges in phantom font.

- Determine whether hidden lines are to be removed for quilts by selecting the appropriate option under **Hidden line removal for quilts**:
  - **Yes**—Hidden lines are removed from the view.
  - **No**—Hidden lines are shown in the view.
- Define whether to show the skeleton model by selecting one of the following options under **Skeleton model display**:
  - **Hide**—Skeleton model is not displayed.
  - **Show**—Skeleton model is shown in the view.
- Define whether to enable or disable hidden lines for crosshatches by selecting the appropriate option under **Hidden line removal for xhatches**:
  - **Yes**—Hidden lines are removed from the view.
  - **No**—Hidden lines are displayed in the view.
- Define how to display cable geometry in the drawing by selecting the required option under **Cable display**:
  - **Default**—Uses the display setting from **Tools** > **Environment**.
  - **Centerline**—Display cable geometry in centerline font.
  - **Thick**—Display cable geometry in thick font.
- Define where the drawing should look for color designations by selecting one of the following options under **Colors come from**:
  - **The drawing**—Drawing colors are determined by drawing settings.
  - **The model**—Drawing colors are determined by model settings.

  Note: Model colors and assigned drawing colors in a drawing for a process assembly are always superseded by any existing process assembly colors.
- Define whether weldment cross-sections should be shown in the drawing by selecting one of the following options under **Weldment xsection display**:
  - **Hide**—Weldment cross-sections are not displayed in the view.
  - **Show**—Weldment cross-sections are shown in the view.

3. To continue defining other attributes of the drawing view, click **Apply** and select the appropriate category. If you have completely defined the drawing view, click **OK**.

Note: Once you have set the display mode for a specific view, it remains set regardless of the setting in the **Environment** dialog box, unless you select **Default** in the **Display Style** list in the **Drawing View** dialog box.

To Modify Individual Edge Display

1. Click **View** > **Drawing Display** > **Edge Display**. The **Select** dialog box opens and **EDGE DISP** menu appears in the menu manager.
2. Select the edge or edges to modify. The selected edges highlight.
3. Define the edge display using the options on the **EDGE DISP** menu. Click **Done** when you have modified the desired edges.

Note: Keep the following in mind when modifying individual edges on a drawing: You cannot use the **EDGE DISP** menu to erase the following items:

- Boundaries of all cross sections
- Outside edges intersecting clipped splines in partial or detailed views
- Cosmetic threads in drawing views (choose **Show and Erase** from the **View** menu; then click the **Cosmetic Feature** button in the **Type** box of the **Show/Erase** dialog box)
- Silhouette edges, except for the following:
    - Silhouette edges of cylinders and cones
    - Silhouette edges of torii that are arcs, that is, the axes of the torii are either parallel or perpendicular to the screen
    - Silhouettes of general surfaces of revolution whose axes are parallel to the screen

  Note: If you erase one silhouette edge, the system erases all silhouette edges that belong to the same surface.

To Modify the Line Style of Assembly Members

1. Click **View** > **Drawing Display** > **Component Display**. The menu manager opens.
2. Select the assembly components. Click **OK** in the confirmation box. The **Memb Style** menu opens.
3. Change the style by selecting a command:

- **Standard**—Displays the selected member view in solid line style.
- **PhantomOpque**—Displays the selected member view in phantom line style.

- **PhantomTrnsp**—Displays the selected member view in phantom line style, but the hidden line removal process does not affect it.
- **User Color**—Creates a user-defined color and assigns it to an assembly member.

4. Click **Done**.

Tip: Using Model Colors in Drawings

You can toggle color display of selected views in Drawing mode between the assigned drawing colors and the colors used in the original model.

Important Points about Using Model Colors in Drawings

- Views that are displayed in the original model colors are plotted (printed) as such.
- Changing the color of individual geometry does not change the colors in views that display the original model colors.
- If you change model colors in the associated model, the new model colors are updated automatically in the drawing, if you set the drawing views to use model colors.
- Hidden lines in drawing views always take on the original standard hidden line color in the model.
- Any assigned process assembly colors always supersede the setting for using model colors or assigned drawing colors in the drawing.

Tip: Simplifying Edge Selection

When drawing views display as No Hidden, you can disallow selection of edges in drawings.

The `select_hidden_edges_in_dwg` configuration file option disallows the selection of No Hidden edges by rejecting edges behind the first surface in the viewing plane. The following requirements apply:

- Selection is disallowed for drawing views in No Hidden display mode only.
- View regeneration is required on cross-sectional views.
- You can disallow the selection of edges in general and cross-section views.

Defining View Origin

Defining the View Origin

By default, the origin of a drawing view is in the center of its outline. You can reset the origin of a drawing view by parametrically referencing model geometry or defining a location on the drawing sheet. Designating an origin identifies the view location on the drawing and prevents it from shifting whenever the model geometry changes.

You can reset the origin of general, auxiliary, and broken views any time after you create them, but it does not change the current position of the view. The effect of the new origin is noticeable only when the system updates the views to reflect changes in model geometry. However, if you reset the origin of a perspective view, it *alters* the current position of the view because the view origin is part of the orientation.

Note: You cannot change the view origins of total unfolded cross-sectional views.

To select the point for the origin, you can choose a model edge, datum curve, datum point, coordinate system, or cosmetic feature entity. However, when selecting a point, keep the following in mind:

- For a general view, the selected point becomes fixed.
- For a projection and auxiliary view, the system passes the selected point onto a ray passing through the origin of the parent view in the direction of projection. This projected point becomes the origin of the view.
- To select a coordinate system as the origin of a newly created view or of a modified view, specify its name as the value for `drawing_view_origin_csys` configuration option . If you do not want the system to use a coordinate system that you set previously, specify the value as `no`.

To Define the View Origin

The placing and positioning of views on drawings uses the view origin as a reference, with the exception of custom view alignments. You can specify the view origin using a location on either the model or the drawing sheet.

1. With a view selected and the **Drawing View** dialog box open, click the **Origin** category. The **View origin options** display in the dialog box.
2. Define the location of the view origin in one of the following ways:

- **View origin**—Define the origin using the model.
    - To customize the view origin, click On item and then select the model reference you want to use as the view origin. The item is listed in the collector. The view will reference this item when positioned on the drawing.
    - To use the model center, which is the default setting, make sure Model center is selected.
- **View location in sheet**—Define the origin using drawing sheet measurements.

Note: If the geometry referenced in a view is suppressed or deleted, the system warns you that model geometry is missing. For views oriented with the suppressed or deleted reference, the view returns to its default position.

Aligning Views

To Align a View

Depending on the type of view, you can position the drawing view on the sheet by aligning the view with another view. For example, you may align a detail view with its parent to ensure that the detail view follows the parent view, if moved.

The view will remain aligned and move like a projection view until it is unaligned. If you need to unalign, clear **Align this view to other view** box in the **Drawing View** dialog box. You can realign a view by changing the alignment references, which are described below.

1. With a view selected and the **Drawing View** dialog box open, click the **Alignment** category. The **View alignment options** display in the dialog box.
2. To bring the selected view in line with another view, click **Align this view to other view**. You are prompted to select the view to align with.

   Note: When the selected view is a projection or auxiliary view, it can be unaligned from its projection parent by clearing **Align this view to other view** and click **Apply**. When the selected view is a projection or auxiliary view which has been unaligned from its parent, after clicking **Align this view to other view** you will not be able to select a view to align it to. The parent view's name will automatically appear in the collector.
3. Select the appropriate view on the drawing. The view name is displayed in the reference collector on the dialog.

   Note: When the selected view is a parent/child to other views, all of the related views highlight in blue and may also move when you modify the alignment.
4. Define how to limit the movement of the view you are aligning:
   - **Horizontal**—The view and the view it is aligned to will lie on the same horizontal line. If the view it is aligned to is moved, the view will move vertically so as to maintain horizontal alignment.
   - **Vertical**—The view and the view it is aligned to will lie on the same vertical line. If the view it is aligned to is moved, the view will move horizontally so as to maintain vertical alignment.

     Click **Apply** to see the view alignment. If the view alignment is not acceptable, repeat the steps above. If the view alignment is appropriate and no changes needs to be made, select the next appropriate category to define or click **Close**.
5. By default, the views are aligned according to their view origin. You can modify where the views align by defining alignment references. Click **Custom** and select a reference, such as an edge, on one of the views. The reference is displayed in the collector on the dialog. Click **Apply** to preview the alignment.

If necessary, you can customize the alignment on the other view. Click **Custom** for the view still aligned with its origin. Select an appropriate reference. Click **Apply** to preview the alignment.

Tip: Aligning Partial Views

If you already have a partial view in the drawing, you can create another aligned partial view as a copy of the original, but with a different boundary defined. You can also align a partial view to a detailed view.

You can align a partial view to another partial view along a selected straight entity This lets you create multiple partial views that selectively show model geometry in the same view orientation, as well as maintain relative placement between these partial views. This can be useful in showing connectors along routed cable assemblies.

The initial partial view—the parent view—determines various properties of successive aligned partial views, such as the following:

- The aligned view uses the same view attributes (for example, attributes specifying whether the view is scaled or exploded) as its parent view.
- You cannot change the scale of the aligned partial view independently, but it does change when you modify the scale of the parent partial view.

The aligned partial view is a dependent view, its location depends on the view to which it was aligned. When you move the parent view, the location of the aligned view also adjusts. You can move the aligned view only in the direction of alignment, closer or farther away from the parent. If you erase the parent view or move it to another sheet, you can move the aligned view without restrictions until you resume the parent view or switch it back.

Drawing View Size

About Setting the Size of a Drawing View

You can set the size of a drawing view so that when you change dimensions of the model, the size of the drawing view stays constant relative to a given model dimension. To do this, specify the model parameter `drawing_scale_factor`, used in Drawing mode as a view scale factor. This parameter is a scale factor that the drawing scale multiplies to determine the actual size of the entire drawing view.

Note: This does not change the value of "d#" in the drawing or the overall drawing scale, only the spatial dimension of the displayed entities.

The following table shows how the system calculates the size of the drawing view using the drawing scale and the `drawing_scale_factor` model parameter.

Calculating the Size of a Drawing View

value of d#	**length para- meter**	**drawing_scale _factor**	**drawing scale**	**Length in the View**	**Comments**
5	5	1	2	10	*Changing drawing scale*. Drawing scale changed to 2; the edge length in the view becomes 10.
5	5	1	1	5	***Initial conditions.*** Model edge has a corresponding dimension d# = 5. If the drawing scale is 1 and the edge length is 5, the "drawing_scale_factor" is set to 1.
10	5	.5	1	5	***Changing dimension value.*** Value of d# changed to 10 in Part or Assembly mode, so "drawing_scale _factor" becomes .5. Makes the calculated length in the view half of its dimension value (it remains 5 (unchanged)).
5	10	2	1	10	***Changing value of "length" parameter.*** Value of "length" parameter changed in the relation to 10, so "drawing_scale_ factor" becomes 2. Multiplies the length of the edges in the view by 2.
5	10	2	2	20	***Changing value of "length" parameter at***

value of d#	length para- meter	drawing_scale _factor	drawing scale	Length in the View	Comments
					a given scale. The edge with the dimension value of 5 appears in the drawing with a scale of 2. Changing "length" parameter to 10 changes "drawing_scale_ factor" to 2, so the calculated length of the edge in the drawing is 20.

To Set the Size of the Drawing View

1. Determine the edge that you want to use as the basis for setting the `drawing_scale_factor`. The system uses the dimension value and the length of this edge in the drawing to calculate it. Whenever you change either one of these variables ("length" or d#), it calculates the length of model edges according to the following equation:

   `length in the view = (dimension value) x(drawing_scale_factor) x (drawing_scale)`

2. Add the following relation to the part or assembly:

   `drawing_scale_factor = length / dimension`

   where length is the parameter corresponding to the current drawing measurement of the above dimension d#, and dimension is a model dimension, d#.

3. Regenerate the model.

Moving and Deleting Views

About Moving Views

To prevent drawing views from being accidentally moved, they are locked in place by default. You move views either horizontally or vertically by selecting and dragging the view. To move the view freely on the drawing, you must unlock it. Select and right-click the view, and click **Lock View Movement**. All views in the drawing, including the selected view, are unlocked.

When views are locked, you can move them by editing the view's exact X-Y location with the **Edit** > **Move Special** command.

- If you accidentally move a view, you can press ESC while the move is in progress to snap the view back to its original position.
- If you move a parent view from which other views were projected, the projected views also move to maintain view alignment.

  This alignment and parent/child relationship between projected views remains intact even as the model changes. You can move general and detailed views to any new location because they are not projections of other views.

To Move a View

To move views by selecting and dragging them, uncheck the **Lock View Movement** check box in the **Tools > Environment** dialog box.

To move a view:

1. Select the view. The view outline is highlighted.
2. Drag the view to a new location by the corner drag handles or by the centerpoint. The cursor changes to a four-sided cross when drag mode is active.

To move a view using exact X and y coordinates:

1. Select the view. The view outline is highlighted.
2. Click **Edit** > **Move Special**. You are prompted to select a point on the selected item.
3. Click a point on the selected item that you want to use as an origin for the move. The **Move Special** dialog box opens. Use the icons to choose a method to relocate the point you selected: either to a point you click on the sheet, or to specific X and Y coordinates you enter in the dialog box.
4. Click **OK** to finish.

Note: If you accidentally move a view, you can press `ESC` while the move is in progress to snap the view back to its original position.

To Switch Views to Another Sheet

1. Select the view to move to another sheet.
2. Click **Edit > Move Item to Sheet**. You are prompted for a target sheet number.
3. Enter the number and press **Enter**. The view is moved to the same coordinates on the target sheet.

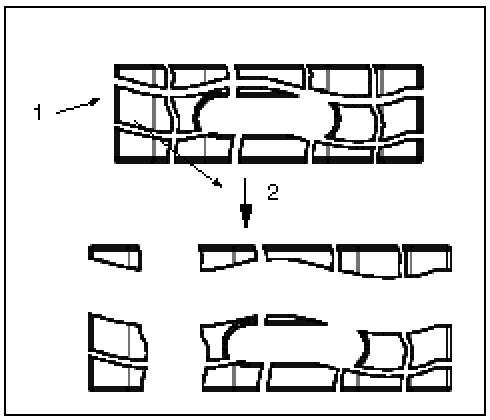
To Delete a View

1. Select the view to delete. The view highlights.
2. Right-click and click **Delete** from the shortcut menu or click **Edit > Delete**. The view is removed.

Note: If the view you selected has a projection child view, it will also be deleted along with it. You can use command undo to undo the deletion.

Tip: Moving Broken Views

When you move a broken view, for any subview (or portion of the view) that you select to move, all subviews to its right and below it move the same distance. To move the entire broken view to a different location on the drawing, select the upper-left subview. This moves the entire view without altering the gaps between the subviews. Selecting any other subview moves all subviews below it and to the right of it the same distance.



1. Select this subview to move the view as a whole.
2. Move vector.

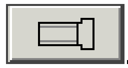
Cosmetic Feature Display

To Display Cosmetic Features in a Drawing

Pro/ENGINEER displays cosmetic features and cosmetic threads automatically in all views unless their sketching plane is perpendicular to the screen. You can also show or erase cosmetic features using the **Show/Erase** dialog box. You can show or erase sketched cosmetic features, weld features, and cosmetic threads in a drawing view without having to show or erase datum planes as well.

To display cosmetic features individually, do the following:

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.

3. Under **Type**, click .
4. Use the following options in the **Options** tab to filter the cosmetic features you want to show in the drawing:
   - **Erased**—Show previously erased items.
   - **Never Shown**—Show items that are yet to be displayed in the drawing.
5. Under **Show By**, use the following options to define where you want to display the cosmetic feature:
   - **Feature**—Show the cosmetic feature for a particular feature on the drawing.
   - **Part**—Show the cosmetic feature for a particular part on the drawing.
   - **View**—Show all the cosmetic features for the features and parts within a particular drawing view.
   - **Feature and View**—Show the cosmetic feature for a feature that appears in several views in one selected view. For example, if you have a feature that appears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to display it.
   - **Part and View**—Show the cosmetic feature for a part that appears in several views in one selected view.
   - **Show All**—Show all the cosmetic features within the drawing.
6. Select the appropriate feature, part, or view to display the cosmetic features. You may need to repaint the drawing for the cosmetic features to be displayed. When the cosmetic features are displayed the **Preview** tab becomes available.
7. Of the cosmetic features previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:
   - **Sel to Keep**—Select the individual cosmetic features to show in the drawing. Any cosmetic feature not selected will be erased.
   - **Sel to Remove**—Select cosmetic features to remove from the drawing. Any cosmetic feature not selected will remain in the drawing.
   - **Accept All**—Keep all the previewed cosmetic features.
   - **Erase All**—Erase all the previewed cosmetic features.

   While you are showing the cosmetic features, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and **Resume Show and Erase**.

If you need to select multiple cosmetic features to keep or remove, you can do this by either holding down the CTRL key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.

8. When you are finished with the preview, click OK in the Select dialog box. The cosmetic features are displayed in the drawing.
9. Click Close to close the Show/Erase dialog box.

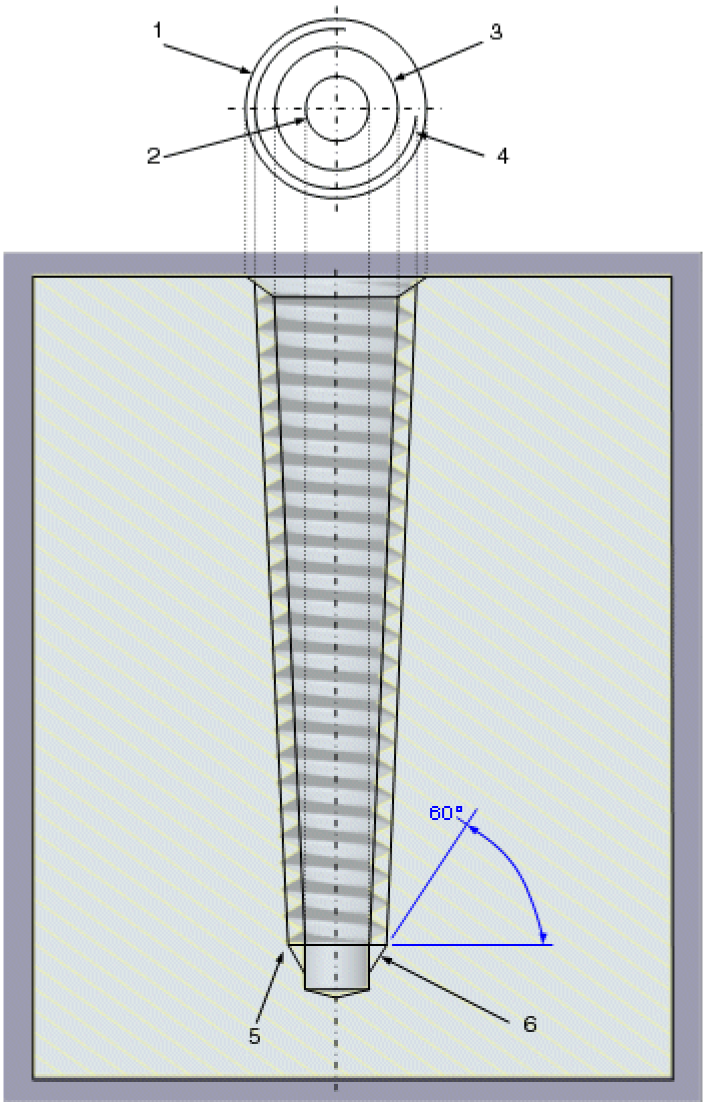
Note:

- You can control the display of threads in a drawing using the `hlr_for_threads` drawing setup file option.
- To display cometic features in the drawing, clear the **With Preview** check box under the **Preview** tab in the **Show/Erase** dialog box. In Pro/ENGINEER, you cannot preview the display of cosmetic features.
- The erasure or display of cosmetic features in a detailed view is always the same as that of the parent; you cannot modify it individually. Click **Note** in the **Type** box of the **Show/Erase** dialog box to erase pattern notes. For example, the notes that display the number of members of a pattern (2 HOLES).

About Displaying Tapered Threaded Holes in a Drawing

You can display tapered threaded holes in a drawing in the standard and cross-section views according to the JIS, ANSI, and ISO standards. To display such holes, use the `thread_standards` drawing setup file option in conjunction with the `hlr_for_threads` configuration option.

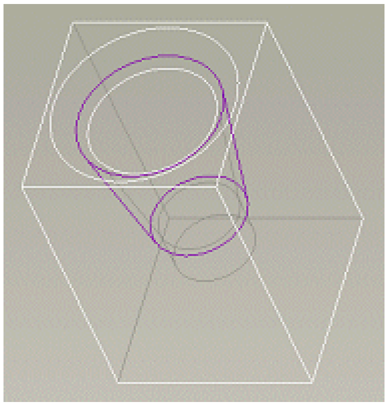
The following figure is a simplified representation of a tapered threaded hole displayed as concentric circles overlaid on a detailed model:



1. Outer diameter of the countersunk hole
2. Innermost minor (drilled hole) diameter
3. Outermost minor diameter (hard geometry)
4. Major diameter of the thread at the outermost intersection surface
5. Arc for the thread from the major diameter at the base
6. Incomplete threads

Example: Displaying Tapered Threaded Holes in a Drawing

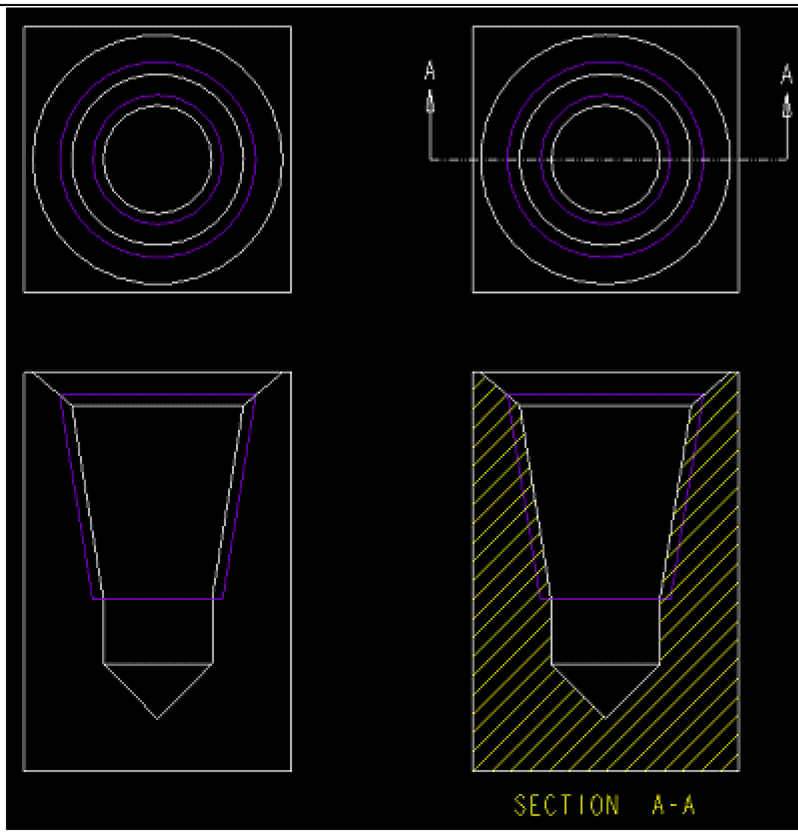
Consider the following model with a countersunk threaded and tapered hole.

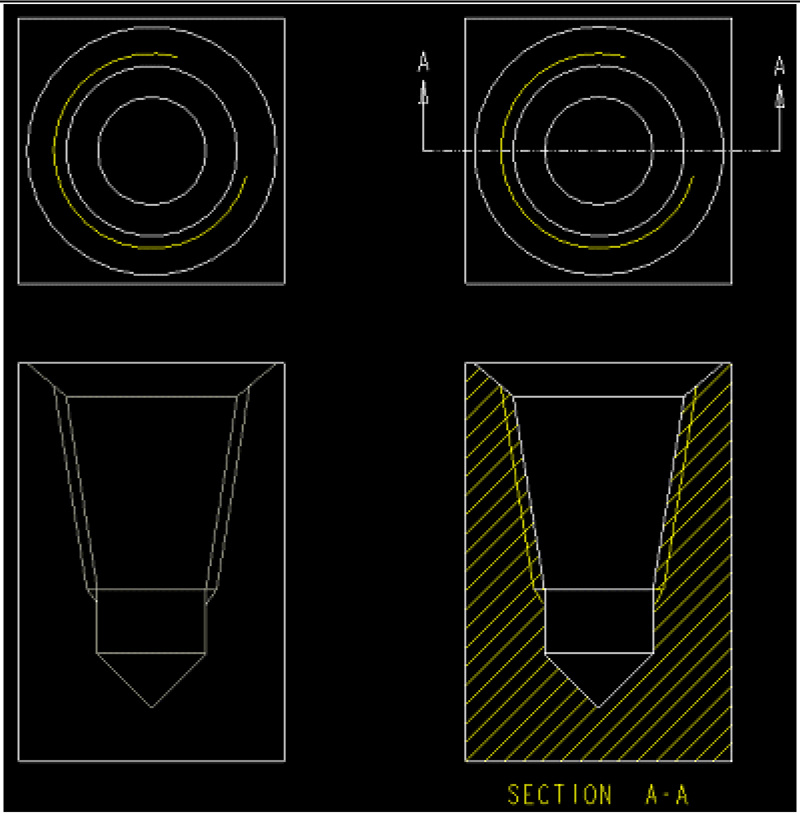


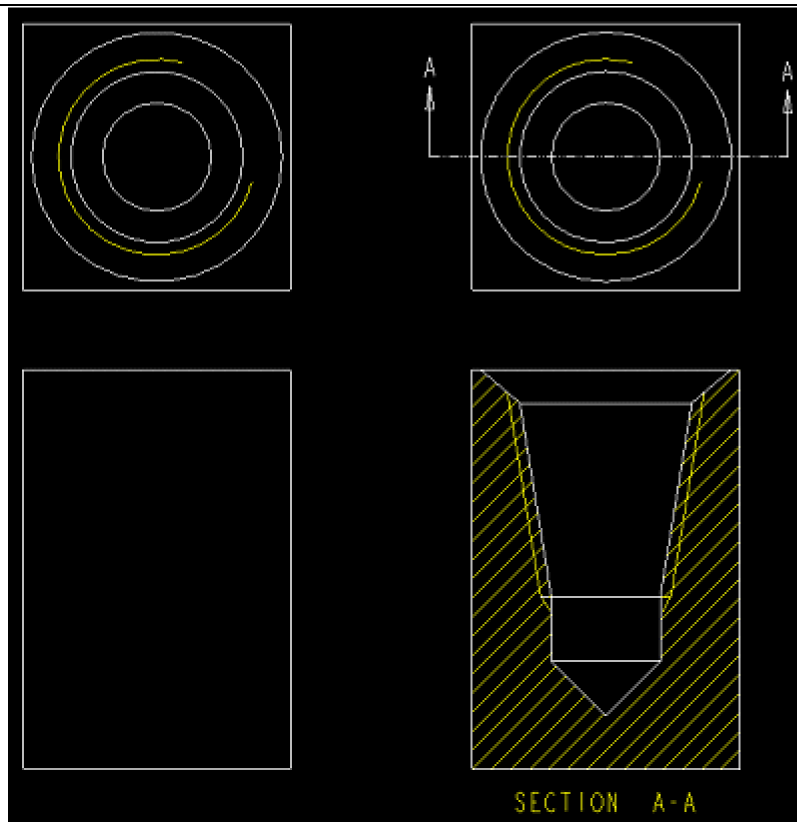
You can represent the hole in the JIS, ANSI, and ISO standards as discussed in the following sections.

JIS Standard

To display a threaded hole according to the JIS standard, set the `thread_standard` drawing setup file option to `std_jis` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of the hole in the Wireframe, Hidden, and No Hidden display states:

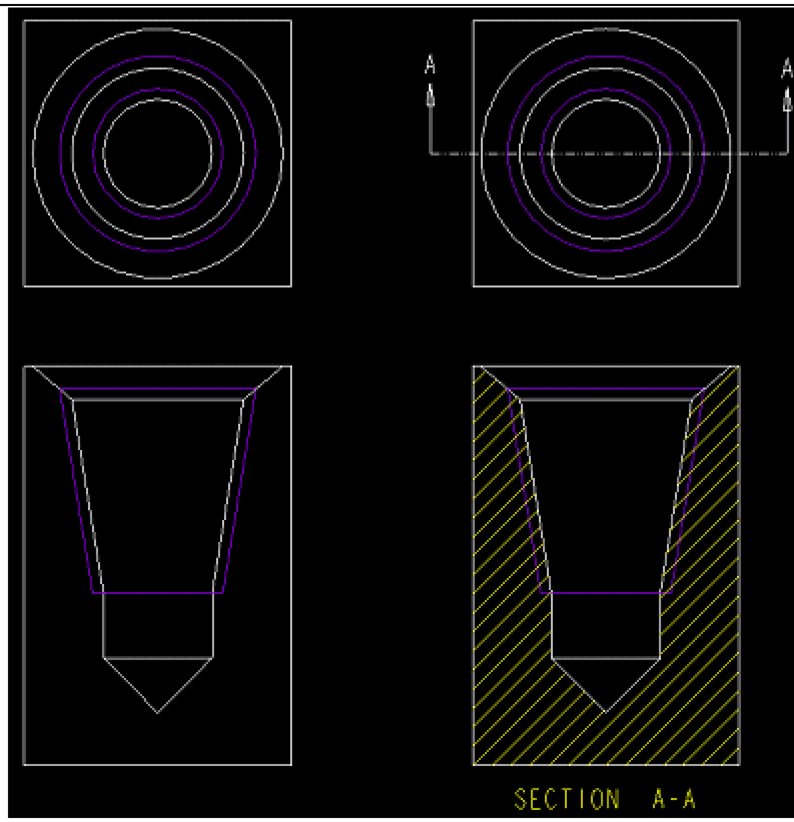
Display State	Display
In the Wireframe display state, the cosmetic thread quilt is highlighted in purple.	

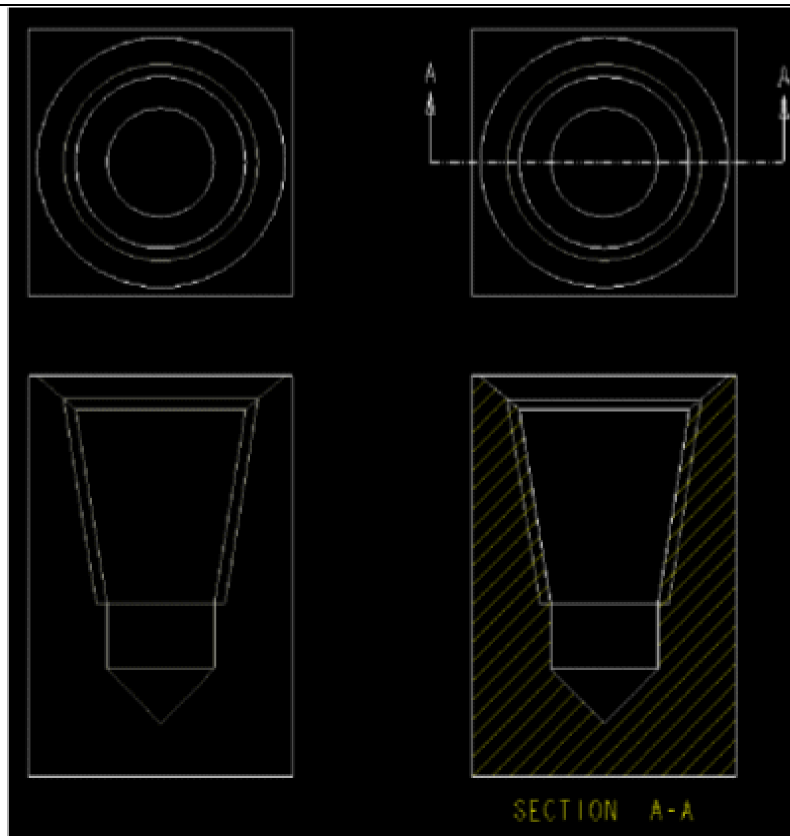
Display State	Display
In the Hidden display state, the cosmetic thread quilt is not displayed. The side and the section views display incomplete threads. The top view displays a 270 degree arc at the outermost major diameter. Solid arcs are displayed for the outermost and innermost minor diameters. The display of the major diameter is blanked at the base.	

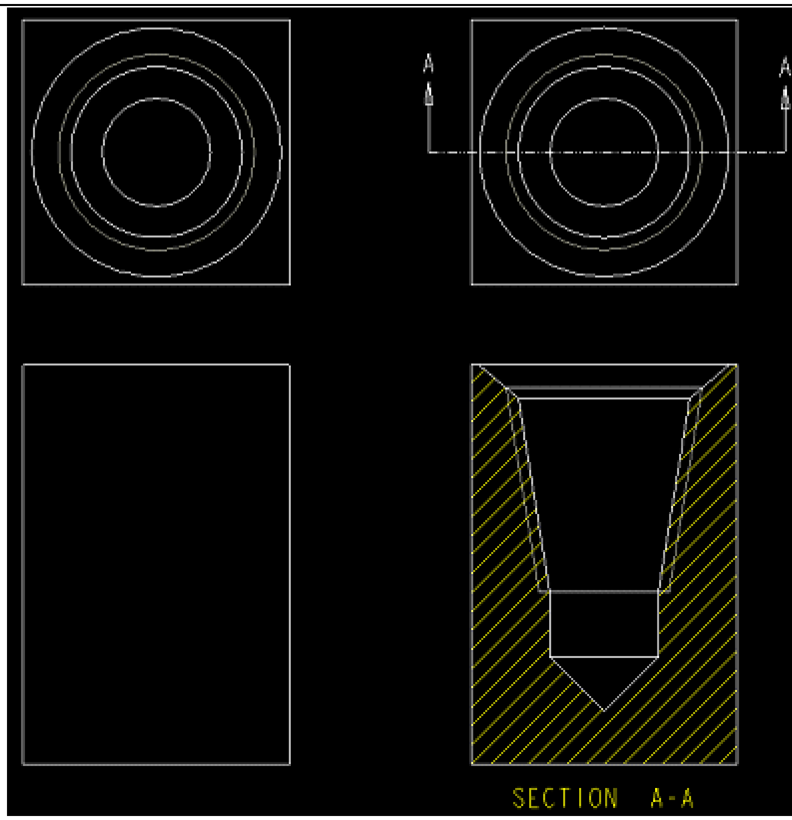
Display State	Display
In the No Hidden display state, the cosmetic thread quilt is not displayed. The side view does not display any holes. The top and the section views are the same as the hidden line view display.	

ANSI Standard

To display a threaded hole according to the ANSI standard, set the `thread_standard` drawing setup file option to `std_ansi` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of the hole in the Wireframe, Hidden, and No Hidden display states.

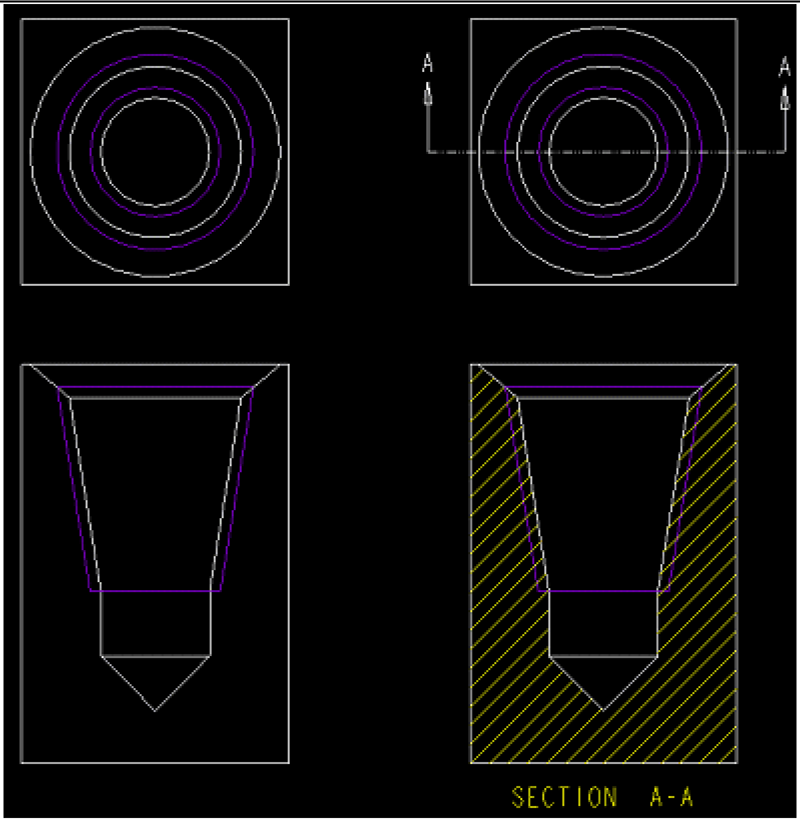
Display State	Display
In the Wireframe display state, the cosmetic thread quilt is highlighted in purple, with the major and minor diameters displayed at the base due to the taper.	

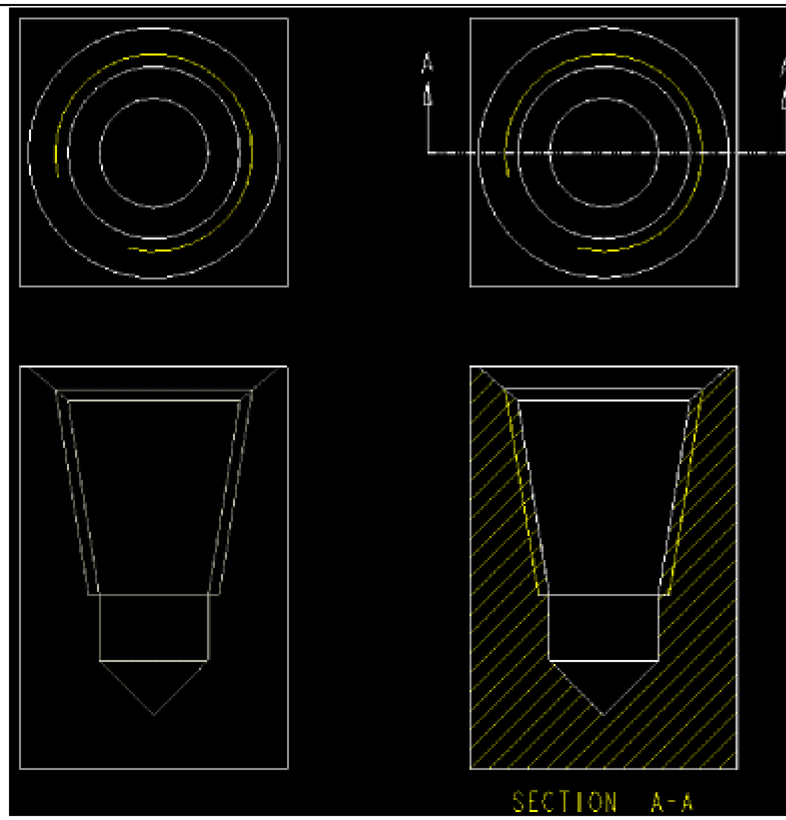
Display State	Display
In the Hidden display state, the cosmetic thread quilt is not displayed. The top view displays solid arcs against the outermost and innermost minor diameters. A gray solid arc is displayed only for the outermost major diameter. The innermost major diameter is blanked.	

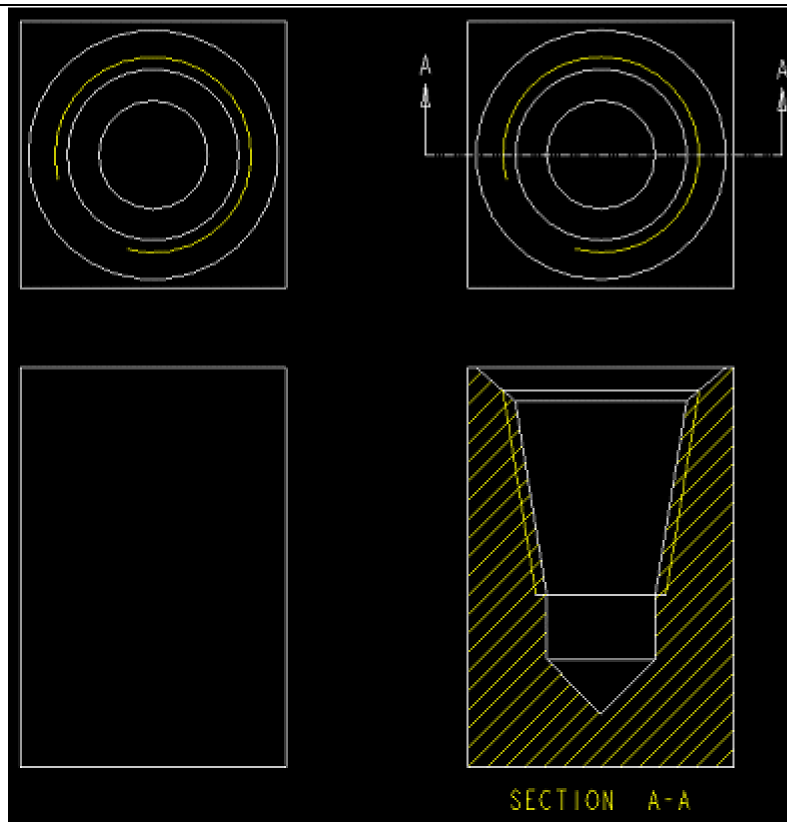
Display State	Display
In the No Hidden display state, the cosmetic thread quilt is not displayed. The side view does not display any holes. The top and the section views are the same as the display of the hidden line view.	

ISO Standard

To display a threaded hole according to the ISO standard, set the `thread_standard` drawing setup file option to `std_iso` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of the hole in the Wireframe, Hidden, and No Hidden display states.

<table>
<tr><th>Display State</th><th>Display</th></tr>
<tr><td>In the Wireframe display state, the cosmetic thread quilt is highlighted in purple, with the major and minor diameters displayed at the base due to the taper.</td><td>
</td></tr>
</table>

Display State	Display
In the Hidden display state, the cosmetic thread quilt is not displayed. The top view displays solid arcs for the outermost and innermost minor diameters. A gray solid arc is displayed only for the outermost major diameter. The innermost major diameter is blanked.	

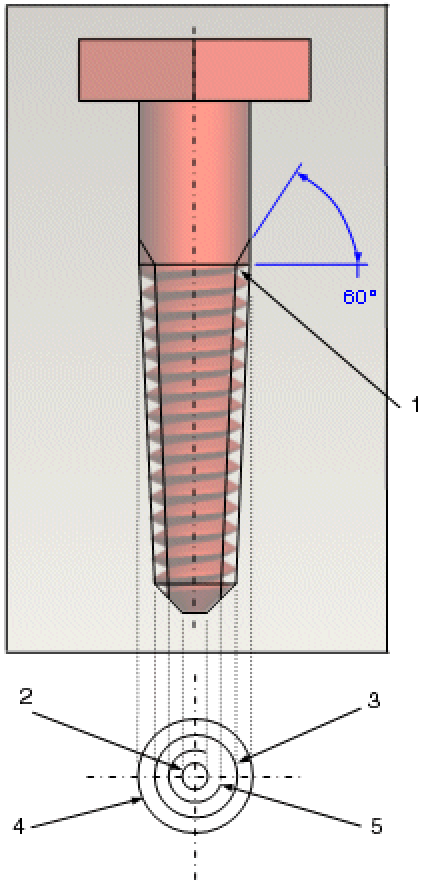
Display State	Display
In the No Hidden display state, the cosmetic thread quilt is not displayed. The side view does not display any holes. The top and the section views are identical to the display of the hidden line view.	

Note: In all the standards, if a tapered thread does not extend to a surface, point, or an edge, the major and the minor diameter arcs are blanked for the additional geometry.

About Displaying Tapered Threaded Shafts in a Drawing

You can display tapered threaded shafts in a drawing in the standard and cross-section views according to the JIS, ANSI, and ISO standards. To display such shafts, use the `thread_standards` drawing setup file option in conjunction with the `hlr_for_threads` configuration option.

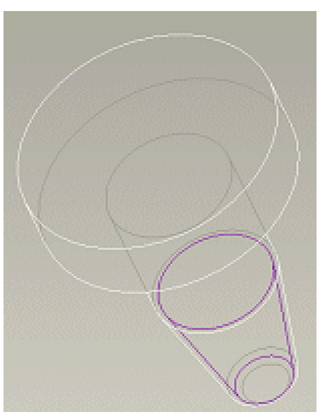
The following figure is a simplified representation of a tapered threaded shaft displayed as concentric circles overlaid on a detailed model:



1. Arc of one of the threads from the minor diameter at the base that is hidden in the top view
2. Outermost edge of the end chamfer
3. Outermost major diameter
4. Innermost major diameter
5. Outermost minor diameter of the thread at the outermost intersection surface

Example: Displaying Tapered Threaded Shafts in a Drawing

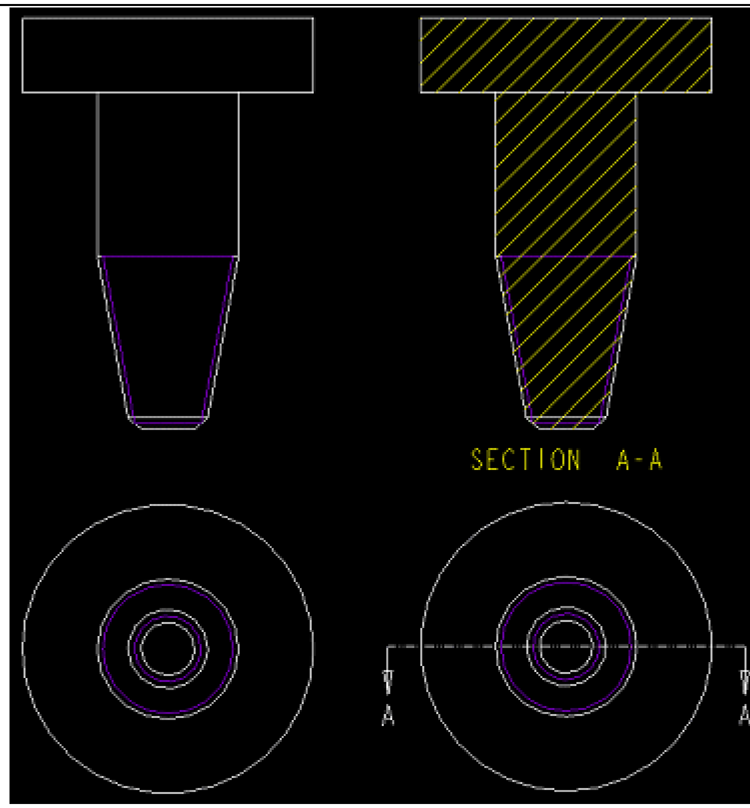
Consider the following model with a threaded tapered shaft and a chamfer.

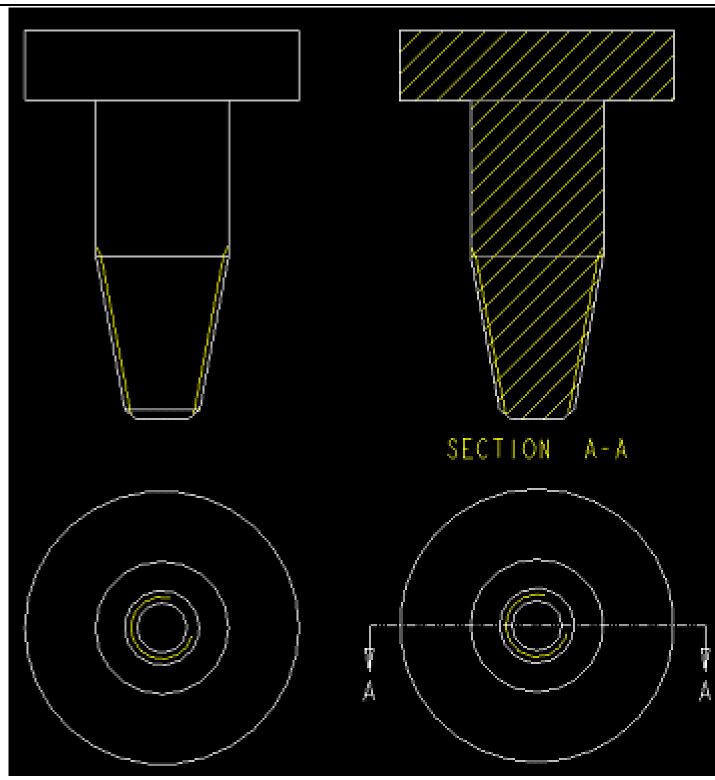
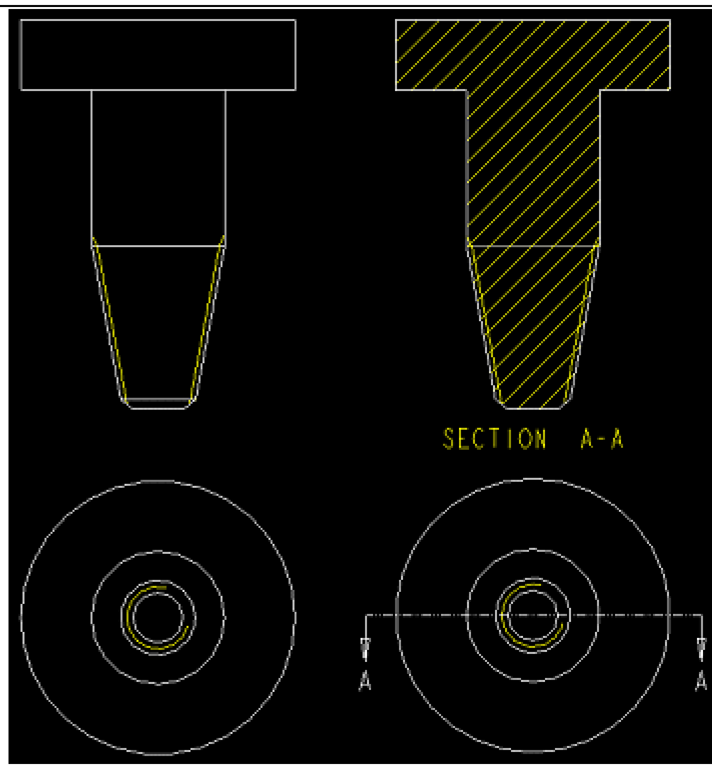


You can represent such a shaft in the JIS, ANSI, and ISO standards as discussed in the following sections.

JIS Standard

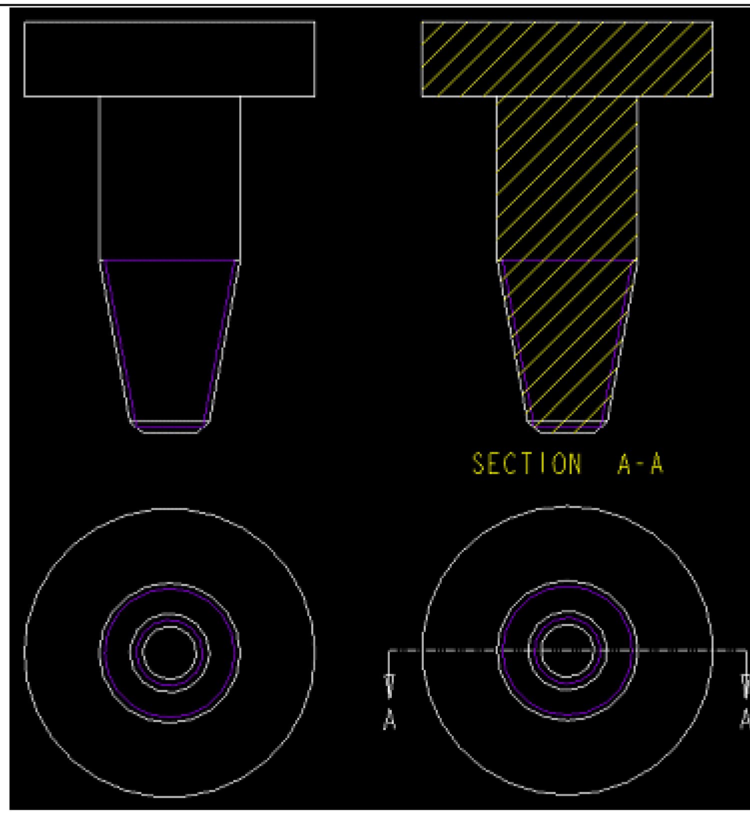
To display a tapered threaded shaft according to the JIS standard, set the `thread_standard` drawing setup file option to `std_jis` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of the shaft in the Wireframe, Hidden, and No Hidden display states:

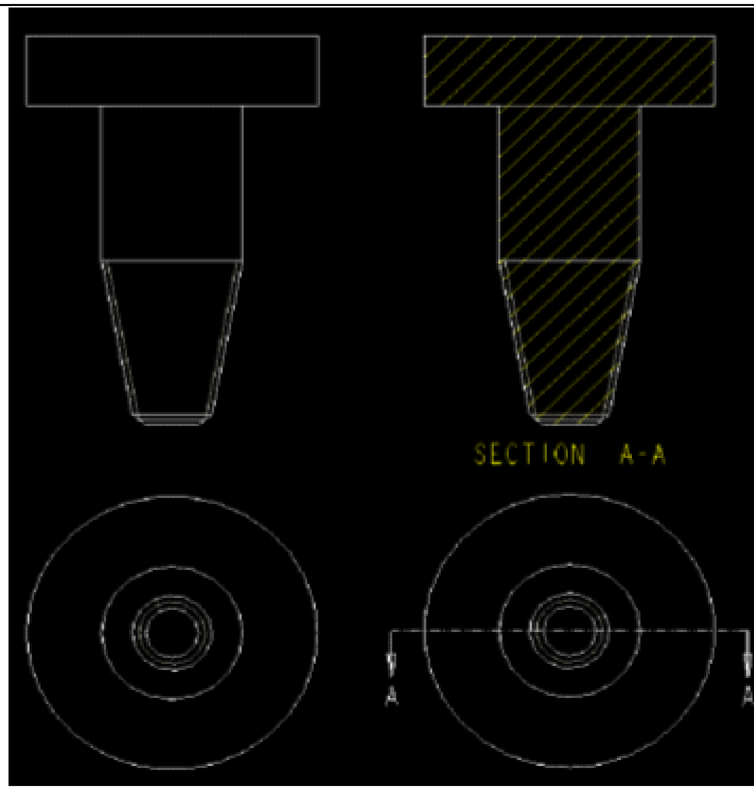
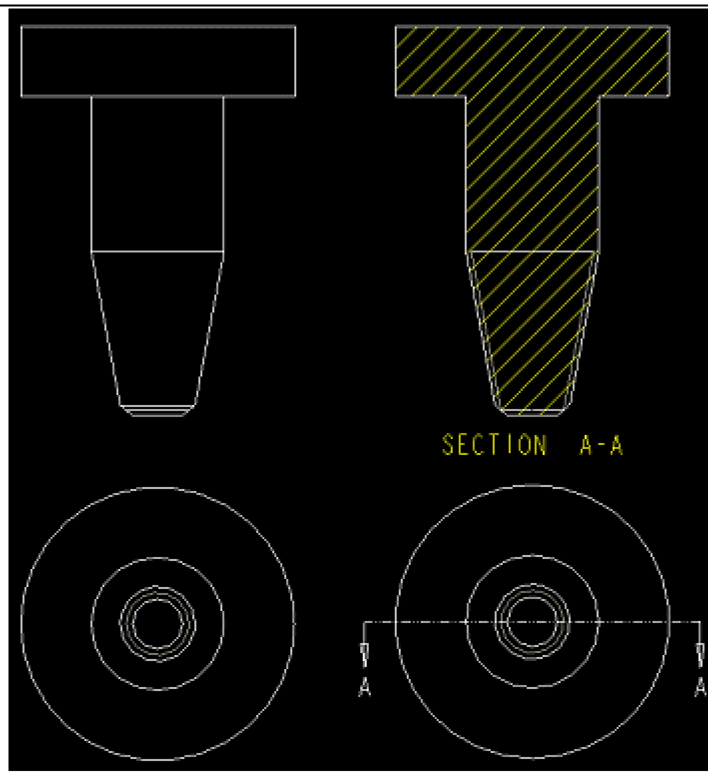
Display State	Display
In the Wireframe display state, the cosmetic thread quilt is highlighted in purple, with the major and minor diameters displayed at the base due to the taper.	

Display State	Display
In the Hidden display state, the cosmetic thread quilt is not displayed. The side and the section views display incomplete threads. The section view displays hidden lines for the top of the quilt and for the back of the bolt head. The top view displays a 270 degree arc at the minor diameter. Solid arcs are displayed for the major diameter and the chamfer. The display of the minor diameter is blanked at the base.	
In the No Hidden display state, the cosmetic thread quilt is not displayed. The side view is identical to the left view. The section view does not display the back of the bolt head. The back of the tapered geometry edge is hidden.	

ANSI Standard

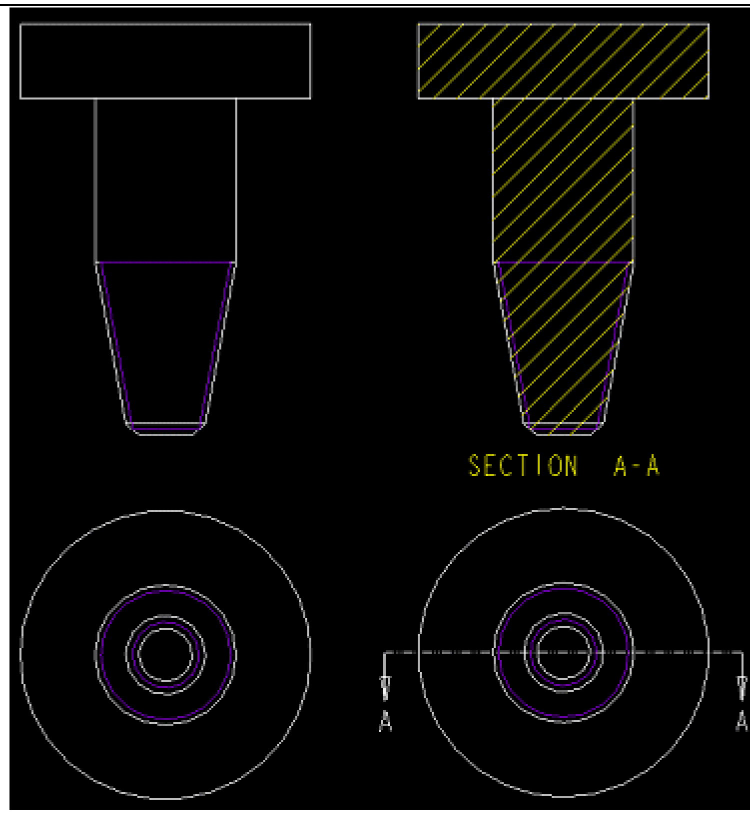
To display a tapered threaded shaft according to the ANSI standard, set the `thread_standard` drawing setup file option to `std_ansi` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of the shaft in the Wireframe, Hidden, and No Hidden display states.

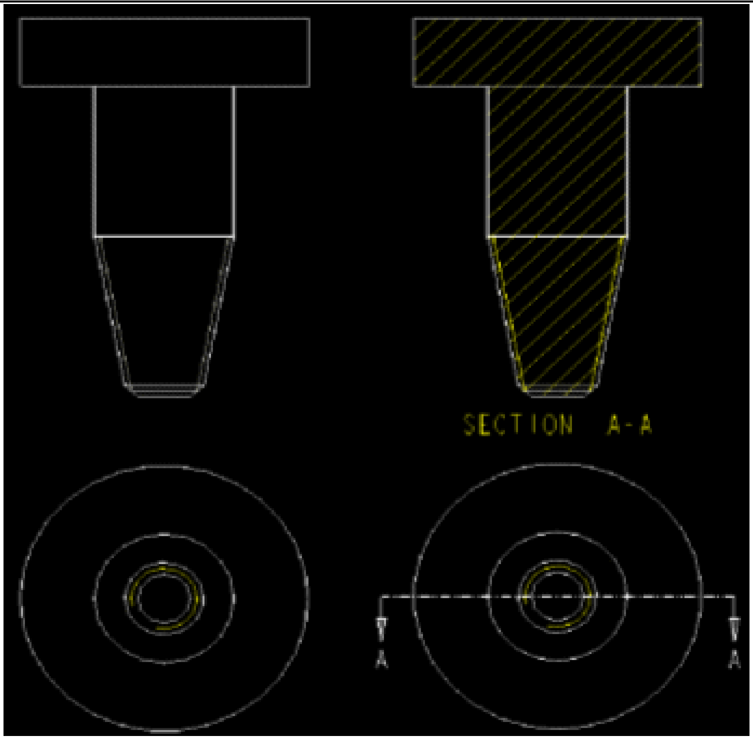
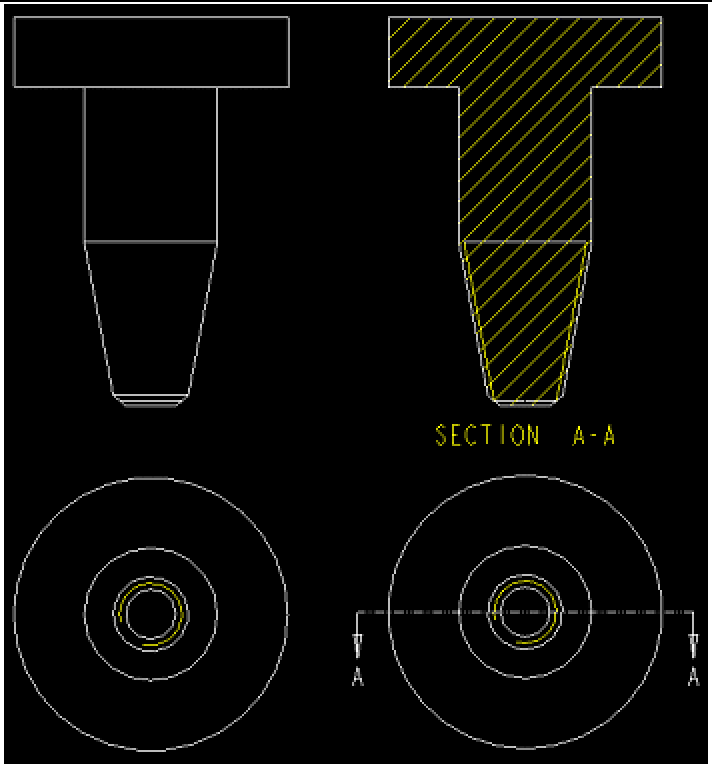
Display State	Display
In the Wireframe display state, the cosmetic thread quilt is highlighted in purple, with the major and minor diameters displayed at the base due to the taper.	

Display State	Display
In the Hidden display state, the cosmetic thread quilt is not displayed. The section view displays a hidden line for the back of the bolt head. The top view displays solid arcs for the outermost and innermost major diameters and the chamfer. A hidden arc is displayed for the outermost minor diameter. The innermost minor diameter is blanked.	
In the No Hidden display state, the cosmetic thread quilt is not displayed. The views are identical to that of the hidden display state except that the section view does not display the back of the bolt head, and the threads in the side view are not displayed.	

ISO Standard

To display a tapered threaded shaft according to the ISO standard, set the `thread_standard` drawing setup file option to `std_iso` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of the shaft in the Wireframe, Hidden, and No Hidden display states.

Display State	Display
In the Wireframe display state, the cosmetic thread quilt is highlighted in purple, with the major and minor diameters displayed at the base due to the taper.	

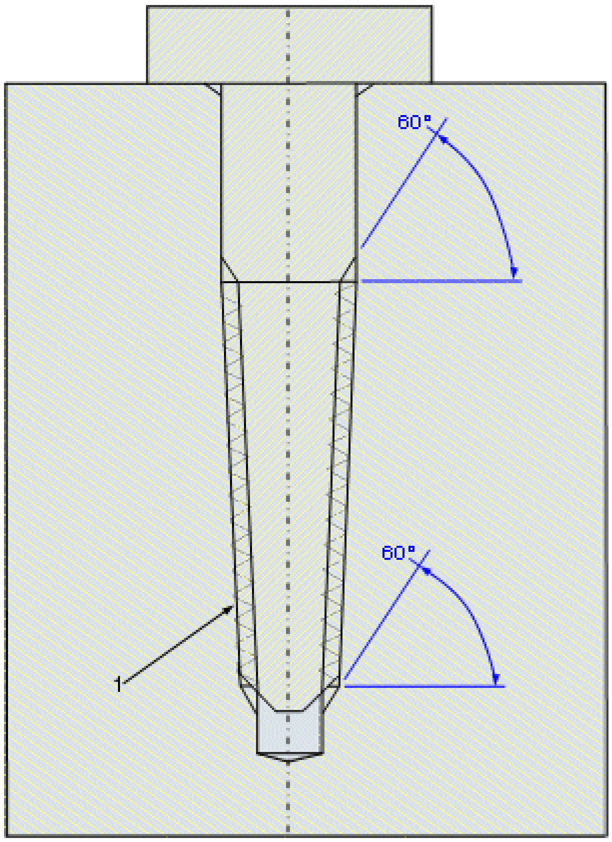
Display State	**Display**
In the Hidden display state, the cosmetic thread quilt is not displayed. The section view displays a hidden line for the back of the bolt head and solid lines for the upper and lower thread extents. The top view displays a 270 degree arc for the outermost diameter highlighted in yellow. The innermost minor diameter is blanked. Solid arcs are displayed for the major diameter and the leading chamfer.	
In the No Hidden display state, the cosmetic thread quilt is not displayed. The side view does not display the threaded surface. The views are identical to that of the hidden display state except that the section view does not display the back of the bolt head, and the threads in the side view are not displayed.	

Note: In all the standards, if a tapered threaded shaft does not extend to a surface, point, or an edge, the major and the minor diameter arcs are blanked for the additional geometry.

About Displaying Threaded Components of an Assembly in a Drawing

You can display tapered threaded shafts and holes of an assembly in a drawing in the standard and cross-section views according to the JIS, ANSI, and ISO standards. To display such shafts and holes, use the `thread_standards` drawing setup file option in conjunction with the `hlr_for_threads` configuration option.

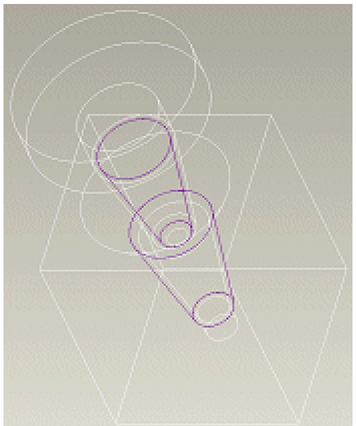
The following figure is a simplified representation of an assembly with a tapered threaded shaft and a hole overlaid on a detailed model:



1. If aligned, the major and minor bolt diameters override the hole diameter

Example: Displaying Threaded Shafts and Holes of an Assembly in a Drawing

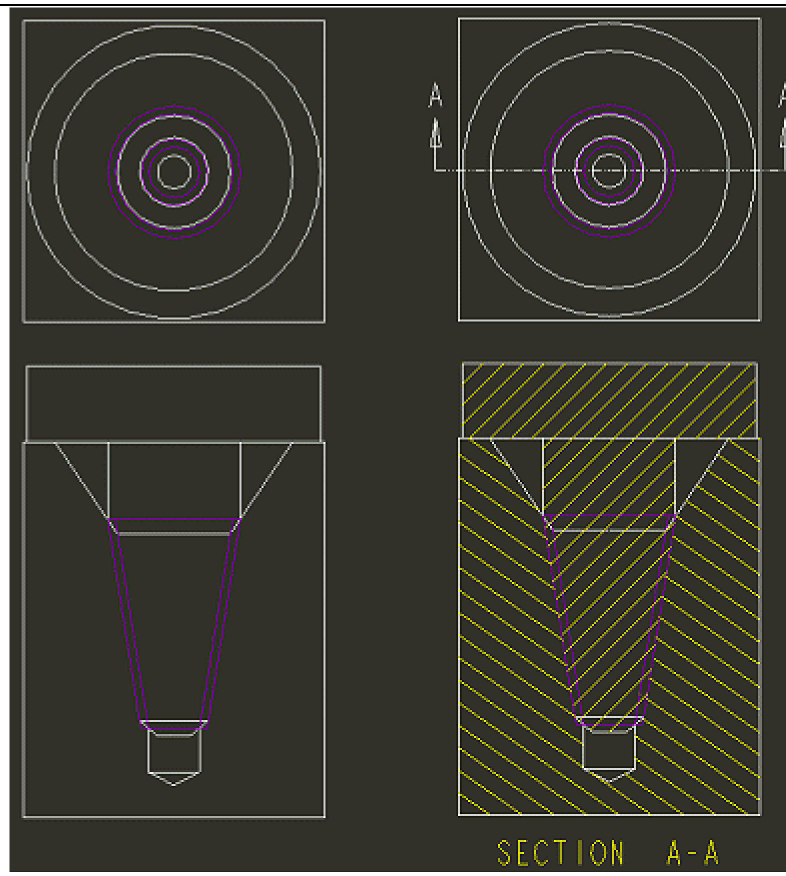
Consider the following model where a threaded tapered bolt with a leading chamfer is inserted into a countersunk, threaded, and a straight hole.

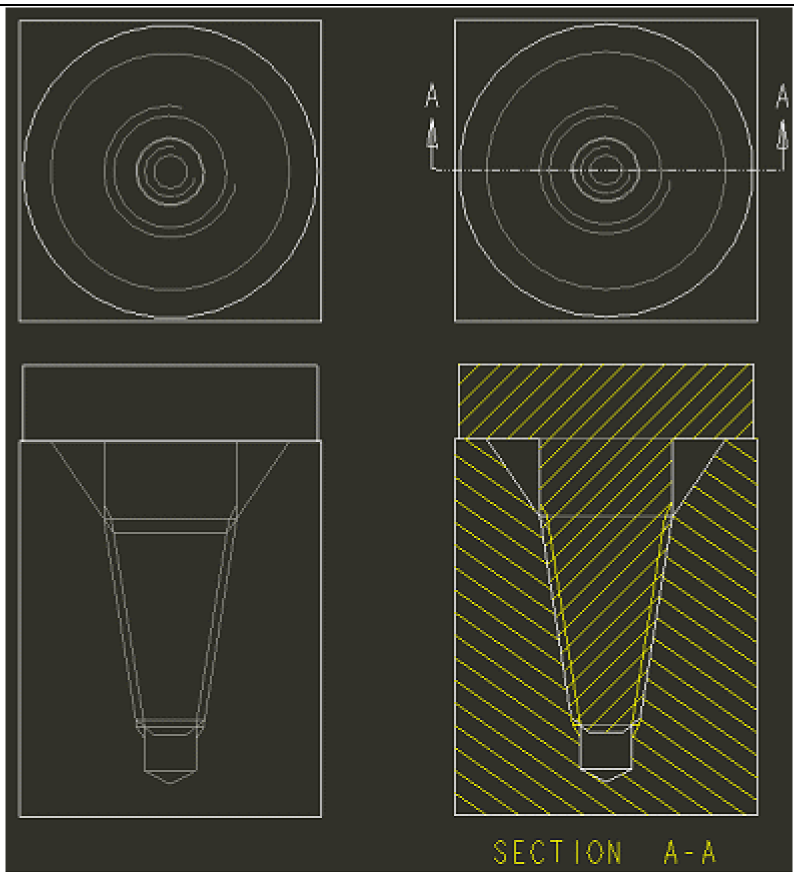


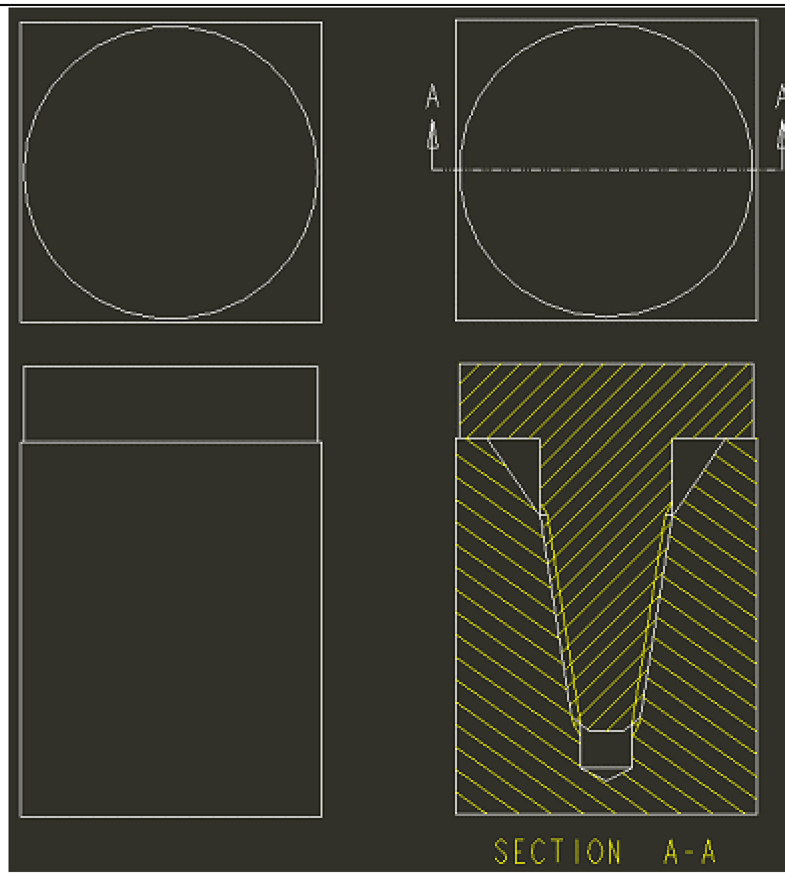
You can represent such a bolt and hole in the JIS, ANSI, and ISO standards as discussed in the following sections.

JIS Standard

To display a tapered threaded bolt and a hole of an assembly according to the JIS standard, set the `thread_standard` drawing setup file option to `std_jis` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of such a bolt and hole in the Wireframe, Hidden, and No Hidden display states:

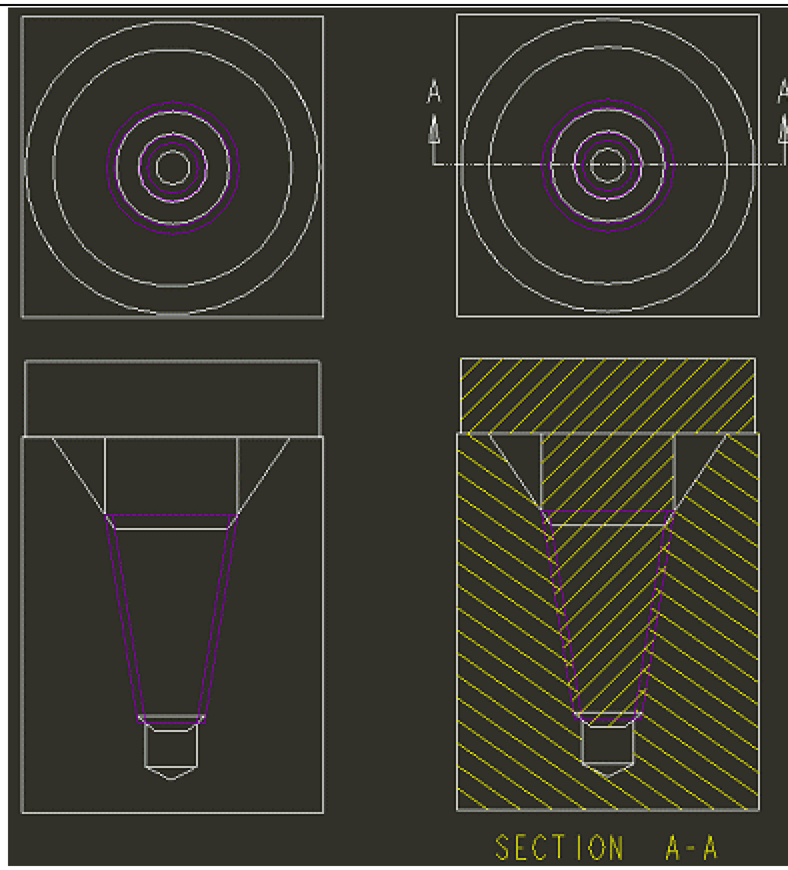
Display State	**Display**
In the Wireframe display state, the cosmetic thread quilt is highlighted in purple. The crosshatch of the shaft overrides that of the hole.	

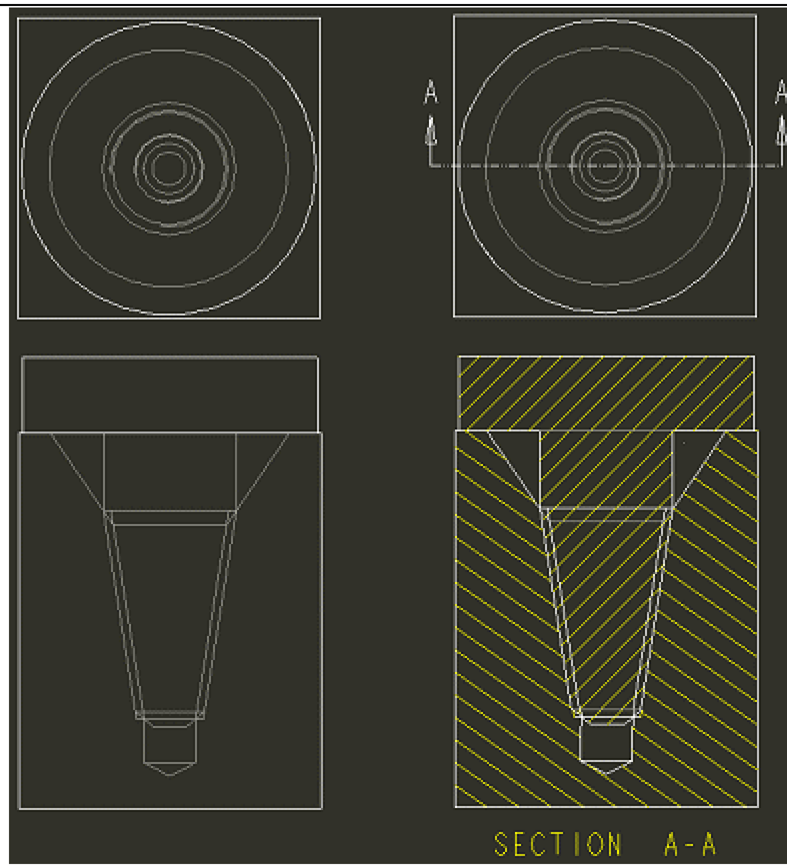
Display State	Display
In the Hidden display state, the cosmetic thread quilt is not displayed. The side and the section views display incomplete threads. 270 degree arcs are created but hidden.	

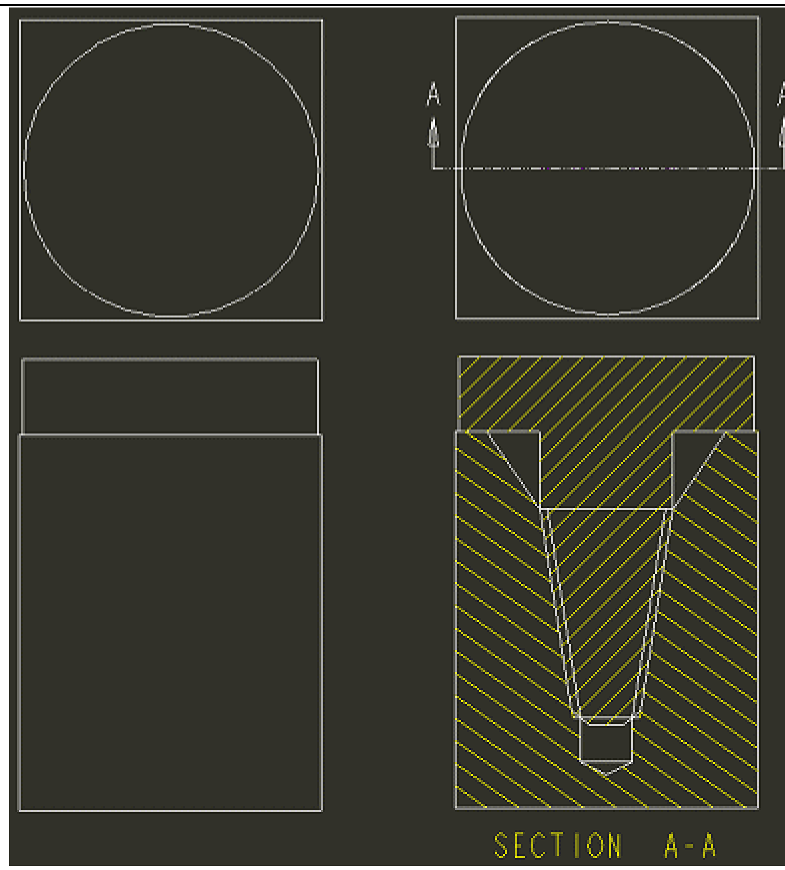
Display State	Display
In the No Hidden display state, the cosmetic thread quilt is not displayed. The side view does not display holes or shafts. The top and the section views are the same as the hidden line view. Hidden lines are not displayed.	

ANSI Standard

To display a tapered threaded bolt and a hole of an assembly according to the ANSI standard, set the `thread_standard` drawing setup file option to `std_ansi` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of such a bolt and hole in the Wireframe, Hidden, and No Hidden display states.

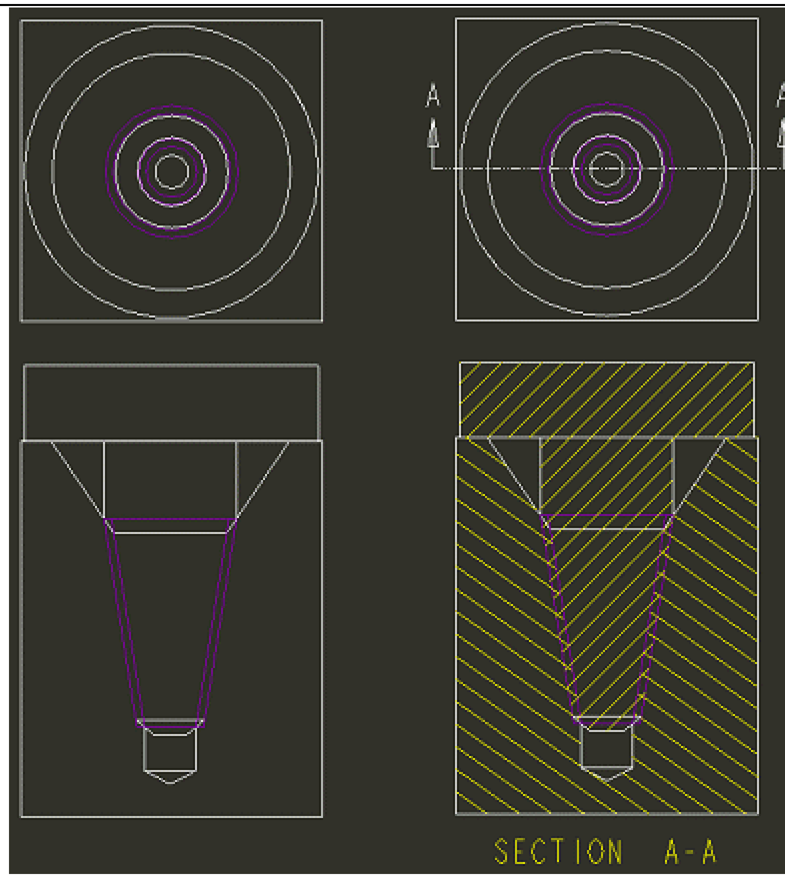
Display State	**Display**
In the Wireframe display state, the cosmetic thread quilt is highlighted in purple. Crosshatches of the bolt and the hole overlap.	

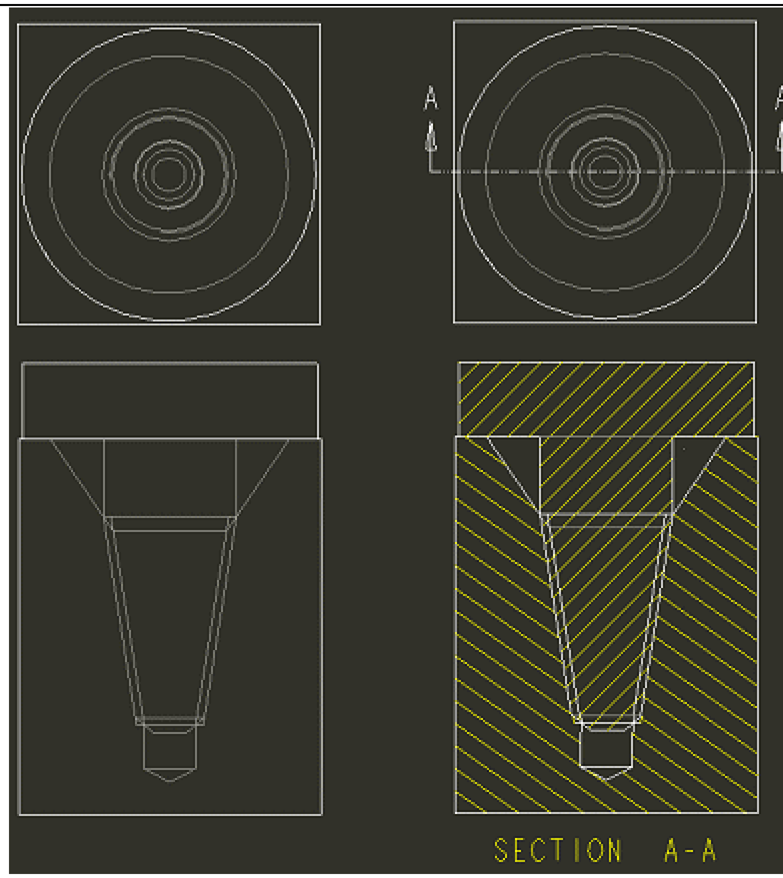
Display State	**Display**
In the Hidden display state, the cosmetic thread quilt is not displayed. Solid gray arcs are displayed for all the threaded major and minor diameters.	

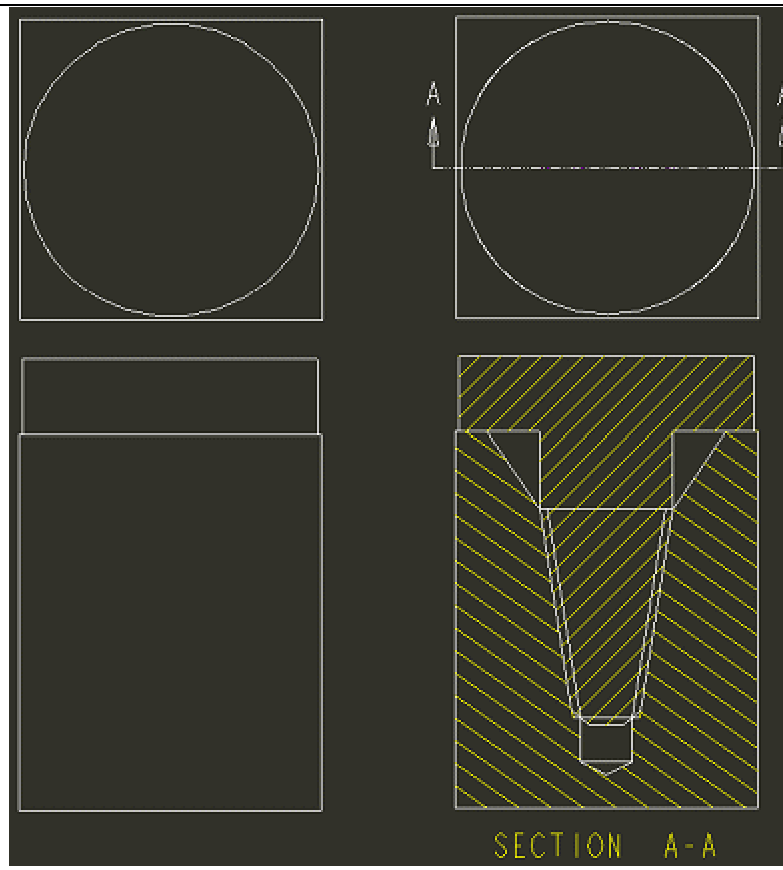
Display State	Display
In the No Hidden display state, the cosmetic thread quilt is not displayed. The side view does not display holes or shafts. The top and the section views are the same as the hidden line view. Hidden lines are not displayed.	

ISO Standard

To display a tapered threaded bolt and a hole of an assembly according to the ISO standard, set the `thread_standard` drawing setup file option to `std_iso` and the `hlr_for_threads` configuration option to `yes`. The following table illustrates the display of such a bolt and hole in the Wireframe, Hidden, and No Hidden display states.

Display State	Display
In the Wireframe display state, the cosmetic thread quilt is highlighted in purple. Crosshatches of the bolt and the hole overlap.	

Display State	Display
In the Hidden display state, the cosmetic thread quilts are not displayed. 270 degree arcs are created but hidden.	

Display State	Display
In the No Hidden display state, the cosmetic thread quilt is not displayed. The side view does not display holes or shafts. The top and the section views are identical to the hidden line view. Hidden lines are not displayed.	

Dimensioning and Detailing Your Models

About Dimensioning and Detailing Your Models

The purpose of creating a drawing is to enable the model to be manufactured. There are several ways to appropriately dimension and detail your models in your drawings. For instance, you can add dimensions and details by:

- **Showing Driving Dimensions**—the information stored within model itself. By default, all dimensions and stored model information are invisible (or erased) when you import the model or assembly into the 2D drawing. These dimensions are actively linked to the 3D model, so you can directly edit the 3D model through the dimension in the drawing. When shown in the drawing, these dimensions are called shown or driving dimensions, because you can use them to drive the shape of the model through the drawing.
- **Inserting Driven Dimensions**—Creating new dimensions within the drawing. These inserted dimensions are called *added* or *driven* dimensions, because their association is only one-way, from the model to the drawing. If dimensions are

changed in the model, all edited dimension values and the drawing are updated, but you can't use these driven dimensions to edit the 3D model.

- Adding Nondimensional Detailing

You can minimize the level of detail by:

- Temporarily erasing either the information shown from the 3D model or newly created details
- Cleanup
- Delete the newly created details (You cannot delete the information passed from the model)
- Placing dimensions and detail items on layers--which enable you to turn the display of information on and off as needed.

The Concept of Showing and Erasing

When you create a 3D model, you simultaneously create various references useful for detailing the model in a drawing, including dimensions, reference dimensions, geometric tolerances, symbols, axes, and others. When you import a 3D model into a 2D drawing the 3D dimensions and stored model information maintain parametric associativity with the 3D model but, by default, they are invisible. You can then selectively choose which 3D model information to show on a particular view, which is the concept of showing and erasing.

Items that you make visible are referred to as shown. These "shown" dimensions are associative to the 3D model in both directions, so you can use them to drive the model dimensions from within the drawing environment.

Items that are not visible are referred to as erased. The erased 3D detailing items are maintained within the 3D file database, unless you delete them from within the 3D model. You cannot permanently delete these 3D detailing items from within the drawing environment. Erased items stay erased from session to session.

You can show or erase all stored 3D model information at any time during the detailing process.

As you show and erase detailing, remember you can only show one instance of a 3D detail item for a feature per drawing. You can move a shown 3D detail item from one view to another, for example, from a general view to a detailed view where it is more appropriate. To provide dimensioning for views when shown (driving) dimensions are "used up" in other views,or if you need to create a dimension that was not used to define the 3D geometry, insert an additional (driven) dimension.

Note:

- You can also show or erase entire drawing views. This does not delete them from the drawing. Erasing views can help in repainting large drawing files. Click **View > Drawing Display > Drawing View Visibility**.

- For short term invisibility, you can use the **Hide** and **Unhide** shortcut commands available from selected parts in the model tree. Hidden objects do not stay hidden from session to session however, and erased objects do.

Dimensioning the Model

About Dimensioning the Model

Dimensioning the models in your drawings can be as simple or complex as required. To meet your drawing needs, you can:

- Show driving 3D model dimensions, which maximizes Pro/ENGINEER model-drawing associativity.
- Insert driven dimensions, which provides flexibility for repeating and nonexistent dimensions.
- Customize the dimension display, which enables you to reformat and reposition dimensions.
- Modify the dimension scheme

To take full advantage of the associativity between the 3D model and the drawing, your first drawing dimensions should be shown from the model (driving dimensions). Of course, there will be instances where you need additional dimensions to show the same value for the same object, for example, a view repeated on another sheet. You should insert driven dimensions in these situations.

Note: You can also add nondimensional detailing, including geometric tolerances, symbols, and text notes.

Displaying Dimensions in Detailed and Partial Views

A dimension that references nonsolid geometric features (axes, datum points, datum planes, and so forth) can be present in a detailed or partial view only when the following requirements are satisfied:

- At least one of the entities being dimensioned must be within the spline boundary.
- This entity must also be within the view boundary of the view—the view boundary is defined by the solid geometry of the part. (Nonsolid geometric features are not considered to be solid geometry.)

Check that these requirements are met before you change a view to a detailed or partial view, or dimensions will disappear from the new view.

The drawing setup file option `clip_dimensions` affects the display of dimensions in detailed views. When you set it to `yes`, dimensions that are completely outside of the view boundary do not appear. Dimensions that cross the view boundary appear, and the leader that seems to have no geometric reference appears with a double arrow (>>). To change the arrow style, set the default arrow style of clipped dimensions by specifying a value for the drawing setup file option `clip_dim_arrow_style` (`double_arrow` is the default).

Saving Dimensions to the Part or Drawing

When you create dimensions in Drawing mode, the configuration file option `create_drawing_dims_only` determines whether the system saves them in the associated part or in the drawing itself.

When set to `no`, (the default), it saves all new model dimensions (not draft dimensions) created in the drawing to the associated part or assembly. Draft dimensions are still saved to the drawing.

When set to `yes`, it saves all new dimensions created in the drawing in the drawing only.

The setting should depend on your work scenario. If you are using Intralink, if the dims are stored in the model, the model will be marked as modified and will have to be re-submitted back to intralink. To avoid this every time you reference a model for drawing, you can set the option to `yes`.

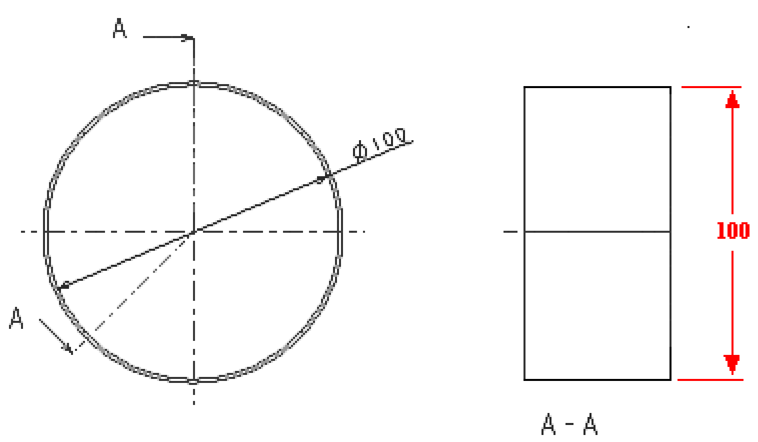
Alternately, if you want to use a set draft datum in a GTOL attached to a dimension, the dimension has to be stored in the model, so you would set the option to no.

Setting the option only applies to new dimensions created, it does not convert existing dimension. If you need to reset the option for an existing dimension, you will need to recreate it.

Tip: Dimensioning Rounds and Revolved Parts

The ability to show or insert dimensions for round features depends on the ability to select silhouette edges within drawings. You can select and use the normal dimensioning methods for silhouette edges.

You can properly dimension the diameters in unfolded views by creating linear dimensions between two silhouette edges. For example, you might dimension a cylindrical part in the following manner:



Showing Model Dimensions

About Showing Model Dimensions

When you import a 3D model into a 2D drawing the 3D dimensions and stored model information maintain parametric associativity with the 3D model but, by default, they are invisible. You can then selectively choose 3D model information to show on a particular view, which is the concept of showing and erasing.

Items that you make visible are referred to as shown. These "shown" dimensions are associative to the 3D model in both directions, so you can use them to drive the model dimensions from within the drawing environment.

While showing model dimensions and detailing, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase.

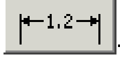
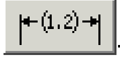
After you place the model dimensions and detailing on the drawing you can adjust their positions on the sheet and customize the format.

Consider the following when you show model dimensions and detailing in your Pro/ENGINEER drawings:

- Only one driving dimension for each model dimension may exist in a drawing. A drawing may have several views of the same object, but only one driving dimension for each feature of the model may be shown.
- It is possible to unintentionally edit the model. If a driving dimension is edited, it turns white in color as a warning that there is a discrepancy between the drawing and the model. When you regenerate the model, the drawing accepts the new dimension. It is possible, using configuration options, to break the link between model and drawing, but this is not the usual Pro/ENGINEER usage.

To Show Dimensions from the 3D Model

You can show the dimensions that have been passed to the drawing from the 3D model. Remember, only one instance of a driving dimension may be shown per drawing file. If you need to repeat these dimensions on other sheets, insert (driven) dimensions.

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.
3. Under **Type**, indicate which type of dimension to show, or both types:
   - 
—Displays the normal model dimensions.
   - 
—Displays reference dimensions.
4. Use the following options in the **Options** tab to filter the dimensions you want to show in the drawing:
   - **Erased**—Show previously erased items.

- **Never Shown**—Show items that are yet to be displayed in the drawing.
- **Switch to Ordinate**—Convert linear dimensions to display as ordinate.

5. Under **Show By**, use the following options to define where you want to display the dimensions:
    - **Feature**—Show the dimensions for a particular feature on the drawing.
    - **Part**—Show the dimensions for a particular part on the drawing.
    - **View**—Show all the dimensions for the features and parts within a particular drawing view.
    - **Feature and View**—Show the dimensions for a feature that appears in several views in one selected view. For example, if you have a feature to dimension that appears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to display it.
    - **Part and View**—Show the dimensions for a part that appears in several views in one selected view.
    - **Show All**—Show all the dimensions within the drawing.
6. Select the appropriate feature, part, or view to display the dimensions. You may need to repaint the drawing for the dimensions to be displayed. When the dimensions are displayed the **Preview** tab becomes available.
7. Of the dimensions previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:
    - **Sel to Keep**—Select the individual dimensions to show in the drawing. Any dimensions not selected will be erased.
    - **Sel to Remove**—Select dimensions to remove from the drawing. Any dimensions not selected will remain in the drawing.
    - **Accept All**—Keep all the previewed dimensions.
    - **Erase All**—Erase all the previewed dimensions.

    While you are showing the dimensions, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and **Resume Show and Erase**.

    If you need to select multiple dimensions to keep or remove, you can do this by either holding down the `CTRL` key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.
8. When you are finished with the preview, click **OK** in the **Select** dialog box. The dimensions are displayed in the drawing.

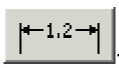
9. Click **Close** to close the **Show/Erase** dialog box.

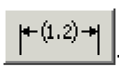
Note: The `show_preview_default` drawing setup file option lets you set the preview tab default to either select to keep or select to remove.

To Show Dimensions in an Assembly Drawing

In assembly drawings, you can display the parameters of assembly features and all assembly components; however, you cannot show the dimensions of subassemblies. The dimensions in an assembly drawing are visible only at the assembly level. You must have the assembly from which a drawing was created in session in order for the dimensions to appear for modification.

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.
3. Under **Type**, indicate which type of dimension to show, or both types:

- 

—Displays the normal model dimensions.
- 

—Displays reference dimensions.

4. Define where you want you want to display the dimensions:

- **Do not show merged feature dimensions**—To display the dimensions of a part without showing the dimensions of a merged feature, do one of the following:
  - Click **Part** and select the part on the screen.
  - Click **Edit > Find** from the main menu, and use the Find dialog box to list the parts of the assembly. Select the part from the list.
- **Show merged feature dimensions**—To display the dimensions of a merged feature in an assembly drawing, select **Feature** in the dialog box and uae the right mouse button shortcut menu **Query box** to select the feature of the part that was used for reference or copy.

Note: When you use automatic replacement to replace an assembly model with another model of the same family, the equivalent dimensions appear if the system shows dimensions in the assembly drawing on a component that you replace with the new instance. If you use manual replacement, it does not preserve the dimensions.

About Automatically Showing Detail Items Upon View Creation

In a drawing view, you can automatically display the annotations attached to a model in 3D mode by setting the `auto_show_3d_detail_items` configuration option to `yes`.

A detail item must also fulfill the following conditions to be displayed automatically:

- All items must have annotation planes.

- The item's annotation plane must be parallel to the view orientation.
- The item's viewing direction must match the viewing direction of the drawing.
- The item's attachment references must be available in the view. In case of on entity and leader references, the attachment types are available in the view.

If these conditions are met, the following detail items are automatically shown upon view creation:

- Notes
- Geometric Tolerances
- Surface finishes
- Symbols
- Driven dimensions
- Reference dimensions
- Model Set Datum Annotation Elements

  Note: The Set Datum Annotation Elements, except the Set Datum Annotation Elements created using **On Geometry** in the **Datum** dialog box, need not fulfil all the conditions required for automatic display in the view. Model set datum planes are displayed in all the views where the set datum plane is perpendicular to the view orientation regardless of the value of the `auto_show_3d_detail_items` configuration option.

- Model set datum axes

  Note: Model set datum axis annotations need not fulfil all the above conditions to be displayed in the view. Model set datum axes are displayed in all the views where the annotation tag plane is parallel to the view plane and the `auto_show_3d_detail_items` configuration option is set to `yes`.

- Model set datum planes

  Note:

  - Model set datum planes need not fulfill all the above conditions to be displayed in the view. Model set datum planes are displayed in all the views where the datum plane is perpendicular to the view orientation regardless of the value of the `auto_show_3d_detail_items` configuration option.
  - The Model Set Datum Annotation Elements created using **On Geometry** in the **Datum** dialog box must fulfil all the conditions required for automatic display on view creation.

- "Free" placed detail items. These are displayed automatically if they meet the conditions for parallelism and viewing direction.

Note:

- Driving dimensions that are not Annotation elements are not displayed on annotation planes and are therefore, not automatically displayed.
- You can automatically show detail items only on the creation of a new view. The detail items are not automatically shown in existing views in the drawing sheet or on modifying an existing view.
- On changing the view orientation, the automatically displayed detail items display behavior that is similar to items that have been inserted manually.
- For assembly and part simplified representations, only detail items whose references are available in the simplified representations and which fulfill the conditions required for automatic display are displayed upon view creation. For example, if all the references of a dimension are included in the simplified representation definition, and the dimension meets the conditions for automatic display, then the dimension is automatically shown. However, if all the references of the dimension are not included in the simplified representation, then the dimension is not shown.

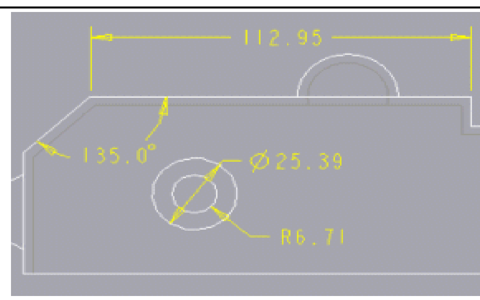
Inserting Dimensions

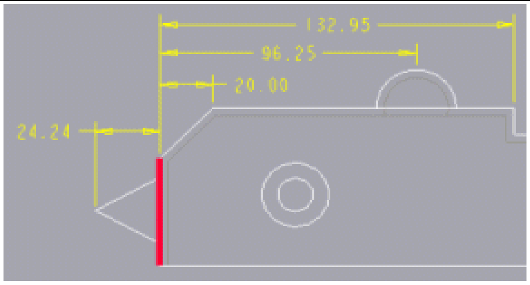
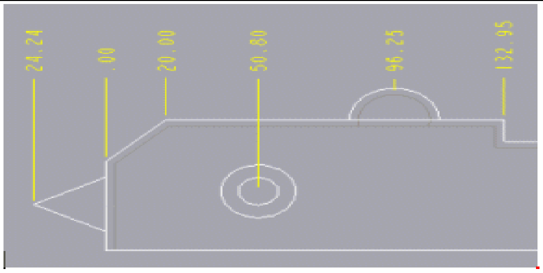
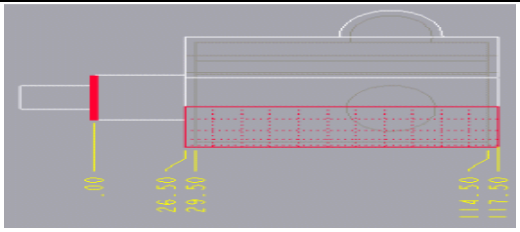
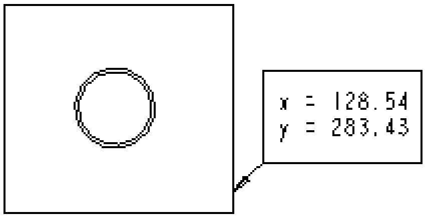
About Inserting Dimensions

A driven dimension is created by the user. This type of dimension reports a value based upon the references selected when the dimension is created. It is not possible to modify the value of a driven dimension as its value is derived from the position of its references. You cannot use a driven dimension to pass values back to the model, as you can with a driving dimension.

Formats of Added Dimensions

There are many types of driven dimensions.

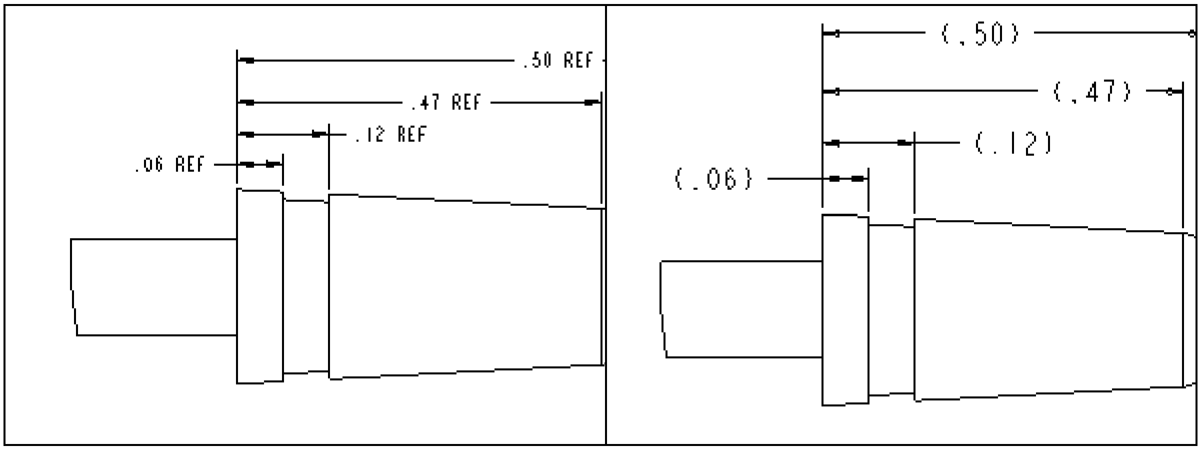
Type	Description	Example
Standard (New References)	Creates a dimension based upon 1 or 2 selected references. Depending upon the references, the result can be an angular, linear, radius, or a diameter dimension.	

Type	Description	Example
Common References	Adds dimensions between a common base object and one or more objects.	
Ordinate	Ordinate dimensions are dimensions that measure a linear distance from an object identified as a baseline.	
Auto Ordinate	Automatically creates ordinate dimensions in parts and sheet metal parts.	
Coordinate Dimension	Lets you assign an existing x- and y- dimension to a label and leader box.	

Reference Dimensions

All reference dimensions are identical to standard dimensions except that they have special notation that indicates they are reference dimensions. You can use the `parenthesize_ref_dim` configuration option and set the value to `yes` to enclose these dimensions in parenthesis. If this option is set to `no`, the text "REF" will follow the dimension value.

The following illustrations show reference dimensions.



To Insert Additional Dimensions

Use added, or *driven* dimensions to repeat dimensions that you have already shown. You can add standard linear or reference type dimensions in linear, common reference or ordinate format. You cannot modify the 3D model through driven dimensions.

 Linear dimension	 Reference dimension

1. Click **Insert > Dimension** or **Insert** > **Reference Dimension**. Click on the following to specify use of:
    - New References
    - Common Reference
    - Ordinate dimension
    - Auto Ordinate
    - Coordinate Dimensions

    The **ATTACH TYPE** menu appears on the menu manager.

2. Select an edge, an edge and point, two points, or a vertex using the ATTACH TYPE menu commands:
    - **On Entity**—Attaches the dimension to the entity at the pick point, according to the rules of creating regular dimensions.
    - **Midpoint**—Attaches the dimension to the midpoint of the selected entity.
    - **Center**—Attaches the dimension to the center of a circular edge. Circular edges include circular geometry (holes, rounds, curves, surfaces, and so forth) and circular draft entities. If you select a noncircular entity, attaches the dimension to the entity, as does **On Entity**.

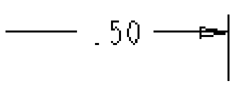
- **Intersect**—Attaches the dimension to the closest intersection point of two selected entities.
- **Make Line**—References the current x- and y-axes in the orientation of the view of the model.

Note: To create a driven dimension of an arc length , select the ends and the middle of an arc; then choose **ARC DIM TYPE** > **Arc Length**.

3. Click the beginning and ending references for the dimension. If you have selected two arcs or circles, use the **ARC PNT TYPE** menu to do one of the following:
   - Click **Center** to create a dimension between the centers of the arcs, ellipses, or circles.
   - Click **Tangent** to create a dimension between the circle, arc, or ellipse edges—the tangency closest to where you selected.
   - Click **Concentric** to create a dimension between two concentric circles or arcs.
4. Middle-click to complete the dimension. You are still in dimensioning mode and you may create another dimension.

To Insert a Reference Dimension

Use added, or *driven* dimensions to enter dimensions with a reference notation. You can add reference dimensions in linear, common reference or ordinate format. (You cannot modify the 3D model through driven dimensions.)

 Linear dimension	 Reference dimension

1. Click **Insert > Reference Dimension**. Use the fly-out menus to specify use of **New References**, a **Common Reference**, or an **Ordinate** dimension. The **ATTACH TYPE** menu appears on the Menu Manager.
2. Select an edge, an edge and point, two points, or a vertex using the **ATTACH TYPE** menu commands:
   - **On Entity**—Attaches the dimension to the entity at the pick point, according to the rules of creating regular dimensions.
   - **Midpoint**—Attaches the dimension to the midpoint of the selected entity.
   - **Center**—Attaches the dimension to the center of a circular edge. Circular edges include circular geometry (holes, rounds, curves, surfaces, and so forth) and circular draft entities. If you select a noncircular entity, attaches the dimension to the entity, as does **On Entity**.
   - **Intersect**—Attaches the dimension to the closest intersection point of two selected entities.

- **Make Line**—References the current x- and y-axes in the orientation of the view of the model.

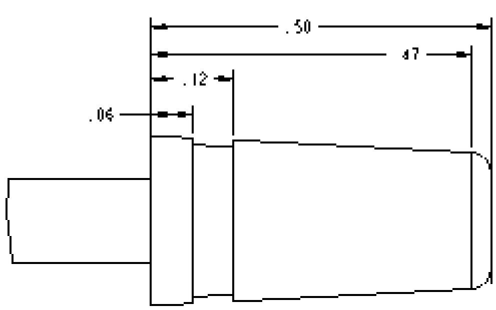
Note: To create a driven dimension of an arc length , select the ends and the middle of an arc; then choose **ARC DIM TYPE** > **Arc Length**.

3. Click the beginning and ending references for the dimension. If you have selected two arcs or circles, use the **ARC PNT TYPE** menu to do one of the following:
   - Click **Center** to create a dimension between the centers of the arcs, ellipses, or circles.
   - Click **Tangent** to create a dimension between the circle, arc, or ellipse edges—the tangency closest to where you selected.
   - Click **Concentric** to create a dimension between two concentric circles or arcs.
4. Middle click to complete the dimension. You are still in dimensioning mode and you may create another dimension.

To Insert Dimensions from a Common Reference

1. Click **Insert > Dimension** or **Reference Dimension,** then **Common Reference**.
2. Click the line to use for the common reference. It highlights in magenta.
3. Click the line to use for the first dimension. Middle click to place the dimension. You are still in dimension mode, referencing the common dimension.
4. Click the line for the second dimension. Middle click to complete.
5. Proceed this way until you have entered all the common reference dimensions, then middle click to exit.

Dimensions from a common reference.

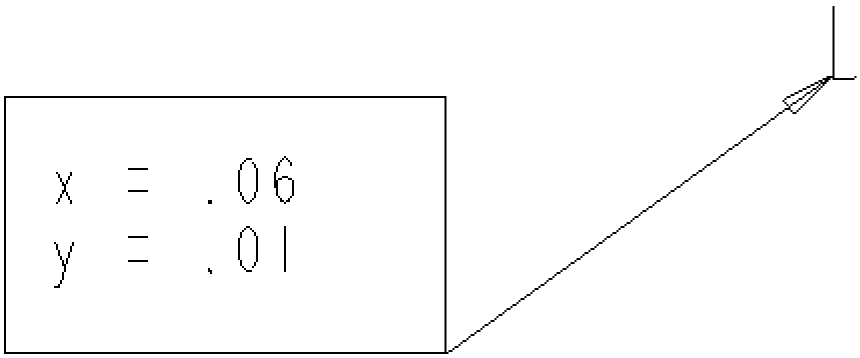


To Insert a Coordinate Dimension

1. Click **Insert > Coordinate Dimension**.
2. Click an edge, entity, or axis where the coordinate dimension leader is attached.

3. Click the location to place the dimension symbol.
4. Select the appropriate x- and y-dimension values from linear dimensions to place in the symbol. The system creates a coordinate dimension with these values.

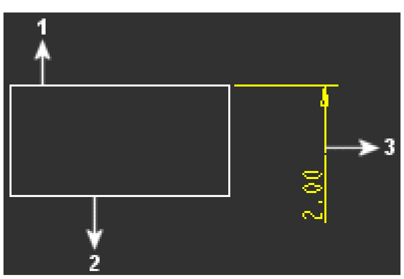
Example: Coordinate Dimension Symbol



To Insert Automatically Clipped Linear Dimensions

You can create automatically clipped linear dimensions that report twice the distance between the selected entities. Clipped linear dimensions are useful for dimensioning revolved protrusions, extrusions and copy geometry.

1. Click **Insert** > **Dimension** > **New References**. The **ATTACH TYPE** menu appears.
2. Click any command on the **ATTACH TYPE** menu and select an edge, straight entity, datum plane, axis or silhouette.
3. Select a second edge, straight entity, datum plane, axis or silhouette as the second reference.
4. Select the first reference again.
5. Middle-click at the required location to place the dimension. A clipped linear dimension, equal to twice the distance between the selected entities, is created as shown in the following figure.



1. The entity selected as the first reference
2. The entity selected as the second reference
3. The automatically clipped linear dimension that is centered about the second selected reference

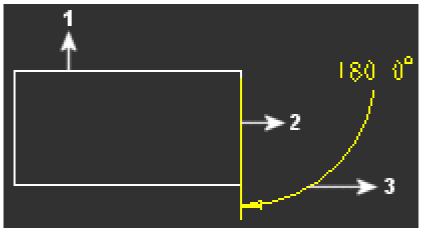
Note:

- It is possible to create an inaccurate dimension if the second reference is not at the center of the required dimension.
- The dimension text is centered about the second selected reference.
- The clipped side of the dimension extends a short distance beyond the centerline.
- If the distance between the selected entities changes, the dimension updates automatically.
- The `clip_dim_arrow_style` drawing setup file option controls the arrow display of the clipped, linear dimension.
- You can also create automatically clipped linear reference dimensions that report twice the distance between the selected entities by clicking **Insert** > **Reference Dimension** > **New References**.

To Insert One-Sided, Clipped, Double Angular Dimensions

You can automatically create one-sided, clipped angular dimensions that report twice the angle between the selected entities.

1. Click **Insert** > **Dimension** > **New References**. The **ATTACH TYPE** menu appears.
2. Click any command on the **ATTACH TYPE** menu and select an entity you want to dimension, as the first reference.
3. Select another entity that causes an angle, as the second reference.
4. Select the first reference again.
5. Middle-click at the required location to place the dimension. A clipped, angular dimension, equal to twice the angle of the selected references, is created, as shown in the following figure.



1. The entity selected as the first reference

2. The entity selected as the second reference

3. The automatically clipped angular dimension

Note:

- The dimension is clipped at the second reference.
- The dimension leader extends to the end of the dimension text when the `angdim_text_orientation` drawing setup file option is set to parallel.

- The dimension text is centered about the second reference.
- The `clip_dim_arrow_style` drawing setup file option controls the arrow display of the clipped, angular dimension.

To Automatically Dimension Radial Patterns

To automatically dimension a radial pattern feature (such as a bolt circle):

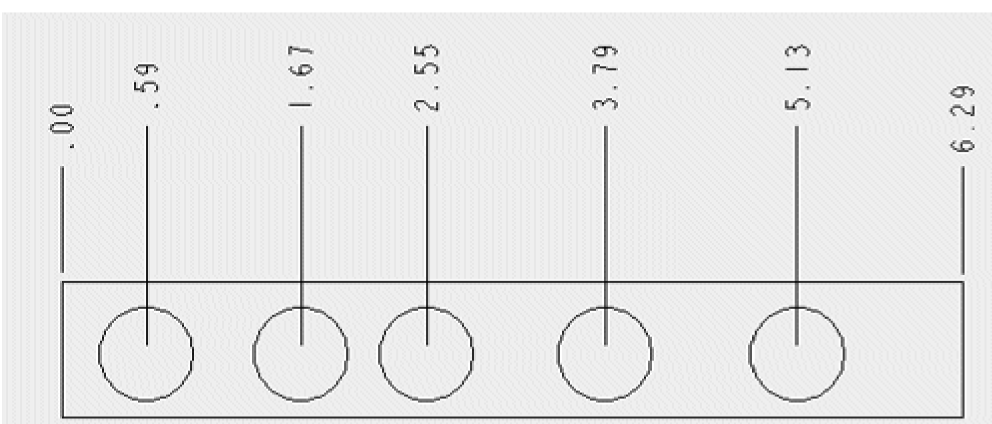
1. Click **Insert** > **Dimension** or **Ref Dim** > **Common Reference**. The Attach Type menu appears on the Menu Manager. You are prompted "Select geometry to use as a common dimensioning reference."
2. In the **Attach Type** menu click **Center**.
3. Select the axis of one of the pattern features. You are prompted to select additional entities to dimension.
4. Select the base point to dimension against. For example, you can select a csys at the center of the pattern or an entity outside of the pattern.
5. Click **OK** in the Select Confirmation window. The Dim Orient menu appears.
6. Select an orientation. You are prompted: `Create Dims to the other members of the feature pattern?`
7. Click **Yes**. The remaining pattern members are dimensioned.

Ordinate Dimensions

About Ordinate Dimensions

Ordinate dimensions use a single witness line with no leader, and are associated with a baseline reference, shown in the next figure as ".00". All dimensions that reference the same baseline must share a common plane or edge.

You can create ordinate dimensions within drawings, using either new or existing start points.



You can control the format display for ordinate dimensions using the following combination of drawing file setup options:

- ord_dim_standard—Sets the standard. For example, std_jis places dimensions along a connecting line that is perpendicular to the baseline and starts with an open circle.
- draw_arrow_style—Controls the style of the arrow and circle—whether they are open, closed, or filled.
- draw_dot_diameter—Sets the diameter for the leader line dots (the circle on the baseline).

Creating Ordinate Driven Dimensions

When creating ordinate driven dimensions, you must create a baseline in the required direction by converting a standard dimension into an ordinate dimension, and then create ordinate dimensions using the baseline reference.

When creating ordinate driven dimensions, keep in mind the following:

- You cannot create ordinate dimensions from a baseline dimension when the configuration file option create_drawing_dims_only is set to yes and a dimension with a d or ad type symbol (denoting either a driving model dimension or an associative draft dimension) is the active ordinate baseline.
- To control the orientation of the ordinate dimension text, set the drawing setup file option orddim_text_orientation to either parallel or horizontal.
- If you set the drawing setup file option ord_dim_standard, you can connect ordinate dimensions measured from one baseline with a line. When extension lines are interconnected, all related dimensions move when you move one.
- You cannot switch ordinate driven dimensions to another view.

To Create Ordinate Dimensions

You can create ordinate dimensions within drawings, using either new or existing start points. While adding a new ordinate dimension to an existing ordinate dimension or ordinate dimension group, you can select the existing baseline, the entity that the baseline is attached to, or any part of an existing ordinate dimension's witness line or text as the reference.

1. Click **Insert** > **Dimension** > **Ordinate**. The **Select** dialog box opens.
2. Select a new starting point or baseline reference or select an existing baseline, the entity that the baseline is attached to, or any part of an existing ordinate dimension’s witness line or text. The selected geometry is highlighted.
3. Select the entity or entities you want to dimension. The selected entities are highlighted.
4. Middle-click at the required location to place the ordinate dimension.

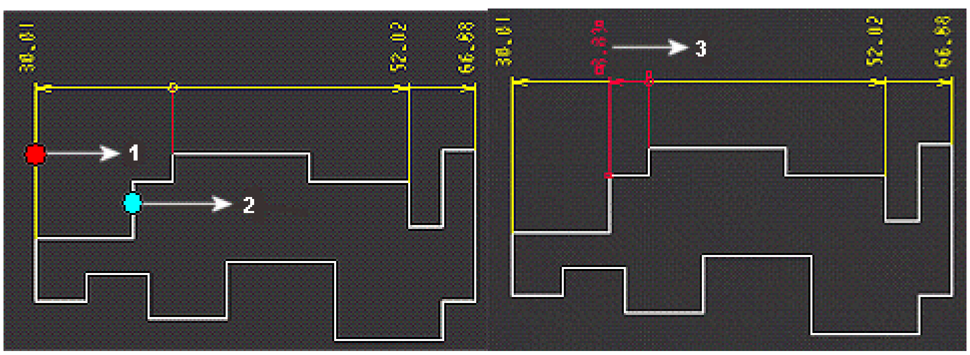
To Create Ordinate Dimensions Automatically

1. Create a drawing of the model for which you want to create ordinate dimensions automatically.
2. In the drawing window, click **Insert** > **Dimension** > **Auto Ordinate**. You are prompted to select one or more surfaces.
3. Select one or more surfaces for which you want to create ordinate dimensions. The **AUTO ORDINATE** menu appears.

   Note: You must select surfaces belonging to the same view of the drawing.
4. Click **Select Base Line**.
5. In the same view from which you selected the surface or surfaces, select a reference line (edge, curve, or datum plane) to create the ordinate dimensions. The ordinate dimensions are created automatically and are displayed in that view.

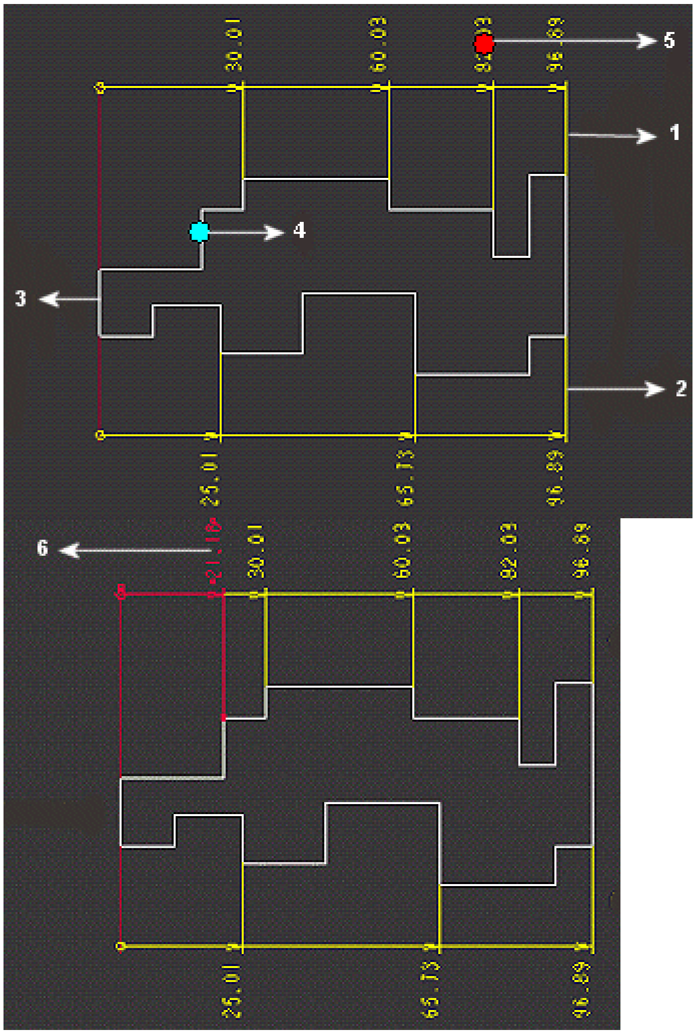
Example: Ordinate Dimensions

Single dimension group



1. An existing ordinate dimension selected as reference
2. The entity selected for dimensioning
3. The new ordinate dimension, created as part of the existing dimension group

Multiple dimension groups



1, 2. Ordinate dimension groups

3. Common entity as baseline reference for both the ordinate dimension groups

4. The dimension text of an existing ordinate dimension selected as reference

5. The entity selected for dimensioning

6. The new ordinate dimension created as part of the first dimension group

Showing Linear Dimensions as Ordinate

You can show dimensions or reference dimensions as linear or ordinate dimensions.

- If the system cannot show a dimension as ordinate from the selected baselines, it does not show it at all.
- If it shows dimensions as ordinate on the drawing, it shows them as ordinate on the model as well.

Typically, Pro/ENGINEER does not show negative or positive values for dimensions unless you set the configuration file option `show_dim_sign` to `yes`. However, if you create dimensions which reference features offset from a coordinate system, or assign a negative value to them, the negative value appears even if you have set this option to "no."

Converting Linear Dimensions to Ordinate

To modify a dimension type from linear to ordinate, you must first establish a reference baseline. For any set of related dimensions (dimensions that share a common plane or edge), you need to create only one baseline reference. If you just created a baseline, it remains set until you set another, or until you exit the MOD DIM TYPE menu. Only one baseline reference can be current (set) at one time. To set another, choose **Set Base** and select the existing baseline reference that you want to set. You can select dimensions that have witness lines coincident with the set baseline and convert them from linear to ordinate dimension type.

When converting the dimension, you can also add a jog to the witness line to improve the spacing of the dimensions.

When you convert an ordinate dimension to linear, the dimension loses its association with the baseline reference. If you convert the dimension back to an ordinate dimension, you can select any appropriate baseline reference.

1. Select any ordinate dimension to convert.
2. Click **Edit > Toggle Ordinate/Linear**.
3. Select the witness line to be the baseline reference. The dimension changes automatically to an ordinate type dimension, indicated with a value of .00.

Notes:

- You cannot select dimensions that do not share the *same* baseline reference.
- When you switch dimensions from linear to ordinate, the baseline reference regenerates. However, if an orphan baseline (one that does not have dimensions associated with it) has driven children (dimensions), the baseline and the children do not regenerate. If many driven dimensions in a drawing reference an orphan baseline, modify the dimensioning scheme by selecting a side from which to attach the baseline. The baseline then regenerates at the correct location. To update children, choose **Edit** > **Regenerate** > **Draft.**
- Baseline references appear on the drawing or in the model as ".00". Using the **Show/Erase** dialog box, you can erase them from the drawing for cosmetic reasons. Also, you can select and delete baseline references that are no longer necessary.
- The dimension to be converted to ordinate must be shown as linear. The following dimensions cannot be converted to ordinate:
  - A diameter dimension shown as linear
  - A centerline dimension

Deleting Ordinate Dimensions

You can delete ordinate dimensions in the following ways:

- Delete ordinate dimensions and its baseline in a single operation by selecting the baseline, right-clicking, and clicking **Delete** on the shortcut menu. If you delete the ordinate dimension baseline, then all the dependent dimensions in the ordinate dimension chain are also deleted.

  Note: When you select the entire ordinate dimension chain including the baseline to delete, Pro/ENGINEER does not ask for confirmation before deleting.

- You can delete an ordinate dimension, independently of the baseline and other ordinate dimensions in the chain, by selecting the dimension you want to delete, right-clicking, and clicking **Delete** on the shortcut menu.
- You can delete the first created ordinate dimension from the ordinate dimension chain independently, by selecting that dimension, right-clicking, and clicking **Delete** on the shortcut menu.

Note:

- You can delete the ordinate dimensions that were created as linear dimensions and then converted to ordinate dimensions.
- You can recover the deleted ordinate dimension by clicking **Edit** > **Undo** immediately after deleting the dimension.
- When you delete the first created dimension in the ordinate dimension chain, the baseline does not change to purple and the other dimensions in the ordinate dimension chain are not affected.

Redefining the References for Ordinate Dimensions

You can redefine the references for ordinate dimensions in the following ways:

- Redefine the reference for an ordinate dimension in the ordinate dimension chain by clicking **Edit Attachment** on the shortcut menu or by clicking **Edit** > **Attachment**. You can then use the **Attach Type** menu that appears to define the new reference.
- You can also redefine the dimension reference by clicking **Edit Attach** in the **Dimension Properties** dialog box for the selected dimension. You can then use the **Attach Type** menu that appears to define the new reference.

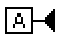
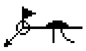
Note:

- You can redefine the references of ordinate dimensions that were created as linear dimensions and then converted to ordinate dimensions.
- You can also redefine the reference for the first dimension in the ordinate dimension chain.
- While redefining the references, if you select a reference that is not parallel to the ordinate baseline, an error message is displayed.
- You cannot redefine the references for multiple selected dimensions.

Detailing with Non-Dimension Items

About Non-dimension Detail Items

Non-dimension detail items include geometric tolerances, datums and axes, and various types of symbols and notes. These items may be passed from the 3D model, or they may be created as part of the drawing. Click below for more information on each:

	Geometric Tolerances
	Datum Planes
	Symbols
	Surface Finish Symbols
	Axes
	Datum Targets
	Balloons
	Text Notes

About Working with Draft Datums

When you create a draft datum in parametric Sketcher mode, the draft datum and the entity that it references maintain a parametric association.

You can manually adjust the ends of the draft datum that you create on the entity, by dragging the end handle. However, you cannot manually adjust the ends of a draft datum that you create using a vertex.

If you change the draft datum to a draft set datum, the parametric association between the draft datum and the entity that it references, is not affected.

The draft datum that references a model entity is similar to a parametric draft sketch entity. That is, if you move, delete, or erase the view of the model entity that the draft datum references, then the draft datum is also moved, deleted, or erased, respectively.

Similarly, the draft datum that references a draft entity is similar to a parametric draft sketch entity. That is, if you move the draft entity that the draft datum references, then the draft datum also moves. However, if you delete the draft entity, the draft datum is not deleted.

Note: The draft datum created in releases prior to Pro/ENGINEER Wildfire 4.0 remain non-parametric to the entity that it references.

Use the `leader_elbow_length` drawing setup file option to set the default length of the leader elbow for a draft datum and model datum. However, use the `set_datum_leader_length` drawing setup file option to set the default length of the leader for a draft set datum and model set datum.

If you change the value of the `set_datum_leader_length` drawing setup file option, the new default length is set for the leaders of new set datums. This will also change the leader length of all existing set datums whose leader lengths have not been previously adjusted.

If you explicitly change the length of the leader for a set datum, the value of the `set_datum_leader_length` drawing setup file option no longer affects the leader length of that leader.

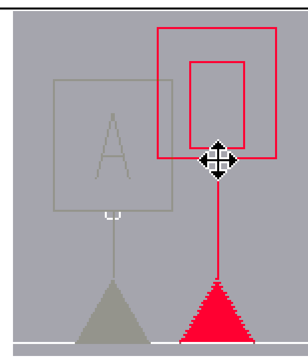
To Adjust the Draft Set Datum

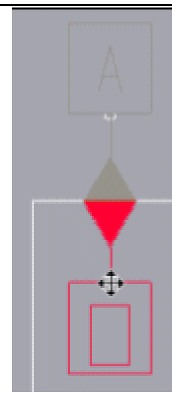
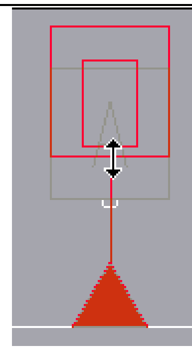
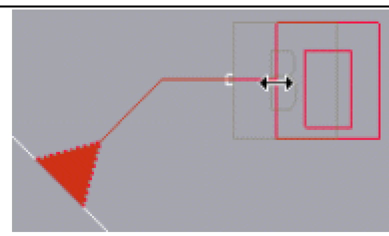
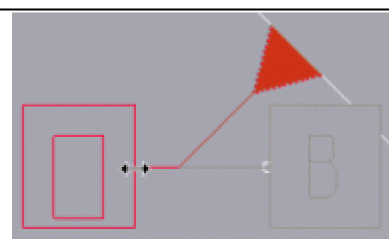
1. Select the draft set datum.
2. Move the pointer over the required handle of the selected draft set datum. The drag direction symbol appears.
3. Drag the handle to the new position.

Performing Dragging Operations on Draft Set Datums

After you create draft set datums, you can adjust their anchor position, leader length, and elbow length by dragging the appropriate handles. You can perform all the dragging operations on these draft set datums in the same way as that of model set datums.

You can perform the following few operations on draft set datums by selecting the associated handles:

Anchor and leader handle	You can drag the anchor and leader handle to adjust the anchor location and the leader length at the same time.	

	If you drag the anchor and leader handle to the other side of the anchor triangle, the anchor triangle flips.	
Leader handle	You can drag the leader handle to adjust the leader length.	
	If you drag the leader handle to the other side of the anchor, the anchor triangle flips.	
Elbow handle	You can drag the elbow handle to adjust the elbow length. **Note**: The elbow handle is available only if a draft set datum already has an elbow.	
	If you drag the elbow handle to the other side of the leader, the tag of the selected draft set datum flips.	

Geometric Tolerances

About Geometric Tolerances in Drawings

Geometric tolerances are the maximum allowable deviation from the exact sizes and shapes specified in the model design. Geometric tolerances are a comprehensive detailing tool that enable you to:

- Specify the critical surfaces on a model part
- Document the relationship between critical surfaces
- Provide information on how the part should be inspected and what deviations are acceptable

Within drawings, you can either show a geometric tolerance from the solid model or create one.

You can attach a geometric tolerance to dimensions (reference, driven, radius, or diameter), set datums, single or multiple edges, or another geometric tolerance. You can also place geometric tolerances as free notes anywhere on the drawing, attach them to leader elbow for notes, or relate them to dimension text.

You can attach multiple lines of additional text and text symbols to a geometric tolerance while creating or editing it. By default, the text style of the additional text is the same as that of the geometric tolerance text. You can edit it independent of the geometric tolerance text.

You can stack multiple geometric tolerances on another tolerance; or, if the first tolerance in a stack is attached to a dimension, you can attach them to the same dimension. As you create each geometric tolerance for a stack, the most recently created geometric tolerance is added to the bottom of the stack. If you set the stacked_gtol_align drawing setup file option to `yes`, then the stacked geometric tolerances automatically align in the control frame.

Note: The default value of the stacked_gtol_align drawing setup file option is `no`.

However, to attach a geometric tolerance directly to other geometric tolerances, dimensions, or datums, it must belong to the same model as the item to which it is attached.

Before you can reference a datum plane or axis in a geometric tolerance, you must set the datum plane or axis. You can attach a model set datum to a model geometric tolerance if the `gtol_datums` drawing setup file option value is set to `std_iso`, `std_iso_jis`, or `std_jis`. You can also attach draft set datums or draft datum axes to draft geometric tolerances.

After you attach a set datum to a geometric tolerance, you can drag or flip it using the set datum box or the drag handles. Within the drawing, you can drag the set datum beyond the control frame, in which case a witness line is created.

You can control whether the set datum attachment should be above or below the geometric tolerance control frame by using the `gtol_datum_placement_default` drawing setup file option.

Unlike dimensional tolerances, geometric tolerances do not have any effect on the part geometry.

Note:

- You can erase or delete geometric tolerances that are shown in drawings. If you delete a shown geometric tolerance, it is deleted in both the drawing and the model.
- You can control the restrictions in the **Geometric Tolerance** dialog box using the `restricted_gtol_dialog` configuration option.
- You cannot mix draft datums or draft axes with model geometric tolerances, or model datums with draft geometric tolerances.
- The `gtol_datum_placement_default` drawing setup file option applies to both model and draft set datums.
- The following rules apply when you attach a geometric tolerance to a dimension in Part mode:
    - If you place a geometric tolerance on a dimension in a part, and then create a drawing using that part, you must first show the dimension in the drawing using the **Show/Erase** dialog box. Otherwise, the geometric tolerance is not displayed.
    - If you attach a geometric tolerance to a part dimension that Pro/ENGINEER cannot display in an assembly drawing, it does not display the geometric tolerance either.

Duplicating Draft Geometric Tolerances

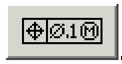
You can use the **Edit** > **Copy** and **Edit** > **Paste** commands to duplicate draft geometric tolerances or a draft geometric tolerance stack. When you duplicate a geometric tolerance, a draft geometric tolerance annotation is created. The original and duplicated geometric tolerances are not parametrically associated. You can update the attributes of the duplicated geometric tolerance regardless of the attributes of the original one.

Note:

- You cannot duplicate model geometric tolerances.
- After duplicating a geometric tolerance, ensure that you update all references and attachments.
- If you select a geometric tolerance stack to create a copy, the entire stack is copied. You cannot duplicate a single geometric tolerance from a geometric tolerance stack.

To Show Geometric Tolerances

You can show geometric tolerances that were created in the Part and Assembly modes of the model.

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.
3. Under **Type**, click .
4. Use the following in the Options tab to filter the geometric tolerances you want to show in the drawing:
    - **Erased**—Show previously erased items.
    - **Never Shown**—Show items that are yet to be displayed in the drawing.
    - **Switch to Ordinate**—Convert linear dimensions to display as ordinate.
5. Under **Show By**, use the following options to define where you want to display the geometric tolerances:
    - **Feature**—Show the geometric tolerances for a particular feature on the drawing.
    - **Part**—Show the geometric tolerances for a particular part on the drawing.
    - **View**—Show all the geometric tolerances for the features and parts within a particular drawing view.
    - **Feature and View**—Show the geometric tolerances for a feature that appears in several views in one selected view. For example, if you have a feature with geometric tolerance that apeears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to display it.
    - **Part and View**—Show the geometric tolerances for a part that appears in several views in one selected view.
    - **Show All**—Show all the geometric tolerances within the drawing.
6. Select the appropriate feature, part, or view to display the geometric tolerances. You may need to repaint the drawing for the geometric tolerances to be displayed. When the geometric tolerances are displayed the **Preview** tab becomes available.
7. Of the geometric tolerances previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:
    - **Sel to Keep**—Select the individual geometric tolerances to show in the drawing. Any geometric tolerances not selected will be erased.
    - **Sel to Remove**—Select geometric tolerances to remove from the drawing. Any geometric tolerances not selected will remain in the drawing.
    - **Accept All**—Keep all the previewed geometric tolerances.
    - **Erase All**—Erase all the previewed geometric tolerances.

While you are showing the geometric tolerances, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and **Resume Show and Erase**.

If you need to select multiple geometric tolerances to keep or remove, you can do this by either holding down the `CTRL` key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.

8. When you are finished with the preview, click **OK** in the **Select** dialog box. The geometric tolerances are displayed in the drawing.
9. Click **Close** to close the **Show/Erase** dialog box.

Note:

- The `show_preview_default` drawing setup file option lets you set the preview tab default to either select to keep or select to remove.
- You can erase or delete geometric tolerances that are shown in drawings. If you delete a shown geometric tolerance it is deleted in both the drawing and the model.

To Insert a Geometric Tolerance into a Drawing

Note: Geometric tolerances are simultaneously updated on the drawing as you create them. You can check your work as you go along and make adjustments, if necessary.

1. Click **Insert** > **Geometric Tolerance**. The **Geometric Tolerance** dialog box opens.
2. Define the type of geometric tolerance to insert:

—Straightness	—Flatness	—Circularity
—Cylindricity	—Line Profile	—Surface Profile
—Angularity	—Perpendicularity	—Parallelism
—Position	—Concentricity	—Symmetry
—Circular Runout	—Total Runout	

3. Using the tabbed pages in the **Geometric Tolerance** dialog box, define the following requirements for your geometric tolerances:
   - Specify the model and the reference entity to which to add the geometric tolerance, as well as place the geometric tolerance on the drawing.
   - Specify the datum references and material conditions for a geometric tolerance, as well as the value and datum reference of a composite tolerance.
   - Specify the tolerance value and the material condition.
   - Specify the geometric tolerance's symbols and modifiers, as well as the projected tolerance zone.
   - Specify additional text that you want associated with a geometric tolerance while creating or editing it.
4. Once you are done defining the geometric tolerance, return to the **Model Refs** tab and click **Place Gtol**. The geometric tolerance is inserted in the drawing. Click **OK** to save the geometric tolerance settings.

   Pro/ENGINEER clears the reference entity selection and placement information from the dialog box, but retains all other data. When you reenter the geometric tolerance creation mode, Pro/ENGINEER retains all the commands in the previous session of the geometric tolerance creation for the object in the current window.

Note:

- After you have created a geometric tolerance, click **New Gtol** to create additional geometric tolerances. After you have modified an existing geometric tolerance, click **New Gtol** to create a copy of the geometric tolerance that was just modified.
- Click **Copy From** to copy the parameter and option settings of an existing geometric tolerance to the geometric tolerance that you are creating or editing.

To Specify Model References for Geometric Tolerances

Complete the following steps to specify the model and the reference entity to which to add the geometric tolerance, as well as place the geometric tolerance on the drawing.

1. Click **Insert** > **Geometric Tolerance**. The **Geometric Tolerance** dialog box opens.
2. Under the **Model Refs** tab, in the **Model** list, the current drawing model is selected by default. You can select the topmost model from the **Model** list or click **Select Model** to select a sub model.
3. To define where you want to place the geometric tolerance, select a reference from the **Type** list under **Reference: To Be Selected**.

   Note: Before you can reference a datum plane or axis in a geometric tolerance, you must set the plane or axis.

4. Click **Select Entity** and select the reference from the drawing. All reference entity types are not available for all geometric tolerance types. You must select a new entity whenever you change the entity type. Pro/ENGINEER does not allow you to complete the creation of a geometric tolerance until you select a reference entity. The reference entity is the geometry or feature that the geometric tolerance controls; you should not use it instead of a set datum or as an attachment type for the geometric tolerance.

5. To define how to place the geometric tolerance, select one of the following placement types from the **Type** list under **Placement: To Be Placed**.

   - **Dimension**—Attach the geometric tolerance to a dimension. The `gtol_dim_placement` drawing setup file option determines its location below the dimension. You can attach multiple geometric tolerances (stacked) to the same dimension, and they behave as a single tolerance when you manipulate the dimension symbol.

   - **Dimension Elbow**—Attach the geometric tolerance to the leader elbow of a radius or diameter dimension that utilizes the elbow-leader style. Attaching a geometric tolerance to a dimension elbow moves the existing dimension text above the elbow. Pro/ENGINEER adjusts the length of the elbow to fit the longest line of text.

     Note: You cannot select a dimension whose text is inside a circle and does not have an elbow. Similarly, if you attach a geometric tolerance to a radius or diameter dimension, you cannot move it inside a circle.

   - **As Free Note**—Place the geometric tolerance anywhere on the drawing. In Part mode, you cannot display free geometric tolerances that were created in Drawing mode.

   - **Note Elbow**—Attach the geometric tolerance to the leader elbow of an existing note. You can attach the geomatric tolerance to two and three dimensional notes with leaders, hole notes, thread notes and ISO leader notes. Attaching a geometric tolerance to a note elbow will move the existing note text above the elbow. The length of the elbow adjusts to fit the longest line of text in the note. Pro/ENGINEER always displays the geometric tolerance horizontally. However, any vertical or angular note text will retain its orientation.

     Note: You cannot attach geometric tolerances to free notes.

   - **With Leader**—Attach the geometric tolerance to multiple edges, including datum quilt edges, with leader lines or to dimension witness lines.

   - **Tangent Leader**—Attach the geometric tolerance to an edge along a leader line that is tangent to the selected edge, orienting the geometric tolerance text box at the same angle as the leader.

   - **Normal Leader**—Attach the geometric tolerance to an edge along a leader line that is perpendicular to the selected edge.

   - **Other Gtol**—Attach the new geometric tolerance to an existing one. You cannot attach the existing geometric tolerance to a dimension.

- **Make Dim**—Create a driven dimension and attach the dimension to it. This dimension belongs to the drawing. The geometric tolerance appears in standard dimension format, but with the geometric tolerance instead of a dimension value.
- **Offset**—Attach the geometric tolerance to a dimension, dimension arrow, geometric tolerance, note or symbol. The geometric tolerance is grouped with the selected reference entity. When you move the entity, the geometric tolerance also moves. Moving the geometric tolerance independently of the entity resets the offset.

6. After you have finished defining additional datum references, tolerance values, or symbols, click **Place Gtol**. The geometric tolerance is inserted in the drawing.

Define Datum References for Geometric Tolerances

Complete the following to specify the datum references and material conditions for a geometric tolerance, as well as the value and datum reference of a composite tolerance.

1. To define datum references for a geometric tolerance that permits datum references, click the **Datum Refs** tab.

   Note: If the Geometric Tolerance dialog box is not open, click **Insert** > **Geometric Tolerance**.

2. Select and define the **Primary**, **Secondary**, and **Tertiary** references as necessary. The lists contain the currently selected datum and all other datums in the current geometric tolerance model. Click to select another set datum or axis.

   You can assign a material condition to the reference by selecting a symbol from the list next to the basic or compound reference:

   - **LMC**—Least material condition
   - **MMC**—Maximum material condition
   - **RFS (with symbol)**—Regardless of feature size
   - **RFS (no symbol)**—RFS, but does not show a symbol in frame

   The references do not need to follow the order prescribed by the primary, secondary, or tertiary tabs. Click **Unordered** to allow two datum references to be listed in the same section of the control frame.

3. To define a composite tolerance, click **Composite Tolerance**. Type a tolerance in the **Value** box and select the desired datum from the **Datum Reference** list:

   - **None**
   - **Primary**
   - **Secondary**

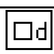
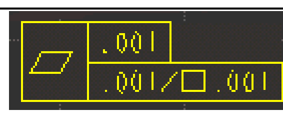
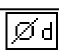
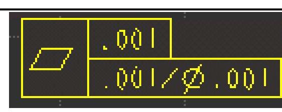
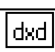
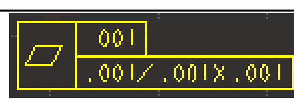
- **Tertiary**

To Set Tolerance Values for Geometric Tolerances

You can set the tolerance values and material condition for geometric tolerances while inserting a geometric tolerance or you can set the tolerance values and material condition for existing geometric tolerances.

1. Select the geometric tolerance, right-click, and click **Properties** on the shortcut menu. The **Geometric Tolerance** dialog box opens.
2. Click the **Tol Value** tab. The **Overall Tolerance** checkbox is selected by default, with the default value displayed.
3. To show only the geometric tolerance without any restricted length, type a value for the geometric tolerance in the **Overall Tolerance** box.
4. Type a new name in the **Name** box, if required.
5. To restrict the length to which the tolerance applies, select the **Per Unit Tolerance** checkbox.
6. Type the required values in the **Value/Unit** and **Unit Length** boxes.

   For geometric tolerances of the type **Flatness**, **Unit Area** has to be specified as a combination of **Value** and **Display**. The **Value** is the numeric value of unit area and **Display** specifies the shape of the unit area. The value and display options are available only for the flatness geometric tolerances. Select from the following display options for unit area tolerance:

Option	Resulting display
	
	
	

   Note:

   - The default value of the **Display** box is the dxd.
   - The size of the square and diameter symbols is determined by the text height in the geometric tolerance.

7. To assign a material condition symbol, select from the **Material Condition** list:
   - **LMC**—Least material condition
   - **MMC**—Maximum material condition

- **RFS(with symbol)**—Regardless of feature size
- **RFS(no symbol)**—Regardless of feature size, but does not show a symbol in frame

8. Click **OK** to close the **Geometric Tolerance** dialog box.

To Designate Symbols for Geometric Tolerances

Complete the following to specify the geometric tolerance's symbols and modifiers, as well as the projected tolerance zone.

1. To designate a symbol for the geometric tolerance, click the **Symbols** tab in the **Geometric Tolerance** dialog box.

   Note: If the **Geometric Tolerance** dialog box is not open, click **Insert** > **Geometric Tolerance**.

2. Specify the symbols and modifiers by selecting the appropriate check boxes under **Symbols and Modifiers**:
   - **ASME Y 14.41 Style**—This option is available for selection only when you select the Surface Profile geometric tolerance.
   - **Statistical Tolerance**
   - **Diameter Symbol**
   - **Free State**
   - **All Around Symbol**
   - **Tangent Plane**
   - **Set Boundary**

3. For geometric tolerances of types Angularity, Perpendicularity, Position, and Parallelism, to establish a projected tolerance zone, select the appropriate option under **Projected Tolerance Zone**:
   - **None**—Do not specify a projected tolerance zone.
   - **Below Gtol**—Indicate the projected tolerance zone without specifying the height, placing it on a separate line below the geometric tolerance.
   - **Inside Gtol**—Indicate the projected tolerance zone without specifying the height, placing it on the same line next to the geometric tolerance.

   Note: You can also specify the height of the projected tolerance zone by selecting **Zone Height**, and typing a value in the box.

4. For a Line Profile or a Surface Profile geometric tolerance, to define a unilateral or bilateral profile boundary, select the appropriate option under **Profile Boundary**:
   - **Unilateral**—Specify a unilateral direction for the geometric tolerance. The tolerance disposition is in the outward or inward direction of the selected profile. By default, the initial tolerance disposition is applied in the outward

direction of the selected profile. **Flip side** allows you to change the profile direction. If you select the **ASME Y 14.41** from the **Symbols and Modifiers**, the **Unilateral** profile boundary is not available for selection.

- **Bilateral**—Specify a bilateral direction for the geometric tolerance. The tolerance disposition is divided evenly about the selected profile.

Alternatively, for a Surface Profile geometric tolerance, if you have selected the **ASME Y 14.41 Style** option, you can also select an unequal disposition for the geometric tolerance.

- **Unequal**—Type a value in the **Value** box under **Unequal** to specify an unequal deposition value for the geometric tolerance. This value represents the portion of the overall tolerance value that you want to apply outward of the selected profile. If you have specified a positive value, the tolerance zone is offset outward from the selected surface. If you have specified a negative value, the tolerance zone is offset inward of the selected surface. The gtol annotation displays the tolerance value, the unequal symbol, and the unequal disposition value.

Note: The gtol annotation displays the unequal symbol only when you set the `gtol_display_style` drawing setup file option to `asme_y1441`.

To Attach Additional Text to a Geometric Tolerance

1. Insert a geometric tolerance using the **Geometric Tolerance** dialog box.

   To select an existing geometric tolerance with or without additional text, select the geometric tolerance, right-click, and click **Properties** on the shortcut menu to open the **Geometric Tolerance** dialog box. Click the **Additional Text** tab to edit the additional text.

   Click the Additional Text tab to edit the additional text.

2. Attach or edit additional text by clicking the appropriate check boxes:

   - **Additional text above**—Click this check box to insert text and text symbols, if required, in the input panel. The text and text symbols are inserted above the geometric tolerance control frame.
   - **Additional text on right**—Click this check box to insert text and text symbols, if required, in the input panel. The text and text symbols are inserted to the right of the geometric tolerance control frame.
   - Click the **Prefix** and **Suffix** check boxes to insert text and text symbols, if required, in the input panel. The text and text symbols are inserted as a prefix or suffix to the geometric tolerance text.

     Note: The prefix and suffix text will have the same text style as the geometric tolerance text. If the geometric tolerance has a material condition associated with it, the prefix text is placed before the tolerance value while the suffix text is placed after the material condition.

About Creating Geometric Tolerances in Assembly Drawings

In assembly drawings you can insert geometric tolerances in the top-level assembly, the subassembly, or in a part.

When you create a geometric tolerance in the top-level model (that is, a part in a part drawing or top assembly in an assembly drawing) the system associates the tolerance with the view in which you have selected a reference entity. Reference datums that you specify for the geometric tolerance must belong to the same top-level model; however, you can select them in any view. You cannot reference draft entities and draft datums.

Geometric tolerances applied to draft entities (draft geometric tolerances) can reference datums of the model. You can attach an assembly geometric tolerance to a dimension, datum, or another geometric tolerance, provided they both belong to the same assembly.

To Insert a Geometric Tolerance in an Assembly Drawing

1. Click **Insert** > **Geometric Tolerance**. The **Geometric Tolerance** dialog box opens.
2. Define the type of geometric tolerance to insert.
3. On the **Model Refs** tab, specify the assembly in the **Model** list. Continue inserting the geometric tolerance in the assembly drawing as you would in any other drawing.

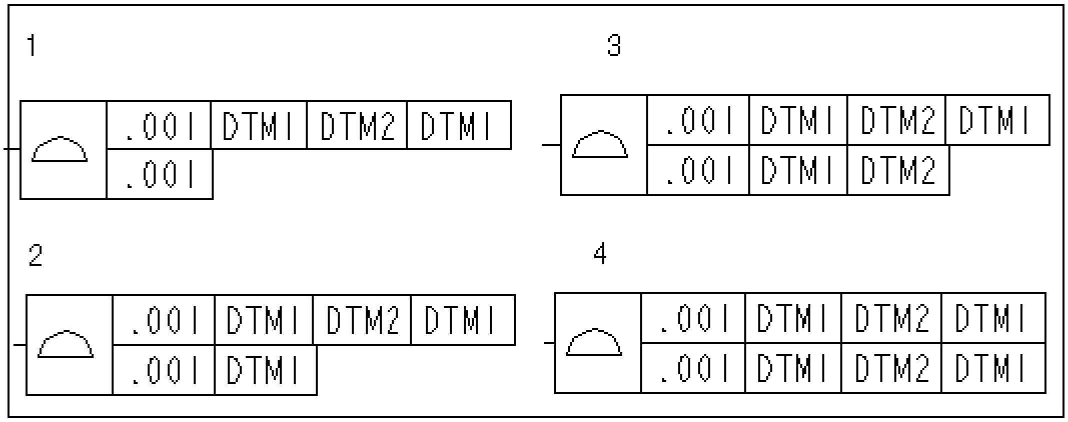
Note: When creating a geometric tolerance in an assembly drawing, if you set the `draw_models_read_only` configuration option to `yes`, Pro/ENGINEER sets the **Drawing** command as the default in the **Model** list in the **Model Refs** tab of the **Geometric Tolerance** dialog box. The **Part** and **Assembly** commands are not available for selection; if you choose them, an error message appears. If you set the `draw_models_read_only` configuration option to `no` and if there are no assembly views on the current page, then the **Part** command is set as the default in the **Model** list in the **Model Refs** tab of the **Geometric Tolerance** dialog box. If an assembly view is present, then **Assembly** is set as the default.

Example: Geometric Tolerance Classes and Types

Class	Type	Symbol	Reference Entity
Form	Straightness	—	Surface of revolution, axis, straight edge
	Flatness	⏥	Plane surface (not datum plane)
	Circularity	○	Cylinder, cone, sphere

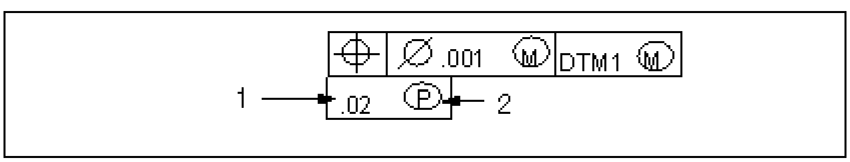
Class	**Type**	**Symbol**	**Reference Entity**
	Cylindricity	⌭	Cylindrical surface
Profile	Line	⌒	Edge
	Surface	⌓	Surface (not datum plane)
Orientation	Angularity	∠	Plane, surface, axis
	Parallelism	//	Cylindrical, surface, axis
	Perpendicularity	⊥	Planar surface
Location	Position	⌖	Any
	Concentricity	◎	Axis, surface of revolution
	Symmetry	⌯	Any
Runout	Circular	↗	Cone, cylinder, sphere, plane
	Total	⌰	Cone, cylinder, sphere, plane

Example: Datum References for a Composite Tolerance



1. No datum reference.
2. Primary datum reference.
3. Secondary datum reference.
4. Tertiary datum reference.

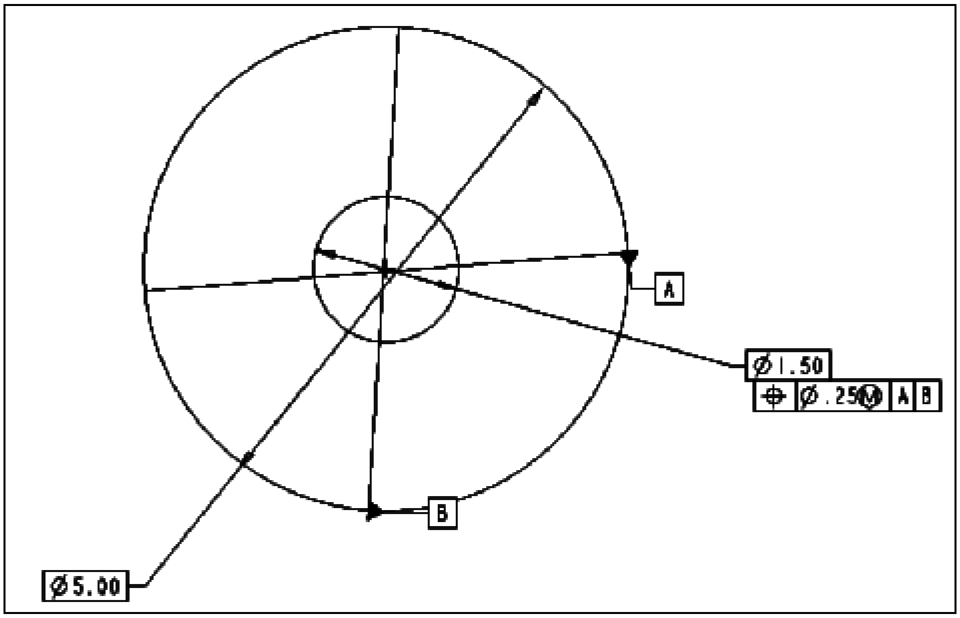
Example: Projected Tolerance Zone



1. The projected tolerance zone height value.
2. The projected tolerance zone height symbol.

Example: Adding a Geometric Tolerance to a Drawing

Following the procedure below, use the characteristics and values given in this table to create a sample geometric tolerance.



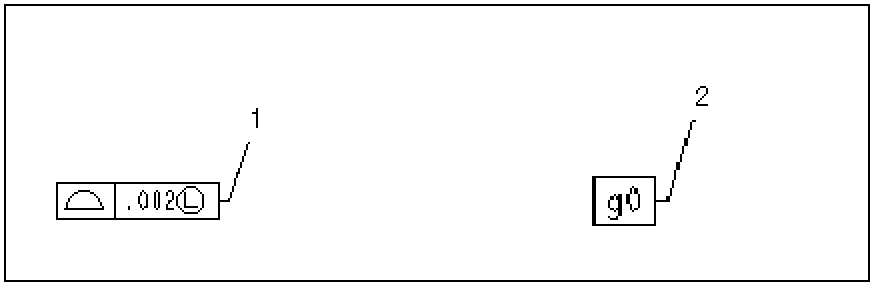
Sample geometric tolerance Characteristics and Values

Characteristic	**Value**
Entity	Hole
Tolerance Location	As part of a diameter dimension
Class and Type	Location/Position (a "true position" tolerance)
Overall Tolerance Value	0.25
Material Condition	MMC
Primary Datum (A) Material Condition	RFS/Default
Secondary Datum (B) Material Condition	RFS/Default

Tip: Adding Geometric Tolerances to Notes

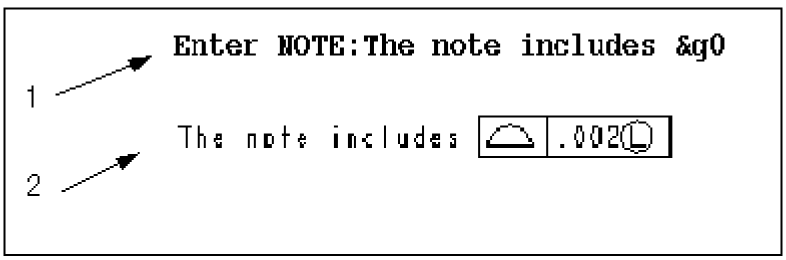
Every geometric tolerance symbol on a drawing has a symbolic representation. You can include a geometric tolerance in a note as a parameter by typing its symbolic value in the format [&g#], where "#" is the number indicating the order in which the geometric tolerance was created.

Example: Symbolic Representation



1. Geometric tolerance symbol.
2. Switched to symbolic format.

Example: Note Creation



1. Type a note at the system prompt.
2. This note appears on the screen.

Modifying Geometric Tolerances

To Modify the Material Condition

1. Select a geometric tolerance.
2. Click **Edit > Properties**. The **Geometric Tolerances** dialog box opens.
3. Click the **Tol Value** tab.
4. Select a material condition from the **Material Condition** list. The system updates the geometric tolerance on the screen.
5. Click **OK**.

To Modify a Datum Reference to a Geometric Tolerance

1. Select a geometric tolerance.
2. Click **Edit** > **Properties**. The **Geometric Tolerances** dialog box opens.
3. Click the **Datum Refs** tab.

4. Do one of the following:
   - Add a new datum reference by converting an existing simple datum reference into a compound reference feature*.* Add a secondary or tertiary datum reference clicking either **Secondary** or **Tertiary**. Select a datum reference from the **Compound** list. Specify a material condition, if needed.
   - Replace a datum reference with another datum*.* Click **Primary**, **Secondary,** or **Tertiary** (whichever is appropriate) in the **Datum References** box; then select a different datum name from the **Basic** or **Compound** list. If the datum that you want to use is not on the list, click **Select...** to select it in the drawing.
   - Remove a datum reference. Click **Primary**, **Secondary,** or **Tertiary** in the **Datum References** box (whichever is appropriate); then select **None** from the **Basic** or **Compound** list.
5. Click **OK**.

To Delete Geometric Tolerances

You can erase or delete geometric tolerances that are shown in drawings. If you delete a shown geometric tolerance it is deleted in both the drawing and the model. However, a geometric tolerance remains intact in a note even if you delete it from the drawing. If you delete it from Part mode, the system replaces it in the note with "***".

You can delete geometric tolerances in the following ways:

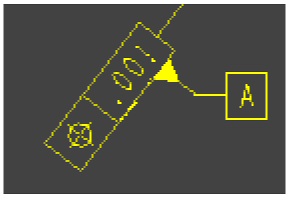
- Right-click the geometric tolerance and click **Delete** on the shortcut menu. You are prompted that deleting a shown geometric tolerance in the drawing also deletes the geometric tolerance in the model. To delete the geometric tolerance, click **Yes**.
- Select the geometric tolerance to delete and click **Edit** > **Delete**. You are prompted that deleting a shown geometric tolerance in the drawing also deletes the geometric tolerance in the model. To delete the geometric tolerance, click **Yes**.

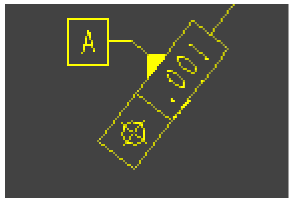
Note: You can undo or redo the deletion of geometric tolerances.

To Flip Set Datum Attachment Side

You can manually move a model set datum that is attached to either a driving dimension or model geometric tolerance; or you can also move a draft set datum that is attached to a draft geometric tolerance, along the geometric tolerance control frame. You can flip the set datum to the opposite side of the geometric tolerance control frame for single geometric tolerances, as shown in the following figure.

Manually Flipped Set Datum





In case of a stack of geometric tolerances, you can flip the set datum from the geometric tolerance at the bottom of the stack to the geometric tolerance at the top, or from top to bottom.

To flip the set datum attachment side, do the following:

1. Select the attached set datum text box you want to move. The set datum, along with the handle points, is highlighted.
2. Drag the selected set datum text box vertically towards to the new location for the set datum (top or bottom).The datum triangle flips direction and is reattached to the control frame.

Note:

- If, after flipping a set datum attachment side, you change its display standard, for example, from ISO/JIS to ANSI, the set datum is displayed in the ANSI standard. If you switch to the previous standard again, the modified display information is saved in memory and you do not have to modify the attachment location again.
- Drag handles exist on set datums that are attached to geometric tolerances. This allows the set datum to flip.

Working with Reference Datums

About Using Set References

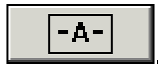
Before you can reference a datum plane or axis in a geometric tolerance or on geometry, you must set it. When you set a datum, its name is enclosed in a rectangle. Once you have set a datum, you can still use it in the usual way to create features and assemble parts.

When displaying set datum planes, keep the following in mind:

- A set datum plane, attached to a dimension, appears in a drawing only when you show the dimension to which it is associated.
- Reference datum planes do not display in a drawing unless they are perpendicular to the screen.

To Set a Datum in Drawing

When you set a datum, you can also choose to attach it to a dimension.

1. Select the name of the datum you want to set.
2. Click **Edit** > **Properties**. The **Datum** dialog box opens.
3. In the **Type** field, click 
.
4. Use the **Placement** field to specify whether the datum is free or part of a selected dimension.
5. Click **OK**.

To Place a Set Datum in a Dimension

1. Select the name of the datum you want to place.
2. Click **Edit** > **Properties**. The **Datum** dialog box opens.
3. Use the **Placement** field to specify whether the datum is free or part of a selected dimension.
4. Click **OK**.

Note: If you set the drawing setup file option `gtol_datums` to `STD_ASME` or `STD_ISO`, you can also place a set datum on a geometric tolerance. Click **In Gtol** and **Pick Gtol** in the **Datum** dialog box; then select a geometric tolerance in the model.

To Graphically Specify the Attachment Location of Set Datum Tags in Geometric Tolerances

In the drawing mode, you can specify the attachment point for a model or draft set datum tag on a geometric tolerance by graphically selecting a point on the geometric tolerance in the drawing view. The attachment point of the set datum tag can be top half or bottom half of the geometric tolerance.

1. Select the set datum plane or axis.
2. Double-click the set datum or click **Edit** > **Properties**. If you have selected a set datum plane, the **Datum** dialog box opens. If you have selected a set datum axis, the **Axis** dialog box opens.
3. Under **Placement**, select **In Gtol**.
4. To place the set datum on top of the geometric tolerance, select the top half of the geometric tolerance in the drawing view. You can also select the top half of a geometric tolerance stack. The set datum tag is attached to the center of the top edge of the selected geometric tolerance or geometric tolerance stack.

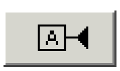
   Alternatively, place the set datum at the bottom of a geometric tolerance or a geometric tolerance stack, by selecting the bottom half of the geometric tolerance, or geometric tolerance stack in the drawing. The set datum tag is attached to the center of the bottom edge of the selected geometric tolerance or geometric tolerance stack.

Note:

- **In Gtol** is available only if you have set the gtol_datums drawing setup file option to std_asme, std_iso, or std_jis.
- You can graphically specify the placement of the model or the draft set datums in geometric tolerances in the drawing mode only.
- The point that you select graphically overrides the value specified by the gtol_datum_placement_default drawing setup file option. For example, if the value of the gtol_datum_placement_default is on_bottom, and you graphically select the top half of the geometric tolerance or geometric tolerance stack, then the set datum tag is attached to the center of the top edge of the selected geometric tolerance or geometric tolerance stack.
- The gtol_datum_placement_default drawing setup file option determines the attachment location of the set datum tag when you select the geometric tolerance using non-graphical means such as the Model Tree or the **Search Tool** dialog box (**Edit** > **Find**).

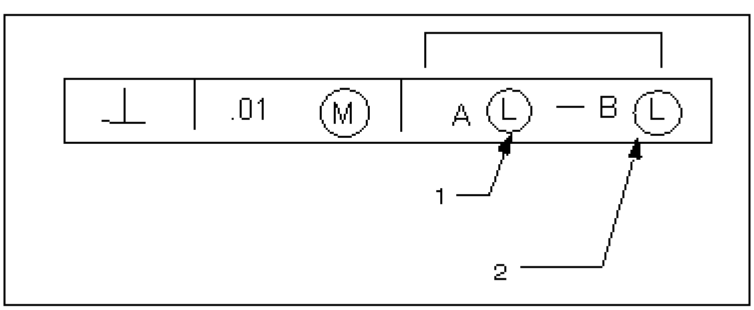
  Note: You cannot select a geometric tolerance stack on the Model Tree or by using the **Search Tool** dialog box.

To Create a Reference Datum Attached to a Cylindrical Surface

1. Set the gtol_datums drawing setup file option to std_iso.
2. Select the axis of the cylindrical surface, right-click, and click **Properties**. The **Axis** dialog box opens.
3. Click 

in the **Axis** dialog box and click **On Geometry** in the **Placement** area.
4. Select the cylindrical surface to which you want to attach the reference datum. The reference datum snaps to the cylindrical surface at the point that you selected on the cylindrical surface.
5. Click **OK**.

Example: Geometric Tolerance Symbol with a Compound Datum



1. First datum feature.

2. Second datum feature.

About the Behavior of Set Datum Tags Created in 3D Mode

Displaying Set Datum Tags Created in 3D Mode

When a set datum tag annotation is defined in the model and then displayed in the drawing, the position and appearance of the set datum tag in the model are retained in the drawing.

Set the `auto_show_3d_detail_items` configuration option to `yes` to display the set datum tag annotations from the 3D model automatically in the drawing. You can also display the set datum tag annotations from the 3D model in the drawing view by using the **Show and Erase** dialog box. The display of a standalone set datum tag annotation or annotation elements in the model overrides any configuration options set in the drawing. For example, if in the model, the set datum tag annotation element is displayed as attached to the model by the edge of a flat surface with the plunger symbol, and in the drawing mode the `gtol_datums` drawing setup file option is set to `std_ansi`, then in the drawing view the set datum tag appears as it was created in the model, even though it does not follow the ANSI standard.

If in a drawing, the set datum tag annotation is displayed in such a way that it cannot be displayed in the model, then the datum tag annotation is not displayed in the solid model, but only on the Model Tree. For example, if the set datum tag is attached to a created dimension or to a geometric tolerance in a drawing that is not visible in the solid model, then it is not displayed in the model, but only on the Model Tree.

Note: You can display the set datum tag annotations in multiple views in a drawing view, and with multiple attachments.

Modifying Set Datum Tags Created in 3D Mode

In a drawing, you can modify the set datum annotation elements created in the 3D mode, but you cannot unset the set datum tag annotation elements. However, you can unset or modify standalone set datum tag annotations created in the 3D mode.

When creating a set datum tag in a drawing, you cannot create a set datum tag similar to the set datum tag annotation in the 3D mode. Similarly, when editing or modifying the set datum tag from the **Datum** or **Axis** dialog box in a drawing view, you cannot create the set datum tag annotation as in the 3D mode. However, in the drawing mode, you can modify a standalone set datum tag annotation that was created in the 3D mode. When modifying a standalone set datum tag annotation that was created in 3D mode, the 3D set datum tag annotation is available for the selected datum in the **Datum** or the **Axis** dialog box.

> **Note**: If a set datum tag annotation of a model is created using the **On Geometry** option in the **Datum** or the **Axis** dialog box, then when editing the set datum tag in the drawing mode, the **On Datum** option is not available. Similarly, if the set datum tag annotation of a model has not been defined using the **On Geometry** option, then when editing the set datum tag annotation in the drawing mode, the **On Geometry** option is not available.

Note:

- Extension lines created in drawings for set datum tags are not updated in the solid model. Extension lines are not supported for annotations in the 3D mode.
- When a set datum tag annotation changes its reference, the tags previously displayed in the drawing for the old datum continue to reference the old datum, and revert to ASME tags if the `gtol_datums` drawing setup file option is set to `std_asme`. If the datum for the new reference is already displayed in the drawing, that is, when the selected reference is a datum, then the tags in the drawing appear correctly. Otherwise, you must use the **Show/Erase** dialog box to display the tags for the new reference.
- Set datum tags annotations in drawings reflect the correct reference of the solid model. If the reference of the set datum tag annotation is changed, then the drawing view does not show the set datum tag that points to an invalid reference. However, attachment points and the text position of the set datum tags are not updated between the drawing and the corresponding model.
- Draft datums do not have the ability to be set to 3D set datum tag annotations.

Datum Planes

About Working with Model Datum Planes

Pro/ENGINEER shows datum planes in the drawing under these conditions:

- Temporarily, when you are orienting the model while creating a drawing view (this is the *only* time it displays a datum plane that is not perpendicular to the screen).
- When the datum is a draft datum. When you have selected **Display Datum Planes** in the **Environment** dialog box (or when you have selected **Planes** in the **Datum Display** toolbar) and the datum planes are perpendicular to the screen.
- When you have set the datum plane as a reference datum and it is perpendicular to the screen.

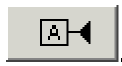
To manipulate set datums, use **Edit** > **Properties** on the menu bar. Set datums appear regardless of the setting in the **Environment** or **Datum Display** dialog boxes. You can erase them from a view by choosing **View** > **Show and Erase** on the menu bar, or by blanking them individually or on a layer.

You can also use shortcut menus (accessed by clicking the right mouse button on an object) to modify and manipulate 3-D set datums in the following ways:

- Move them to a different location on the drawing sheet.
- Redefine them.
- Erase and unerase them.

To Show Datum Planes

If you set a reference datum plane within the model it automatically displays in drawing views where the reference datums are perpendicular to the screen (on edge). If you erase a set datum plane, use this procedure to show it again.

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.
3. Under **Type**, click .
4. Use the following options in the Options tab to filter the set reference datum planes you want to show in the drawing:
   - **Erased**—Show previously erased items.
   - **Never Shown**—Show items that are yet to be displayed in the drawing.
5. Under **Show By**, use the following options to define where you want to display the set datum planes:
   - **Feature**—Show the reference datum planes for a particular feature on the drawing.
   - **Part**—Show the reference datum planes for a particular part on the drawing.
   - **View**—Show all the reference datum planes for the features and parts within a particular drawing view.
   - **Feature and View**—Show the reference datum planes for a feature that appears in several views in one selected view. For example, if you have a feature with a datum plane that appears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to display it.
   - **Part and View**—Show the reference datum planes for a part that appears in several views in one selected view.
   - **Show All**—Show all the reference datum planes within the drawing.
6. Select the appropriate feature, part, or view to display the reference datum planes. You may need to repaint the drawing for the datum planes to be displayed. When the reference datum planes are displayed the **Preview** tab becomes available.
7. Of the reference datum planes previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:
   - **Sel to Keep**—Select the individual set datum planes to show in the drawing. Any reference datum planes not selected will be erased.
   - **Sel to Remove**—Select set datum planes to remove from the drawing. Any reference datum planes not selected will remain in the drawing.

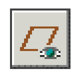
- ○ **Accept All**—Keep all the previewed set datum planes.
- ○ **Erase All**—Erase all the previewed set datum planes.

While you are showing the reference datum planes, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and **Resume Show and Erase**.

If you need to select multiple reference datum planes to keep or remove, you can do this by either holding down the `CTRL` key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.

8. When you are finished with the preview, click **OK** in the **Select** dialog box. The datum planes are displayed in the drawing.
9. Click **Close** to close the **Show/Erase** dialog box.

Note:

- You can display non-set datum planes by clicking  on the Datum Display toolbar in the Pro/ENGINEER graphics window or by selecting **Planes** in the **Datum Display** dialog box that opens when you click **View** > **Display Settings** > **Datum Display**.
- The `show_preview_default` drawing setup file option lets you set the preview tab default to either select to keep or select to remove.

To Insert a Model Datum Plane

In Drawing mode, you can create datum plane features in a model.

1. Click **Insert** > **Model Datum** > **Plane**.
2. In the **Datum** dialog box, type a name in the **Name** box. If you do not specify a name, the system assigns the name as DTM#.
3. Do one of the following:
   - ○ Click **Define**, and then constrain the datum by choosing commands from the **DATUM PLANE** menu; then click **Done**.
   - ○ Click **On Surface** to place the datum plane feature on a planar model surface.
4. Under **Type**, choose the type of datum you want to create.
5. If you are creating a set datum, click **On Datum** or **In Dim** to specify its placement.

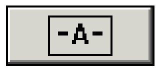
When the system has fully constrained the datum, it creates it; however, it displays the datum *only* if it is perpendicular to the screen and you have selected **Display Datum Planes** in the **Environment** dialog box (or if you have selected **Planes** in the **Datum Display** dialog box).

6. To continue creating datums, click **New**, type another name in the **Name** box, and click **Define...** or **On Surface...**.

To Set a Datum Plane of a Model in a Drawing

1. In the drawing view of a model, select the datum plane of the model and double-click. The **Datum** dialog box opens.

   Alternatively, select the datum plane of the model and click **Edit** > **Properties**. The **Datum** dialog box opens.

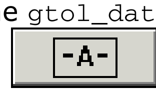
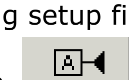
2. Under **Type**, click  to convert the datum plane of the model to a set datum plane.

3. To define the placement of the set datum tag, under **Placement** select one of the following:

   - **On Datum**—Attach the tag of the set datum plane of the model to itself. This is the default.
   - **In Dim**—Attach the tag of the set datum plane of the model in a dimension.
   - **On Geometry**—Attach the tag of the set datum plane of the model to a geometry such as a model arc or a model circle.

   Alternatively, if you set the `gtol_datums` drawing setup file option to `std_iso`, `std_jis`, or `std_asme`, you can also select a geometric tolerance as an attachment option for the set datum tag of the model.

   - **In Gtol**—Attach the tag of the set datum plane of the model on a geometric tolerance.

   Note: When you set the `gtol_datums` drawing setup file option to `std_iso`, `std_jis`, or `std_asme`, changes to in the **Datum** dialog box.

4. Click **OK** to close the **Datum** dialog box.

Note:

- Click  to change the set datum plane of the model back to an unset datum plane.

To Rename a Model Datum

1. Select the datum you want to rename.
2. Click **Edit** > **Properties**. The **Datum** dialog box opens.

3. Type a new name in the **Name** box.

4. Click **OK** to save the name change and to close the dialog box.

To Modify the Display of a Set Datum Plane

You can lengthen or shorten set datum planes by moving or clipping them. Moving a datum plane moves either side of the datum. Clipping moves the end opposite to the datum name.

1. Select the datum you want to modify. The datum is highlighted and its handles display.

2. Do one of the following:

    - To move the datum, select anywhere in the drawing window and the datum text moves to a corresponding location along the datum. The name of the datum also flips to the side of the datum where you selected.

    - To clip the datum, select anywhere and the datum plane moves to a corresponding point along the datum. The datum name does not move.

To Create a Draft Datum Plane

1. On the Pro/ENGINEER menu bar, click **Insert** > **Draft Datum** > **Plane**.

2. Click to begin drawing the datum plane.

3. Extend the datum plane to the proper orientation and click to place the endpoint.

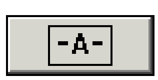
4. Type the name for the datum plane. Press Enter.

Note: To move or reorient draft datums, select a datum and then click anywhere in the drawing window to relocate or clip it. You can also click **Edit** > **Transform** > **Rotate** or **Translate**.

To Set a Draft Datum Plane

1. In a drawing view, select the draft datum plane and double-click. The **Datum** dialog box opens.

    Alternatively, select the draft datum plane and click **Edit** > **Properties**. The **Datum** dialog box opens.

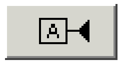
2. Under **Type**, click 

to change the draft datum plane to a draft set datum plane. Pro/ENGINEER encloses the plane in a feature control frame.

3. Click **OK** to close the **Datum** dialog box.

Alternatively, if you set the `gtol_datums` drawing setup file option to `std_asme`, `std_iso`, or `std_jis`, you can also select an attachment option for the draft set datum plane as follows:

1. Select the draft datum plane and double-click. The **Datum** dialog box opens.

2. Under **Type**, click  to convert the draft datum plane to a draft set datum plane.
3. Under **Placement**, select from the following options to attach the tag of the draft set datum plane to different objects:
   - **On Datum**—Attach the tag of the draft set datum plane to itself. This is the default option.
   - **In Gtol**—Attach the tag of the draft set datum plane on a geometric tolerance.
4. Click **OK.** The **Datum** dialog box closes and the set datum plane is placed at the selected location.

Note:

- You can control the initial length of the leader line of the draft set datum plane tag using the `leader_elbow_length` drawing file setup option.
- Click  to change the draft set datum plane back to an unset draft datum plane.

To Control the Size and Shape of Datum Points

To control the size of model datum points and sketched datum points in Drawing mode, use the following drawing setup file options:

- Specify a value for the drawing setup file option `datum_point_size` in the **Options** dialog box. To open the **Options** dialog box, click **File** > **Properties**, then click **Drawing Options** on the Menu Manager. Select the `datum_point_size` drawing setup file option and change its value. The default size of datum points is `.3125`.
- To display draft or datum points as a cross, dot, circle, triangle, or square, specify a value for the drawing setup file option `datum_point_shape`.

Note: These drawing setup file options control the display of points in Drawing mode only. To modify the datum point symbol display for Part or Assembly mode, set the configuration file option `datum_point_symbol`.

To Erase a Set Datum

1. On the Pro/ENGINEER menu bar, click **View** > **Show and Erase**.
2. In the **Show/Erase** dialog box, click **Erase**.
3. Click the **Datum Plane** button in the **Type** box; then select a button in the **Erase By** box to specify the feature, view, or part.
4. Select the item to erase.

To restore it, click **Show**, the **Datum Plane** button in the **Type** box, and a button in the **Show By** box to specify the feature, view, or part.

Erasing a Set Datum from a Member of an Assembly

If a drawing has several occurrences of the same part in an assembly, you can erase a set datum from a particular member of the assembly without causing the selected set datum to disappear from other occurrences of the member.

Note: You can erase only set datum planes from the drawing. To remove any other model datum planes (not draft datums) from display uncheck **Display Datum Planes** in the **Environment** dialog box, or uncheck **Planes** from the **Datum Display** dialog box (**View** > **Display Settings** > **Datum Display**).

Datum Axes

About Working with Model Axes

You can manipulate the axes of cylindrical or conical surfaces independently of the **Environment** dialog box settings. However, keep in mind the following:

- The system shows the axis as a line whenever you view it from the side, or as a cross hair whenever you orient the axis normal to the screen.
- In Drawing mode, if you select **Datum Axes** in the **Environment** dialog box, axis names appear on the drawing. You can control the axis name display independently of the axis using the **Axis Tags** check box in the **Datum Display** dialog box (**View** > **Display Settings** > **Datum Display**).
- You can create axes in the drawing, however, these axes belong to the drawing, not to the model.
- A draft axis always appears as a centerline.

By right-clicking a selected axis and using the shortcut menu, you can modify set and unset model axes in the following ways:

- Move them to a different location on the drawing sheet.
- Modify their attachment.
- Redefine them.
- Erase and unerase them.

To Show Datum Axes on a Drawing

When you show datum axes the axes of cylindrical and conical surfaces display on the drawing and erases axes that the system shows in a drawing. Because axes are inherited from the model you cannot delete them from within the drawing.

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.
3. Under **Type**, click .

4. Use the following options in the **Options** tab to filter the set datum axes you want to show in the drawing:
   - **Erased**—Show previously erased items.
   - **Never Shown**—Show items that are yet to be displayed in the drawing.
5. Under **Show By**, use the following options to define where you want to display the axes:
   - **Feature**—Show the set datum axes for a particular feature on the drawing.
   - **Part**—Show the set datum axes for a particular part on the drawing.
   - **View**—Show all the set datum axes for the features and parts within a particular drawing view.
   - **Feature and View**—Show the set datum axes for a feature that appears in several views in one selected view. For example, if you have a feature with a datum axis that appears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to show it.
   - **Part and View**—Show the set datum axes for a part that appears in several views in one selected view.
   - **Show All**—Show all the set datum axes within the drawing.
6. Select the appropriate feature, part, or view to display the reference datum planes. You may need to repaint the drawing for the set axes to be displayed. When the set axes are displayed the **Preview** tab becomes available.
7. Of the set axes previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:
   - **Sel to Keep**—Select the individual set datum axes to show in the drawing. Any set axes not selected will be erased.
   - **Sel to Remove**—Select set datum axes to remove from the drawing. Any set axes not selected will remain in the drawing.
   - **Accept All**—Keep all the previewed set datum axes.
   - **Erase All**—Erase all the previewed set datum axes.

   While you are showing the datum axes, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and **Resume Show and Erase**.

   If you need to select multiple datum axes to keep or remove, you can do this by either holding down the `CTRL` key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.

8. When you are finished with the preview, click **OK** in the **Select** dialog box. The set axes are displayed in the drawing.
9. Click **Close** to close the **Show/Erase** dialog box.

Note:

- You can display non-set datum axis using  on the toolbar in the Pro/ENGINEER graphics window.
- Showing or erasing an axis in a radial pattern affects corresponding axes in all pattern instances.
- The `show_preview_default` drawing setup file option lets you set the preview tab default to either select to keep or select to remove.

To Insert a Model Datum Axis

The system displays draft axes in leader style with centerline font.

1. On the Pro/ENGINEER menu bar, click **Insert** > **Model Datum** > **Axis**.
2. In the **Axis** dialog box, type a name in the **Name** box.
3. Click **Define**.
4. Constrain the datum axis by choosing commands from the **DATUM AXIS** menu; then choose **Done**.
5. To continue creating datum axes, click **New**, type another name in the **Name** box, and click **Define**.

To Create a Draft Axis

1. Click **Insert** > **Draft Datum** > **Axis**.
2. Click on the required location in the drawing to start drawing the axis.
3. Move the mouse pointer to extend the axis to the proper orientation (the axis line drags with the mouse). Place the endpoint by clicking again.
4. Type the name of the axis. The system displays the name.
5. To move or reorient the axis, you can move or clip it by selecting the axis and then clicking anywhere in the drawing window, or choose **Rotate** or **Translate** from the **Edit** > **Transform** menu.

To Rename a Model Datum Axis

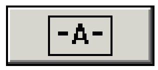
1. Select the datum axis you want to rename, right-click and select **Properties** from the shortcut menu.
2. In the **Axis** dialog box, type a new name in the **Name** box.
3. Click **OK** to close the dialog box. The axis is renamed.

Note: You cannot rename draft datum axes.

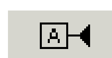
To Set a Draft Datum Axis

1. In a drawing view, select the draft datum axis and double-click. The **Axis** dialog box opens.

   Alternatively, select the draft datum axis and click **Edit** > **Properties**. The **Axis** dialog box opens.

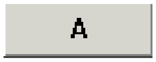
2. Under **Type**, click  to change the draft datum axis to a draft set datum axis. Pro/ENGINEER encloses the axis in a feature control frame.
3. Click **OK** to close the **Axis** dialog box.

Alternatively, if you set the `gtol_datums` drawing setup file option to `std_asme`, `std_iso`, or `std_jis`, you can also select an attachment option for the draft set datum axis as follows:

1. Select the draft datum axis and double-click. The **Axis** dialog box opens.
2. Under **Type**, click  to convert the draft datum axis to a draft set datum axis.
3. Under **Placement**, select from the following options to attach the tag of the draft set datum axis to different objects:
   - **On Datum**—Attach the tag of the draft set datum axis to itself. This is the default option.
   - **In Gtol**—Attach the tag of the draft set datum axis on a geometric tolerance.
   - **On Geometry—**Attach the tag of the draft set datum axis to a geometry such as a model arc, a model circle, a draft arc, or a draft circle.
4. Click **OK.** The **Axis** dialog box closes and the set datum axis is placed at the selected location.

Note:

- You can control the initial length of the leader line of the draft set datum axis tag using the `leader_elbow_length` drawing file setup option.
- When you change the attachment option of a draft set datum axis tag from **On Datum** to **In Gtol** or **On Geometry**, the original draft set datum axis disappears.
- When you change the attachment option of a draft set datum axis tag from **On Geometry** to **On Datum**, or from **In Gtol** to **On Datum** the original draft set datum axis reappears.
- You can control the display of the set datum triangles as filled triangles or empty triangles using the `set_datum_triangle_display` drawing setup file option.

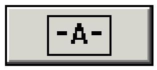
- Click 

to change the draft set datum axis back to an unset draft datum axis.

To Set a Datum Axis of a Model in a Drawing

1. In the drawing view of a model, select the datum axis of the model and double-click. The **Axis** dialog box opens.

   Alternatively, select the datum axis of the model and click **Edit** > **Properties**. The **Axis** dialog box opens.

2. Under **Type**, click 

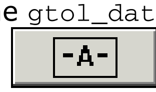
to convert the datum axis of the model to a set datum axis.

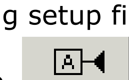
3. To define the placement of the set datum tag, under **Placement** select one of the following:

   - **On Datum**—Attach the tag of the set datum axis of the model to itself. This is the default.
   - **In Dim**—Attach the tag of the set datum axis of the model in a dimension.
   - **On Geometry**—Attach the tag of the set datum axis of the model to a geometry such as a model arc or a model circle.

   Alternatively, if you set the `gtol_datums` drawing setup file option to `std_iso`, `std_jis`, or `std_asme`, you can also select a geometric tolerance as an attachment option for the set datum tag of the model.

   - **In Gtol**—Attach the tag of the set datum axis of the model on a geometric tolerance.

   Note: When you set the `gtol_datums` drawing setup file option to `std_iso`, `std_jis`, or `std_asme`, 

changes to 

in the **Axis** dialog box.

4. Click **OK** to close the **Axis** dialog box.

Note:

- Click 

to change the set datum axis of the model back to an unset datum axis.

Axis Display Options

By setting the drawing setup file option `axis_interior_clipping` to `no`, you can set the axis display according to these three ANSI Y14.2M standard requirements:

- Line starts and ends with a dash.
- Perpendicular axes of a centerline intersect at a short dash.
- Very short centerlines are unbroken.

Axes that are set to CTRLFONT actually are shown as CTRLFONT_S_L or CTRLFONT_L_L, depending on their orientation. You can control them by setting line_style_length to CTRLFONT_S_L or CTRLFONT_L_L, not CTRLFONT. To control axes that are perpendicular to the view, set line_style_length to CTRLFONT_S_L; to control axes that are parallel to the view, set it to CTRLFONT_L_L.

This option affects the axis display in the following ways:

- Axes parallel to the screen appear in one line in a centerline font, beginning and ending with a long dash.
- Axes perpendicular to the screen appear in two lines, each of them drawn in two segments.
- Perpendicular lines intersect at a short dash.
- When any axis is too short, it appears in a solid line font.

Using the drawing setup file option radial_pattern_axis_circle, you can set the display mode for axes of rotation that are perpendicular to the screen in radial pattern features. When you set the option to yes, a circular shared axis appears, and the axis lines pass through the center of a rotational pattern. However, this only affects patterns that you create using the **Dim Pattern** command.

The following restrictions apply when you set this drawing setup file option to yes:

- You cannot move, clip, or modify the line style of a circular shared axis that appears once you choose the **Show** command button from the **Show/Erase** dialog box (accessed by clicking **View** > **Show and Erase** on the menu bar).
- In a clipped view, the system only displays portions of lines and circles that are inside a view boundary.
- You cannot use radial patterns of group features.
- You cannot show radial pattern axis circles for reference patterns.

To Modify the Line Style of a Model or Draft Axis

1. On the Pro/ENGINEER menu bar, click **Format > Line Styles**. The **LINE STYLES** menu opens in the Menu Manager.
2. Select the axis name or the axis itself (for model axis features). The **Line Style** dialog box opens.
3. In the **Line Style** dialog box, select a line style from the **Style** list.
4. Click **Apply**. The axis acquires the specified line style. To reset it to the old style, click **Reset** and **Apply**.

   Note: When you set the drawing setup file option axis_interior_clipping to no, you can still change the axis line style. However, this disables the automatic adjustment of a pattern.

To Create a Break in a Model Axis Line

1. In the Graphics window, select the axis line in which you want to create the break.
2. On the Pro/ENGINEER menu bar, click **Insert > Break**. The BREAK menu opens in the Menu Manager. **Add** is the default selection.
3. Select the first location on the axis line to begin the break.
4. Select the second location to finish the break. The system creates the break.

To Delete a Portion of a Normal-to-Screen Axis

1. Select an axis line, and then click **Edit** > **Delete**.
2. To resume the display of all portions of the axis, click **View > Show and Erase** The **Show/Erase** dialog box opens.
3. Click **Axis** in the **Type** box.
4. Click **Show All**. The deleted axis lines are redisplayed and highlighted in a different color. If **Preview** is selected in the dialog box, click **Accept All**. All axis lines are restored to the display.

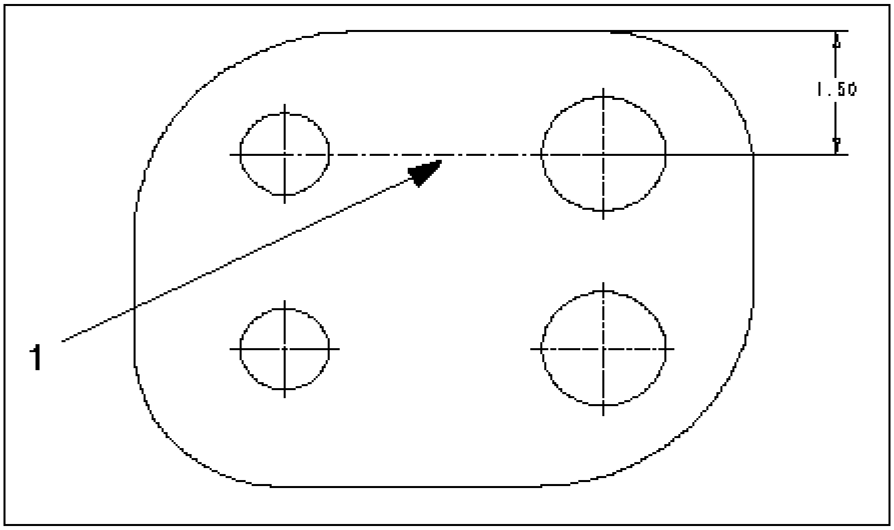
   Note: If you choose the normal-to-screen axis, the system erases all of the lines for that axis.

To Create an Axis Symmetry Line

Using the following procedure, you can create an extension between axis cross hairs showing that the axes are symmetrical about a dimension.

1. On the menu bar, click **Insert** > **Symmetry Line Axis**.
2. Select two axes that are perpendicular (normal) to the plane of the screen (shown as cross hairs). To extend the symmetry line to either side, grab a handle of the axis with the mouse pointer and click anywhere outside of the line.

Example: Axis Symmetry Lines

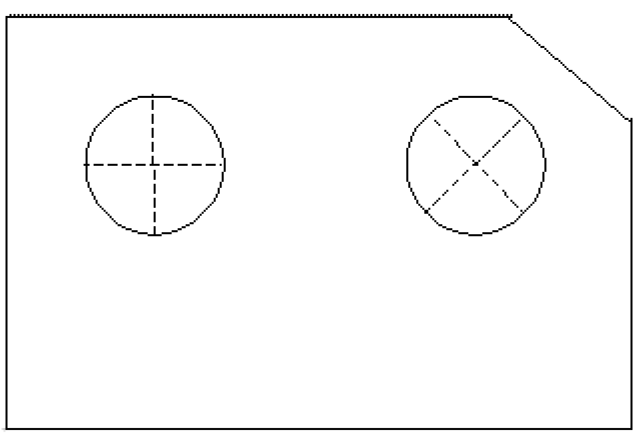


1. Axis symmetry line.

About Rotating Axes

Using the **Edit** > **Attachment** command you can rotate model axes that are normal to the screen. When you rotate the axis, it rotates according to another reference entity that you select, such as an edge or point, as shown in the following diagram. The rotated axis remains normal (perpendicular) to the screen; however, the cross reorients according to your specifications.

Rotating Axes



The **Rotate Axis** menu displays the following options:

- **Through Geom**—Picks an edge, datum point, or center of an axis normal to the screen through which the axis line that is closest to that point passes.
- **Pick Point**—Picks a point on the screen through which the axis line passes.

- **Parallel**—Picks a linear edge or datum curve to which the axis line is parallel.
- **Horizontal**—Returns the axis to its standard orientation.
- **Enter Angle**—Specifies an angle for axis rotation about the X-axis.

Symbols

About Symbols

Symbols are collections of draft geometry and text that either serve as a simple labeling object, or that represent more complicated objects such as assemblies or electrical components. The symbol may show parameters as text notes. When the instance is placed, the parameters read the values associated with the drawing.

If you redefine the symbols appearance, all of the instances in the drawing update to show the new appearance.

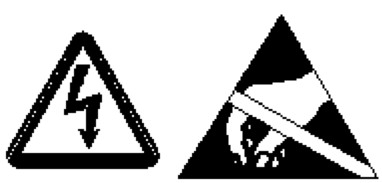
The Symbol Definition and the Symbol Instance

A symbol, when created, is added to its parent drawing's symbol gallery as a symbol *definition*. When you add the symbol to the drawing, it is added as a copy of the definition called a symbol *instance*.

To use the symbol in a different drawing you must save it as a `.sym` file. Pro/ENGINEER can set up a default symbols directory using the configuration option `pro_symbol_dir`.

Simple Vs. Generic Symbols

Symbols may be defined and stored as *simple* or *generic* symbols. Simple symbols have fixed graphic and text content, and each instance appears identical to the next. The symbols below are examples of simple symbols.



Above: Simple Symbols - Unchanging graphics and text

Generic symbols are composed of different *groups* of graphic elements saved within the definition. You can choose to include one or more of the groups whenever you place an instance. In this way you can create different looking symbols from one "generic" symbol definition.

Symbol Graphic and Placement Properties

A symbol definition is a collection of properties that determine its attachment leader characteristics, its graphic appearance and graphic groups if any, and its ability to read parameters from the drawing. When you place an instance, you can change the properties of the instance from those of the definition. For example, you can change the leader attachment style for one instance, or change which groups display in a generic instance, without redefining the style for the remainder of the instances on the sheet.

If scalability is enabled in the symbol definition, you can set a height for the instance. If there are parameters with pre set values as part of the symbol, you can select different values for different instances.

Symbols Passed from 3D Models

If symbols have been applied to an annotation plane in the 3D model, you can use the **Show/ Erase** dialog box to show or erase them, as you can with any dimension or object imported from 3D. When shown in 2D, the symbol appears in the view plane, regardless of the view orientation.

Symbol Storage and Placement Options

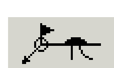
Pro/ENGINEER ships with a collection of commonly-used drawing symbols, stored under the *symbols* directory. You can browse the subdirectories of this area for any symbol type you need.

The **Symbol Gallery** (**Format** > **Symbol Gallery**) is a collection of symbol definitions that have instances within the drawing. Use the Symbol Gallery to define new symbols, or redefine existing symbols.

The **Symbol Palette** (**Insert** > **Drawing Symbol** > **From Palette**) is a `.drw` file dedicated to storing frequently used instances of symbols. It is a convenient way of identifying and placing common symbols.

The **Custom Drawing Symbol** dialog box (**Insert** > **Drawing Symbol** > **Custom**) lets you create custom instances from generic definitions. When you place the instance, you can specify the graphics that is shows or hides, the size or color, and the values of variable text the symbol may include.

To Show 3D Symbols in Drawings

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.
3. Under **Type**, click .
4. Use the following options in the **Options** tab to filter the symbols that you want to show in the drawing:
   - **Erased**—Show previously erased items.
   - **Never Shown**—Show items that are yet to be displayed in the drawing.

5. Under **Show By**, use the following options to define where you want to display the symbols:

    - **Feature**—Show the symbols for a particular feature on the drawing.
    - **Part**—Show the symbols for a particular part on the drawing.
    - **View**—Show all the symbols for the features and parts within a particular drawing view.
    - **Feature and View**—Show the symbols for a feature that appears in several views in one selected view. For example, if you have a feature with symbols that appears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to display it.
    - **Part and View**—Show the symbols for a part that appears in several views in one selected view.
    - **Show All**—Show all the symbols within the drawing.

6. Select the appropriate feature, part, or view to display the symbols. You may need to repaint the drawing for the 3D symbols to be displayed. When the 3D symbols are displayed the **Preview** tab becomes available.

7. Of the symbols previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:

    - **Sel to Keep**—Select the individual symbols to show in the drawing. Any symbols not selected will be erased.
    - **Sel to Remove**—Select symbols to remove from the drawing. Any symbols not selected will remain in the drawing.
    - **Accept All**—Keep all the previewed symbols.
    - **Erase All**—Erase all the previewed symbols.

    While you are showing the symbols, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and **Resume Show and Erase**.

    If you need to select multiple symbols to keep or remove, you can do this by either holding down `CTRL` key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.

8. When you are finished with the preview, click **OK** in the **Select** dialog box. The symbols are displayed in the drawing.

9. Click **Close** to close the **Show/Erase** dialog box.

Note: The `show_preview_default` drawing setup file option lets you set the preview tab default to either select to keep or select to remove.

To Insert a Symbol from the Symbol Palette

The symbol palette is a `.drw` file dedicated to storing frequently used symbol instances. These symbols are usually simple symbols, without graphic groups or variable text.

1. Click **Insert** > **Drawing Symbol** > **From Palette**. The **Symbol Instance Palette** opens. Commonly used symbols are arranged in two sections. The left section stores the symbols as free-placement types. The right section shows the same symbols as On-Entity placement types.
2. Place the desired symbol, in one of the following ways:

- To place a free-placement symbol,
    a. Click the symbol you want to place. The symbol is highlighted.
    b. Move the cursor off of the palette and onto the drawing. The symbol is attached to the cursor.
    c. Drag the cursor to the symbol location and left click to place it. The symbol is placed, and the instance remains on the cursor to let you place another one. To cancel placement, click the right mouse button.
- To place an **On Entity** symbol,
    a. Select the symbol to place. The symbol is highlighted, but it is not attached to the cursor.
    b. Click to select the attachment object in the drawing. The symbol is added, and you may continue to select attachment points for more symbols.
    c. Click **OK** in the **Select** dialog box to stop placing symbols.

To Insert a Custom Symbol

1. Click **Insert** > **Drawing Symbol** > **Custom**. The **Custom Drawing Symbol** dialog box opens.
2. In the **Symbol** name box, select the name of the symbol from the list. The list shows symbols that are associated with the drawing, if any. If the symbol you want has not been added to the drawing, browse to locate it in the symbol library, or click **New** to create a new symbol definition.
3. Under **Placement**, select the placement type of the symbol from the following options in the **Type** list:
    - **Free**—Symbol is placed freely at any location on the screen.
    - **On Entity**—Symbol is directly attached to a reference. You can select an edge, entity, dimension, dimension witness line, coordinate system, curve or another symbol entity as the reference.
    - **On Intersect**—Symbol is placed at the intersection of two references.
    - **At vertex**—Symbol snaps to the selected vertex.

This placement point is available for selection only when the selected symbol is defined with the **On Entity** attribute.

- **Normal to Entity**—Symbol is placed normal to a reference. You can select an edge, entity, dimension, dimension witness line, coordinate system, curve or another symbol entity as the reference.

  Note: You can place only a symbol that has **Normal to Entity** as one of its allowed placement types on the extension line of a chamfer dimension.

- **With Leaders**—Symbol is placed with a leader attached to a reference. The **New Leader** and **Arrow Head** lists are available for selection.
- **Tangent Ldr**—Symbol is placed with a leader that is tangent to a reference. The **New Leader** and **Arrow Head** lists are available for selection.
- **Normal Leader**—Symbol is placed with a leader that is normal to a reference. The **New Leader** and **Arrow Head** lists are available for selection.
- **Offset**—Symbol is placed offset from a reference.
- **Absolute Coordinates**—Symbol is placed at the specified location that is defined as X and Y coordinates with reference to the drawing origin.

  Note: Absolute Coordinates is available only when the selected symbol is defined with the **Free** attribute.

- **Relative Coordinates**—Symbol is placed at the specified location defined by the relative X and Y offsets with reference to the last selected point in the drawing. This option is not available if a last selected point does not exist.

  Note: Relative Coordinates is available only when the selected symbol is defined with the **Free** attribute.

4. Under **Properties**, define the height, angle and color of the symbol.
5. Under **Origin**, **Default** uses the origin of the symbol as defined in the symbol definition for the symbol placement. To specify a custom origin for the symbol instance, click **Custom**.
6. Use the **Grouping** tab to specify which groups will show in the instance.
7. Use the **Variable Text** tab to select the content of notes appearing with the symbol.
8. Move the pointer on the drawing sheet. The symbol is attached to the pointer.
9. Click to place the symbol at a required location. You can continue placing the instance of that symbol at other locations. You can also select another symbol and place it at a required location.
10. When you have finished placing the symbols, click **OK** to close the **Custom Drawing Symbol** dialog box.

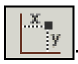
To Move a Symbol

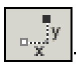
You can move a symbol by selecting it and dragging it to the required location or by using the **Move Special** dialog box.

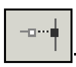
1. Select the symbol that you want to move, right-click, and click **Move Special** on the shortcut menu. The **Move Special** dialog box opens.

   Alternatively, select the symbol that you want to move and click **Edit** > **Move Special**. The **Select** dialog box opens. Select the symbol you want to move. The **Move Special** dialog box opens. The point that you select on the symbol acts as a origin to calculate the X and Y coordinates.

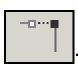
2. Select from the following to move the symbol:

   - —Move object to location defined as X and Y coordinates. Type the required values in the X and Y boxes.

   - —Move object to location defined by relative X and Y offsets. Type the required values in the X and Y boxes.

   - —Snap object to a specific reference point on an entity.

     Valid entity selections are highlighted when you move the pointer in the graphics window..

   - —Snap object to a specific vertex.

     Valid vertices, including intersections, are highlighted when you move the pointer in the graphics window.

3. Click **OK**. The symbol is moved to the specified location.

Note: You can move any drawing entity using the **Move Special** dialog box.

To Show Surface Finish Symbols

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.

2. Click **Show**.

3. Under **Type**, click .

4. Use the following options in the **Options** tab to filter the surface finish symbols that you want to show in the drawing:

   - **Erased**—Show previously erased items.

   - **Never Shown**—Show items that are yet to be displayed in the drawing.

5. Under **Show By**, use the following options to define where you want to display the surface finish symbols:

- **Feature**—Show the surface finish symbols for a particular feature on the drawing.
- **Part**—Show the surface finish symbols for a particular part on the drawing.
- **View**—Show all the surface finish symbols for the features and parts within a particular drawing view.
- **Feature and View**—Show the surface finish symbols for a feature that appears in several views in one selected view. For example, if you have a feature with surface finish symbols that appears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to display it.
- **Part and View**—Show the surface finish symbols for a part that appears in several views in one selected view.
- **Show All**—Show all the surface finish symbols within the drawing.

6. Select the appropriate feature, part, or view to display the surface finish symbols. You may need to repaint the drawing for the surface finish symbols to be displayed. When the surface finish symbols are displayed the **Preview** tab becomes available.
7. Of the surface finish symbols previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:
   - **Sel to Keep**—Select the individual surface finish symbols to show in the drawing. Any surface finish symbols not selected will be erased.
   - **Sel to Remove**—Select surface finish symbols to remove from the drawing. Any surface finish symbols not selected will remain in the drawing.
   - **Accept All**—Keep all the previewed surface finish symbols.
   - **Erase All**—Erase all the previewed surface finish symbols.

   While you are showing the surface finish symbols, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and **Resume Show and Erase**.

   If you need to select multiple surface finish symbols to keep or remove, you can do this by either holding down the `CTRL` key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.
8. When you are finished with the preview, click **OK** in the **Select** dialog box. The surface finish symbols are displayed in the drawing.
9. Click **Close** to close the **Show/Erase** dialog box.

Note: The `show_preview_default` drawing setup file option lets you set the preview tab default to either select to keep or select to remove.

Defining Symbols

About Defining Symbols

Use the **Redefine** command in the **Format > Symbol Gallery** menu to add, move, delete, or change detail items composing a symbol.

You can modify a symbol by doing the following:

- Create new entities composing a generic symbol.
- Modify existing entities composing a generic symbol.
- Change the attachment style.
- Change group definition by adding or removing entities from groups, or deleting one or all groups.

Effect of Symbol Redefinition on Instances

When you redefine a symbol, it affects the display of all subsequent instances and all symbol instances that you have added to a drawing using the **By Reference** command.

To Insert Surface Finish Symbols

1. Click **Insert** > **Surface Finish**. The **GET SYMBOL** menu appears.
2. Click **Name**. The **SYMBOL NAMES** menu appears, listing all the symbols that are currently in the drawing. If symbols have the same names, but are stored in different symbol directories, Pro/ENGINEER identifies them in the **SYMBOL NAMES** menu by a number within parentheses (#).
3. Click the symbol you want. The **INST ATTACH** menu appears.
4. Select from the following attachment commands on the **INST ATTACH** menu:
    - **Leader**—Attach an instance using leaders. Select from the available commands on the **ATTACH TYPE** menu and the **GET POINT** menu to place the symbol. When prompted, type a surface finish value between 0.001 and 2000. Middle-click to complete the placement of the symbol.
    - **Entity—**Attach an instance to edge, or entity. Select an entity to place the symbol. When prompted, type a surface finish value between 0.001 and 2000 and middle-click to complete the placement of the symbol.
    - **Normal**—Attach an instance normal to edge, or entity. Select an entity to place the symbol. Use the **DIRECTION** menu if you want to flip the direction of the symbol. Middle-click to complete the placement of the symbol.

- **No Leader**—Place an instance without leaders and unattached to geometry. Use the **GET POINT** menu to place the symbol at a location you want. Click **Quit** on the **GET POINT** menu to complete the placement of the symbol.
- **Offset**—Place an instance without leaders relative to an entity. Select a draft entity followed by the location where you want to place the symbol. Middle-click to complete the placement of the symbol.

5. Click **OK** in the **Select** dialog box to return to the **INST ATTACH** menu to continue placing surface finish symbols.

   You can also place surface finish symbols using one of the following methods:

   - Click **Pick Inst** on the **GET SYMBOL** menu to select any instance of a symbol available in the drawing. The **INST ATTACH** menu appears. Repeat step 4 to insert the symbol.
   - Click **Retrieve** on the **GET SYMBOL** menu to retrieve a symbol that is stored locally. Browse to the directory from where you want to retrieve the symbol, select the symbol, and click **Open**. The **INST ATTACH** menu appears. Repeat step 4 to insert the symbol.

6. Click **Done/Return** after you have finished placing the surface finish symbols.

Setting the Symbol Directory

About the System Symbols Area

The symbols directory contains libraries of Pro/ENGINEER symbols that are available with the Detailed Drawings module. This area is usually *read-only*.

The Welding Symbols Library, for example, provides a collection of generic system symbols according to the ANSI standard, and a collection of symbols according to the ISO standard. Using this library, you can create a variety of welding, brazing, and examination symbols in a drawing. Before you create an instance, familiarize yourself with the procedure for adding custom instances.

To Set the User-Defined Symbols Area

Pro/ENGINEER is set up to store and access symbols in two different areas:

- *user-defined symbols area* is the default storage area for special or user-created symbols.
- *system symbols area* is usually designated as read-only and contains standard symbols provided with the Detailed Drawings module, such as the Welding Symbols Library.

To specify the directory in which you want to store user-defined symbols, set the configuration file option `pro_symbol_dir`. This automatically creates a search path to the specified directory. Pro/ENGINEER saves all symbols into and retrieves them from this directory by default if you add this option to your configuration file. If you change the location, the system does not delete symbols used in the drawing; once you add them, it stores the definitions locally in the drawing.

You should establish a single directory as your *user* library for all standard symbols. If you do not specify one, the system searches in the current working directory. You can change the default area for user-defined symbols by entering a new value for the `pro_symbol_dir` configuration file option. When you change this directory, you do not have to modify the configuration file; however, this change is valid only for the current Pro/ENGINEER session. Use this option to define new symbols that you store in a local or temporary directory; you can still easily retrieve symbols from the standard symbols area.

To Change the Symbol Directory

The symbol directory is determined by the configuration option `pro_symbol_dir`.

1. Click **Format > Symbol Gallery**. The **SYM Gallery** menu appears on the Menu Manager.
2. Click **Symbol Dir**.
3. Type the pathname of the directory for the new `pro_symbol_dir`.
4. To restore the default pathname in the current session, do one of the following:
   - Repeat this process and substitute the original pathname.
   - Click **Utilities > Options** from the Pro/ENGINEER menu bar and reload the configuration file in which this option resides.

To Store a Symbol

You do not have to save a symbol in order to continue using it in the drawing. However, to make symbols available for use in other drawings, you must save them to disk as `.sym` files.

You can save any symbol in the drawing as an individual `.sym` file.

1. Click **Format > Symbol Gallery**. The **SYMBOL** menu appears.
2. Click **Write**.
3. Specify the symbol by choosing **Name** or **Pick Inst** from the **GET SYMBOL** menu.
4. Type the offset directory path from the directory specified by `pro_symbol_dir` in which to store the symbol.

Symbol Types

About Symbol Types

While you can use either standard supplied symbols, or customize your own, the drawing symbols are characterized in the following manner:

- **Simple**: Each instance, or placement, of the symbol is identical. The graphic and textual content remain unchanged.
- **Generic**: Each instance contains familiar geometry or textual content, yet variations can exist. The graphic and textual content are chosen from a common group of symbol attributes.
- **Surface Finish**: Generic surface finish symbol that consists of building blocks, or groups.
- **Weld**: Generic system symbols according to the ANSI or ISO standards for creating welding, brazing and examination symbols in drawings.

Simple Symbols

About Simple Symbols

To define a simple symbol, you must draw the symbol geometry, then specify attributes, origin or attachment point, and any variable text notes needed. A simple symbol does not include grouped geometry, as does a generic symbol.

To Define a Simple Symbol

1. Create the draft geometry and add notes to include as fixed or variable text in the symbol. The text size and placement should be proportional to the geometry. Symbol text and geometry remain proportional when you modify the symbol height.
2. Click **Format** > **Symbol Gallery**. The **SYM GALLERY** menu appears.
3. Click **Define**.
4. Type a name for the symbol and press Enter. The **SYMBOL EDIT** menu appears.
5. Use the drafting tools to define the symbol shape or click **Copy Drawing** to copy entities from the format or drawing into the symbol edit window. Click **Done** when finished. The **Symbol Definition Attributes** dialog box opens.
6. Select the desired attributes from the **Symbol Definition Attributes** dialog box; then click **OK**. Using the **Symbol Definition Attributes** dialog box, you can specify the following symbol attributes:

- Allowed Placement Types
  - Free, on an entity, or normal to an entity
  - Left, right, or radial leader
- Symbol Instance Height
  - Fixed
  - Variable in drawing units, model units, or text-related
- Position and other characteristics

    - At a fixed text angle
    - With an elbow
    - Mirror image of the original geometry or text
- Variable text

  If you insert text notes with a backslash before and after them (\notetext\ ) in the symbol definition, these notes become "variable text" notes. They appear in the **Var Text** tab of the **Symbol Definition Attributes** dialog box. Variable text means that you may enter a selection of values in the preset values text box for each variable text note. For example, if your note had the text

  "`Drafter Name \names\`,"

  "names" would appear in the **Var Text** tab. You could enter a list of employees names in the pre set values area to select from when the instance is placed. When you place the instance, you can select a name to follow "Drafter Name" from a drop down list in the **Custom Drawing Symbol** dialog box. (Select the symbol and click **Properties** on the right mouse button shortcut menu to show the dialog box.)

  You may also enter a parameter as the preset value for the variable text, for example "&draftername". If there is a "draftername" parameter in the drawing, the symbol note will return its value.

7. Click **SYMBOL EDIT** > **Done**. The system confirms that it has successfully defined the symbol.

Generic Symbols

About Defining Generic Symbols

A generic symbol defines a family of similar symbols; it contains all entities pertaining to this family. You can arrange geometry and text in the generic symbol in groups and subgroups, creating a tree structure of symbol definition.

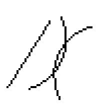
The tree definition structure helps you create symbol instances. When you specify the groups to include, the system assembles an instance out of predefined blocks, or groups. A group can consist of other subgroups and independent entities. The system always selects these independent entities on each level when you select the group to which they belong. Subgroups of different subgroups can share the same name.

Each level of symbol definition, containing more than one group, is characterized by the group attribute restricting the selection of groups at the specified level. That is, you can choose **Exclusive** from the GROUP ATTR menu to define groups, so that you include only one of them in the symbol instance, or you can use the **Independent** command to define them, so that you can select any number of groups (or none).

To Create a Tree Definition Structure

This procedure walks you through creating a generic symbol with a multi level structure.

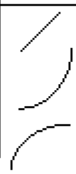
1. Click **Format** > **Symbol Gallery**. The **SYM Gallery** menu appears on the menu manager.
2. Click **Define**.
3. Type the symbol name, `[FILLET]`. A second Pro/ENGINEER window opens with the Menu Manager open to the **SYMBOL EDIT** menu. Use this window to create the symbol.
4. Use the drafting tools to create new geometry.
5. Click **SYMBOL EDIT** > **Groups** > **Create**.
6. Type the group name as ARROW_SIDE. Select all of the entities that belong to the arrow side except the reference line. The system does not include the reference line in any group because it must appear in all symbol instances. After you select all entities, click **OK**.
7. To create another top-level group, choose **SYM GROUPS** > **Create**, type the name of the group [OTHER_SIDE], and select entities located on the other side of the reference line. After you select all entities, click **OK**.
8. For this example, top-level groups are exclusive. Click **SYM GROUPS** > **Group Attr** > **Exclusive**.
9. To create subgroups of the top-level groups, choose **SYM GROUPS** > **Change Level**.
10. The **TOP LEVEL** menu displays the list of groups at the current level and the **This Level** command. Click **ARROW_SIDE**; all entities pertaining to this group appear in the symbol edit window.
11. Click **ARROW_SIDE > This Level**.
12. To specify subgroups at the current level, choose **Create**, type the group name, and select the corresponding geometry or text line. When creating subgroups, use the following table:

Group name	What you select on the screen
WELD_SIZE	/weld_size/
LENGTH	/length/
PITCH	/pitch/
CONTOUR	
FINISH	 letters C, G, H, M, R, U on top of one another

Note: The system does not include the fillet in any subgroups because it must always be in any arrow-side instance.

By default, the system sets the group attribute to **Independent**; therefore, you do not have to use the **GROUP ATTR** menu. Groups are independent because you can include all groups in a single instance.

1. To specify the subgroups of the group **CONTOUR**, choose **Change Level** > **CONTOUR**. The symbol edit window displays entities from the current group.
2. Click **SYM GROUPS** > **Create**. Create these three groups:

Group name	What you select on the screen
FLUSH CONVEX CONCAVE	

3. Click **SYM GROUPS** > **Group Attr** > **Exclusive** (you can choose only one of the preceding subgroups under the group **CONTOUR** at one time).
4. Click **Change Level** > **UP** > **FINISH**. To select the proper character on the screen, use **Query Sel**. Create groups as follows:

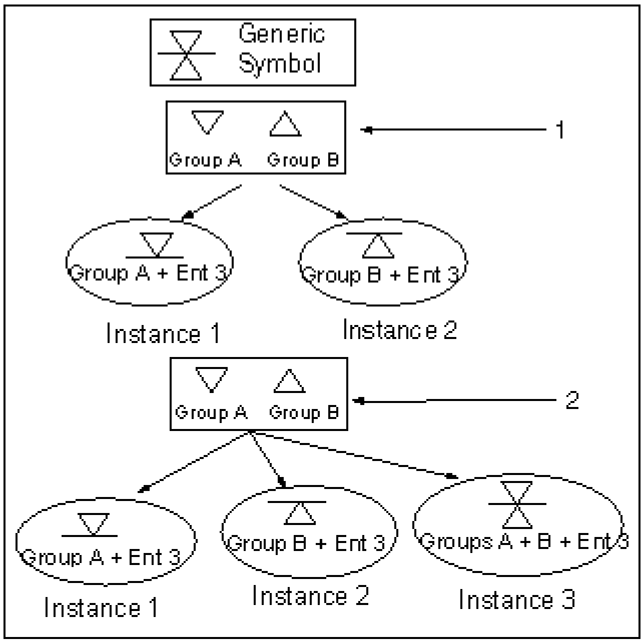
Group name	What you select on the screen
CHIP	C
GRIND	G
MACHINE	M
HAMMER	H
ROLL	R
UNSPECIFIED	U

5. Click **SYM GROUPS** > **Group Attr** > **Exclusive**. You have completely described the symbol along the arrow-side branch.
6. To specify subgroups along the other-side branch, choose **Change Level** > **UP** to return to the **TOP LEVEL** menu.
7. Click **OTHER_SIDE** and proceed to create groups, following a procedure similar to the one described in Steps 8 through 19.
8. When you finish the symbol definition, choose **SYM GROUPS** > **Done**.
9. Click **SYMBOL EDIT** > **Attributes**.
10. From the **Symbol Definition Attributes** dialog box, select items characterizing the attachment point, leader type, and symbol size. For this example, click **Left Leader** and **Variable**.
11. Select the leader origin on the left side of the symbol; then click **OK**.

12. Click the **Variable Text** tab to type values for notes created between slashes as variable text entries.

13. The system informs you that it has defined the symbol. To save the generic symbol on a disk, choose **SYMBOL EDIT** > **Write**. Type the directory path or accept the default.

Example: Creating a Generic Symbol Definition



Note: The line is not included in any group because it must appear in all instances.

1. Groups A and B are mutually exclusive.

2. Groups A and B are independent.

Parametric Weld Symbols

About Working with Parametric Weld Symbols

You can show weld symbols in drawings for welds that were created in Welding, as well as redefine the system-supplied parametric weld symbols to improve your flexibility and productivity.

Showing Symbols for Welds Created in Welding

You can show and erase weld symbols in drawings that correspond to welds that were created in an assembly using Welding. You can also do the following:

- Modify the number of decimal places (*num digits*) shown in dimensions contained within a weld feature.
- Add or delete leader lines to weld symbols.

To display weld symbols from Welding in drawings according to the ANSI or ISO standard, set the value of the drawing setup file option `weld_symbol_standard` to `STD_ANSI` or `STD_ISO`, respectively.

Restrictions on the Use of Weld Symbols in Drawings

When you are working with weld symbols, the following restrictions apply:

- The display of the weld feature does not affect the display of the weld symbol, and it does not update its point of attachment when you blank, resume, erase, or show the weld feature. However, when the system first shows a weld symbol, its default attachment adapts to the display status of its feature.
- The system shows a weld symbol only once in a drawing, similar to assembly geometric tolerances and surface finishes.
- Revision 15.0 and later revisions support the following compound welding symbols:
    - Reinforced Welds: square groove, bevel groove, flared bevel groove, and J-groove (all reinforced welds can also be two-sided).
    - Two-sided Welds: fillet, square groove, V-groove, bevel groove, U-groove, J-groove, flared V-groove, flared bevel groove.

Modifying the Number of Decimal Places of a Fillet Weld Feature

When you are using **Num Digits** to modify a value of a fillet weld feature (simple, two-sided, or reinforcing a groove) that has differing leg length values shown in the symbol, such as "L1 x L2," the number of decimal places shown in the two values are linked. That is, if you select this portion of the symbol for modification, the system highlights the entire "L1 x L2," and both values change to the number of decimal places you specify.

To Regroup Weld Symbol Instances

To regroup weld symbol instances, set the configuration file option `sym_leader_orient_move_text` to `yes` (the default is `no`). The system then regroups an instance after you move the text.

User-Defined Parametric Weld Symbols

You can replace the Pro/ENGINEER-supplied library of system weld symbols with user-defined ones. After you define the symbols, the system uses them for automatic weld annotation. By customizing your weld symbols in advance, you can increase your flexibility and productivity throughout the processes of creating and modifying drawings.

> **Note:** Before performing this procedure, you should copy the system-supplied welding symbol library to a backup directory.

However, when creating user-defined weld symbols, the following restrictions apply:

- All of the groups that existed in the original definition must remain in the new definition, and you cannot add new ones or change the names of existing ones.

- If you add new variable text, or change the name of an existing piece of variable text, the new name must be the same as that of the existing variable text in the original.
- The height type of the symbol instance must be the same in the new user-defined symbol as it was in the original.
- The **Left Leader** and **Right Leader** placement types must both exist in the new user-defined weld symbol.

ISO Welding Symbols

This section presents examples of symbols that enable you to create welding, brazing, and examination symbols in a drawing according to the ISO standard. To help you create symbol instances, it provides the following information for each symbol:

- The symbol name
- An example of an instance created from this symbol and the associated menu picks
- Listings of groups and subgroups in menu format reflecting a tree symbol definition structure

You can also refer to **Glossary of Menu Picks** below for an explanation of menu selections for welding symbols.

Welding Symbols Library

The Welding Symbols Library, available with Detailed Drawings, provides a collection of symbols based on the standards of ISO-2553-1984. Using this library, you can create a variety of welding, brazing, and examination symbols in a drawing.

To retrieve a symbol from the Welding Symbol Library, choose System Syms from the Select File menu to access the Pro/ENGINEER symbols area.

Glossary of Menu Picks

This glossary describes terms that are unique to the ISO Welding Symbol Library and identifies information you are prompted to enter. Menu pick names track the ISO Welding Symbol Library to the fullest extent possible.

In the ISO Welding Symbol Library, two lines form the reference line, one solid and one dashed. Weld symbols attached to the solid line are "arrow-side." Weld symbols attached to the dashed line are "other-side." The dashed line is only omitted for symmetrical welds. When creating a welding symbol, you first select whether the weld is arrow side, other-side, or symmetrical. If you choose ARROW_SIDE or OTHER_SIDE, the system prompts you to create the weld symbol above or beneath the reference line. It then places the dashed line appropriately in the symbol depending on whether it is arrow-side or other-side.

Table 1: ISO Weld Characteristics Menu Selections

MENU PICK	**OPTION**	**DESCRIPTION**
ABOVE_REF		Creates the weld symbol above the solid reference line.
AS_ABOVE_ REF		Indicates (for combination welding symbols only) whether the arrow-side welding symbol is placed above or beneath the reference line.
BENEATH_REF		Creates the weld symbol beneath the solid reference line.
BUTT_SIZE		Refers to the final size of the weld, depending on the symbol type (arrow-side, other-side, symmetrical, or staggered). It might have the suffix "AS" (arrow-side) or "OS" (other-side) to indicate the appropriate weld side.
CONTOUR	SMOOTH_BLEND	Describes the final shape of the weld. This choice indicates that the toes of the weld are blended smoothly.
FINISH		In the ANSI standard, this specifies the method of finishing. In the ISO standard, the system adds a machine finish surface texture symbol above the contour symbol.
FILLET_SIZE		Refers to the final size of the weld, depending on the symbol type (arrow-side, other-side, symmetrical, or staggered). It might have the suffix "AS" (arrow-side) or "OS" (other-side) to indicate the appropriate weld side.

MENU PICK	OPTION	DESCRIPTION
FIL_SIZE		Refers to the final size of the weld, depending on the symbol type (arrow-side, other-side, symmetrical, or staggered). It might have the suffix "AS" (arrow-side) or "OS" (other-side) to indicate the appropriate weld side.
HOLE_DIA		Refers to the diameter of a plug weld.
OS_ABOVE_ REF		Indicates (for combination welding symbols only) whether the other-side welding symbol is placed above or beneath the reference line.
PROJECT_DIA		Refers to the diameter of a projection weld.
SEAM_WIDTH		Refers to the width of a seam weld.
SPOT_DIA		Refers to the diameter of a spot or fusion weld.
STYLE		Refers to the longitudinal dimension of a weld. Two choices are available:
	CONTINOUS	Weld joint continuously for the specified length.
	INTERMITTENT	Creates weld beads of specified length, distance between adjacent weld elements, and specific number of beads.
SYMMETRICAL		Refers to a welding symbol for creating identical welds on both arrow-side and other- side.
TAIL		Creates a tail on the opposite end of the welding symbol from the leader attachment. Provides two choices:
	REFERENCE	Enters reference text or note.
	WELD_PROCESS	

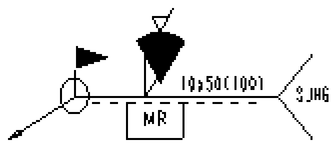
MENU PICK	OPTION	DESCRIPTION
		Specifies a welding process.
WELD_SIZE		Refers to the final size of the weld, depending on the symbol type (arrow-side, other-side, symmetrical, or staggered). It might have the suffix "AS" (arrow-side) or "OS" (other-side) to indicate the appropriate weld side.
WELD_TYPE		For combination welding symbols, indicates the type of weld being created and whether it is created arrow-side or other-side (e.g., FILLET_AS).

Bevel Butt Symbol: Bevel_Butt.sym

EXAMPLE 1

PICKS

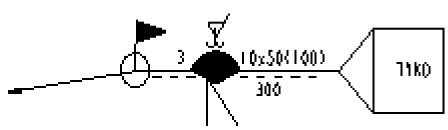
1. arrow_side
2. above_ref
3. style
4. contour
5. back_type
6. finish
7. leader_orient
8. intermittent
9. convex
10. strip
11. left
12. removable
13. field
14. all_around
15. tail
16. weld_process



EXAMPLE 2

PICKS

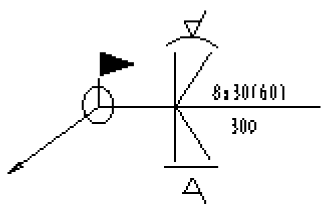
1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference



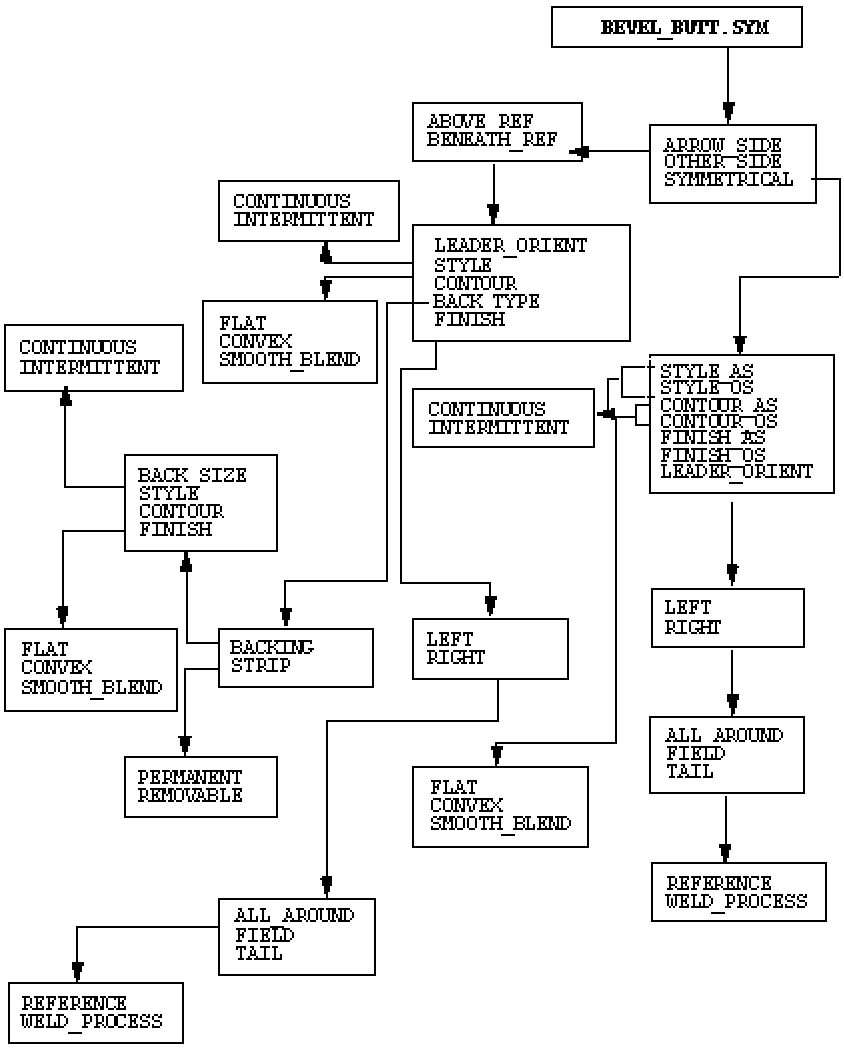
EXAMPLE 3

PICKS

1. symmetrical
2. style_as
3. style_os
4. contour_as
5. contour_os
6. finish_as
7. finish_os
8. leader_orient
9. intermittent
10. continuous
11. convex
12. flat
13. left
14. all_around
15. field



Symbol Definition Structure

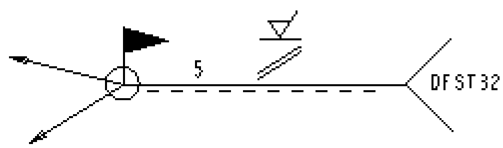


Inclined Joint Symbol: Inclined_Joint.sym

EXAMPLE 1

PICKS

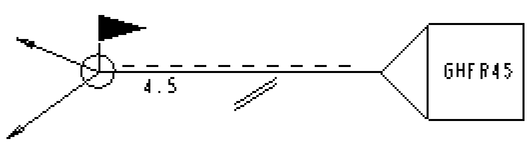
1. arrow_side
2. weld_size
3. finish
4. flat
5. leader_orient
6. left
7. field
8. all_around
9. tail
10. weld_process



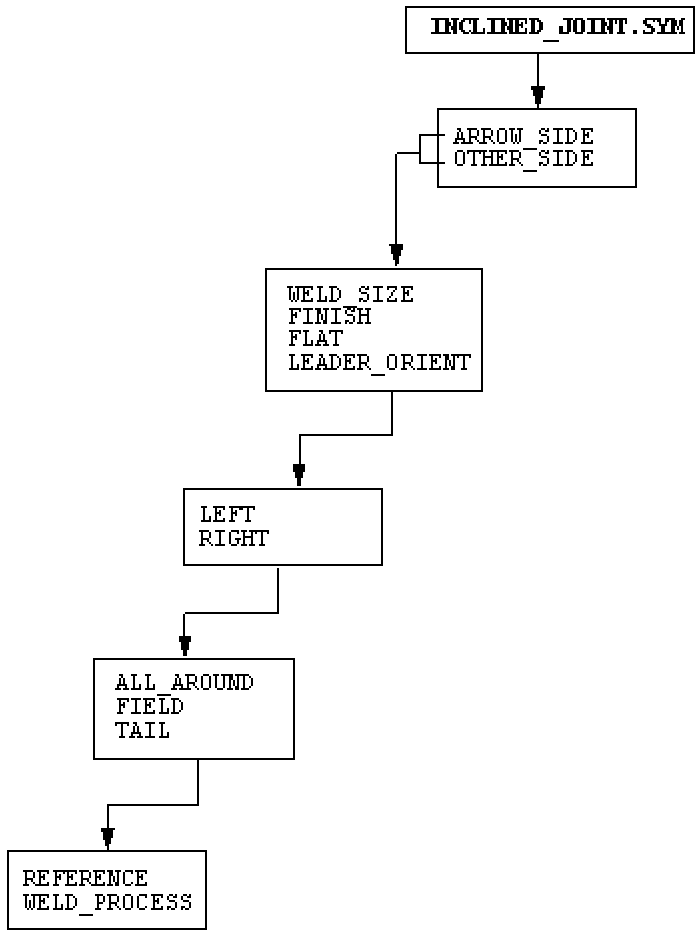
EXAMPLE 2

PICKS

1. other_side
2. weld-size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. reference



Symbol Definition Structure

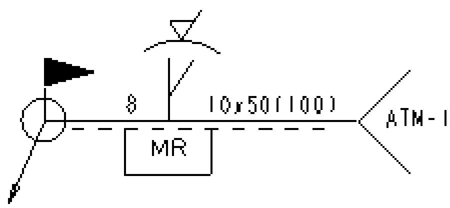


Bevel Butt with Broad Root Face: Br_Root_Bevel.sym

EXAMPLE 1

PICKS

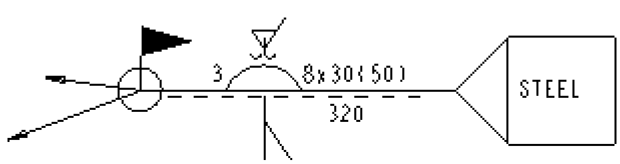
1. arrow_side
2. above_ref
3. weld_size
4. style
5. contour
6. back_type
7. finish
8. leader_orient
9. intermittent
10. convex
11. strip
12. left
13. removable
14. field
15. all_around
16. tail
17. weld_process



EXAMPLE 2

PICKS

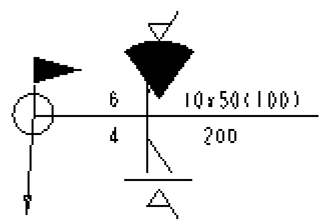
1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference



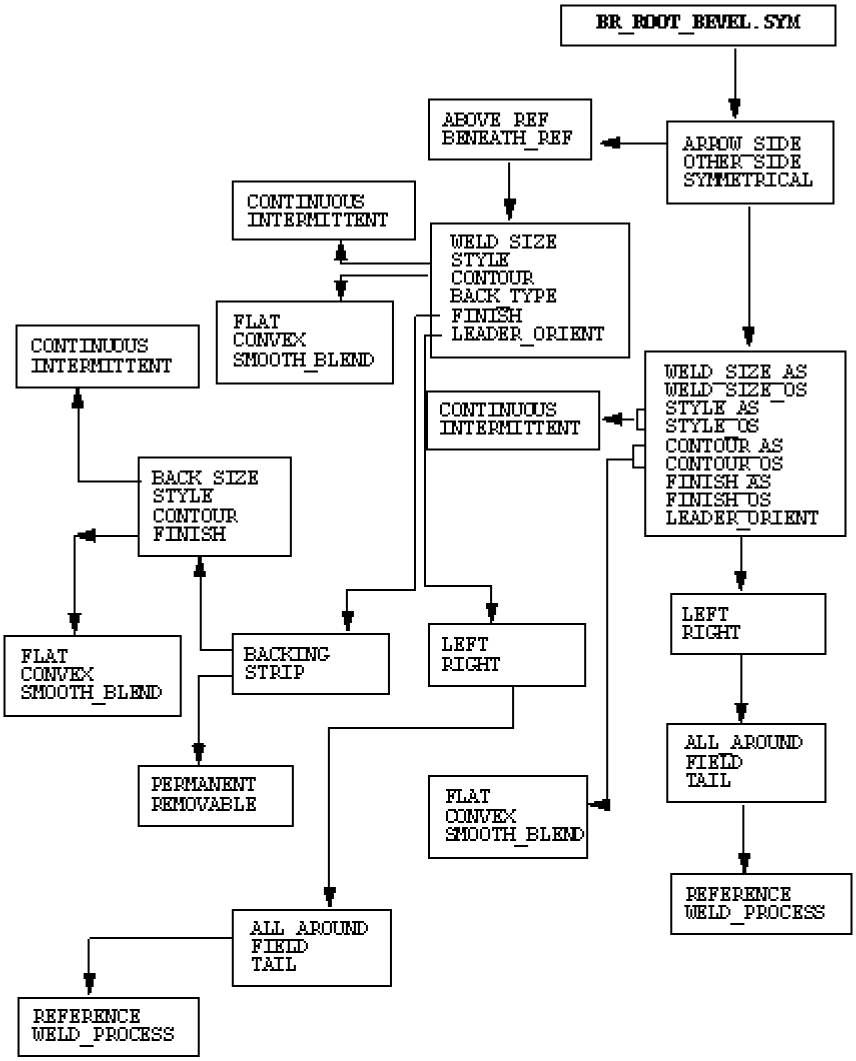
EXAMPLE 3

PICKS

1. symmetrical
2. weld_size_as
3. weld_size_os
4. style_as
5. style_os
6. contour_as
7. contour_os
8. finish_as
9. finish_os
11. intermittent
12. continuous
13. convex
14. flat
15. left
16. all_around
17. field



Symbol Definition Structure

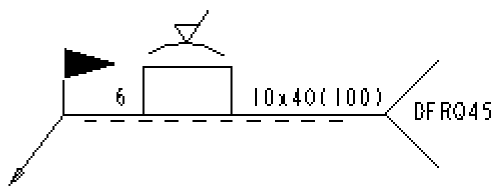


Slot Symbol: Iso_Slot.sym

EXAMPLE 1

PICKS

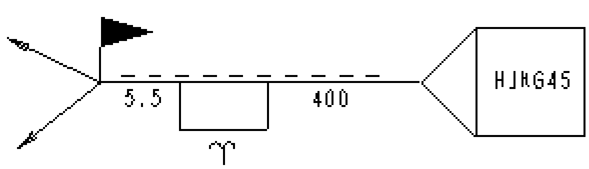
1. arrow_side
2. slot_width
3. style
4. contour
5. finish
6. leader-orient
7. intermittent
8. convex
9. left
10. field
11. tail
12. weld_process



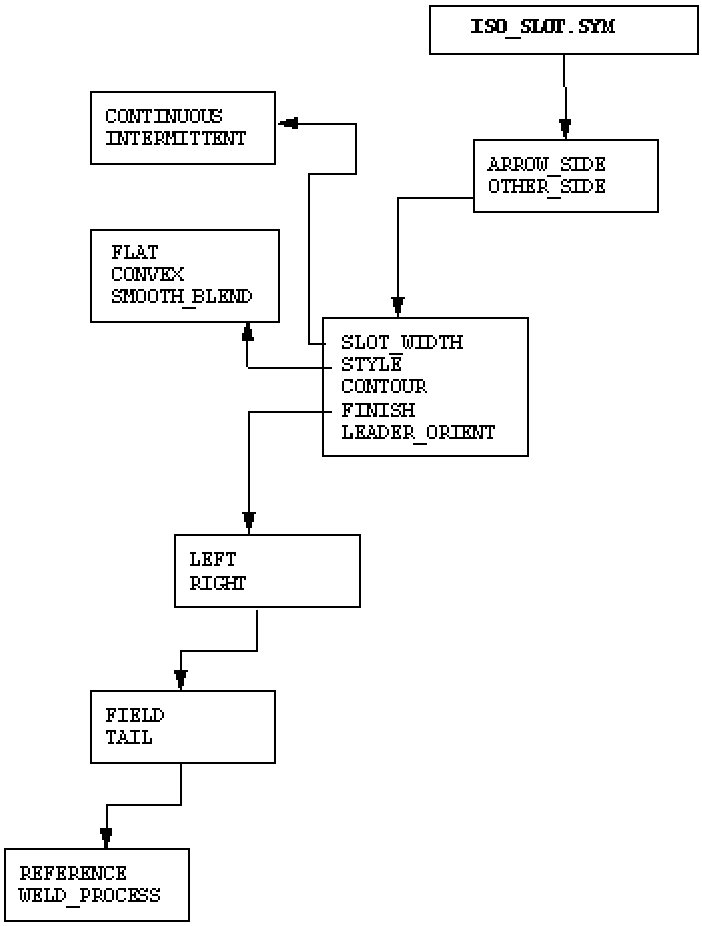
EXAMPLE 2

PICKS

1. other_side
2. slot_width
3. style
4. contour
5. leader_orient
6. continuous
7. smooth_blend
8. left
9. field
10. tail
11. reference



Symbol Definition Structure

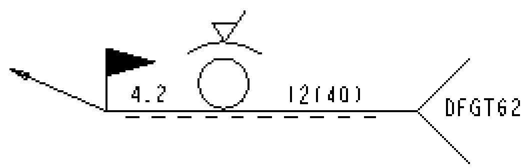


Spot Symbol: Iso_Spot.sym

EXAMPLE 1

PICKS

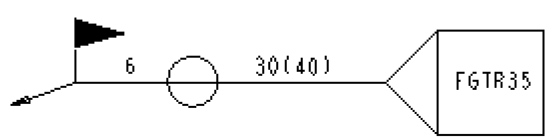
1. fusion
2. above_ref
3. spot_dia
4. number_pitch
5. contour
6. finish
7. leader_orient
8. convex
9. left
10. field
11. tail
12. weld_process

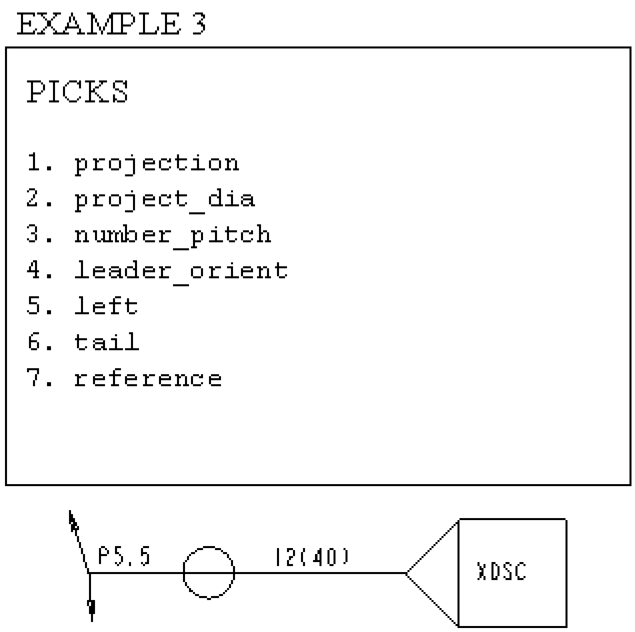


EXAMPLE 2

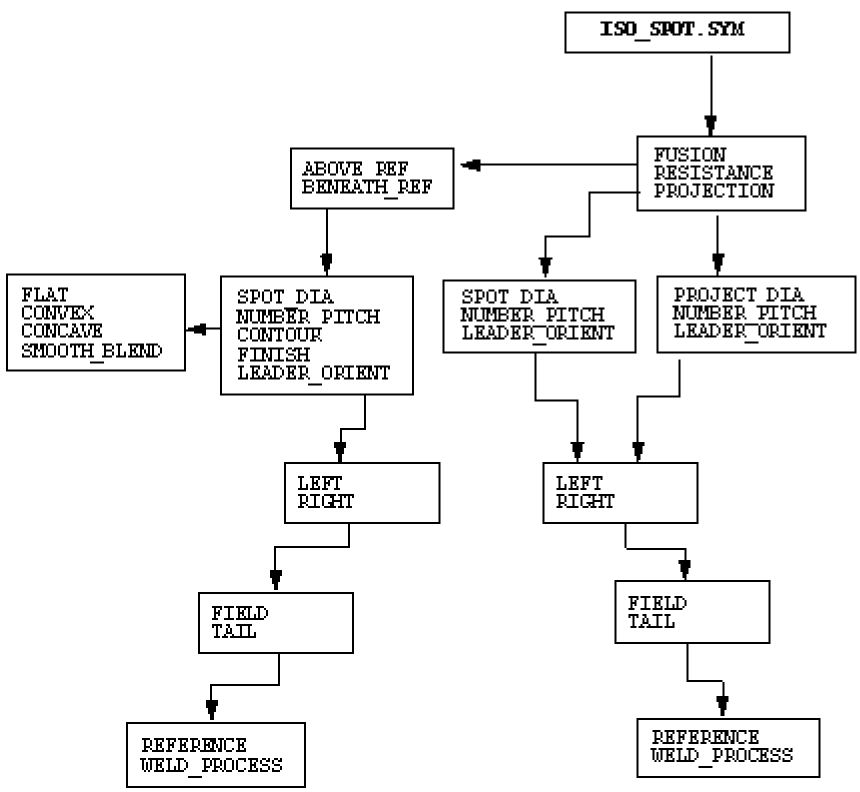
PICKS

1. resistance
2. spot_dia
3. number_pitch
4. leader_orient
5. left
6. field
7. tail
8. reference





Symbol Definition Structure

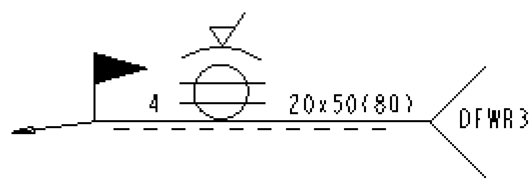


Seam Symbol: Iso_Seam.sym

EXAMPLE 1

PICKS

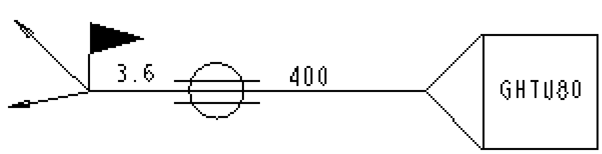
1. fusion	11. field
2. above_ref	12. tail
3. seam_width	13. weld_process
4. style	
5. contour	
6. finish	
7. leader_orient	
8. intermittent	
9. convex	
10. left	

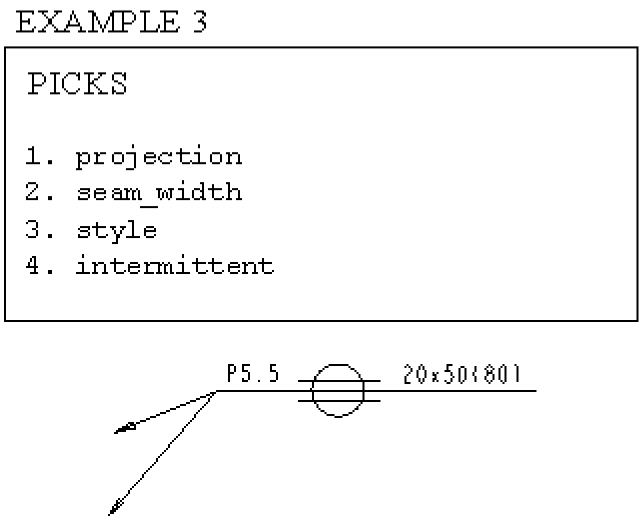


EXAMPLE 2

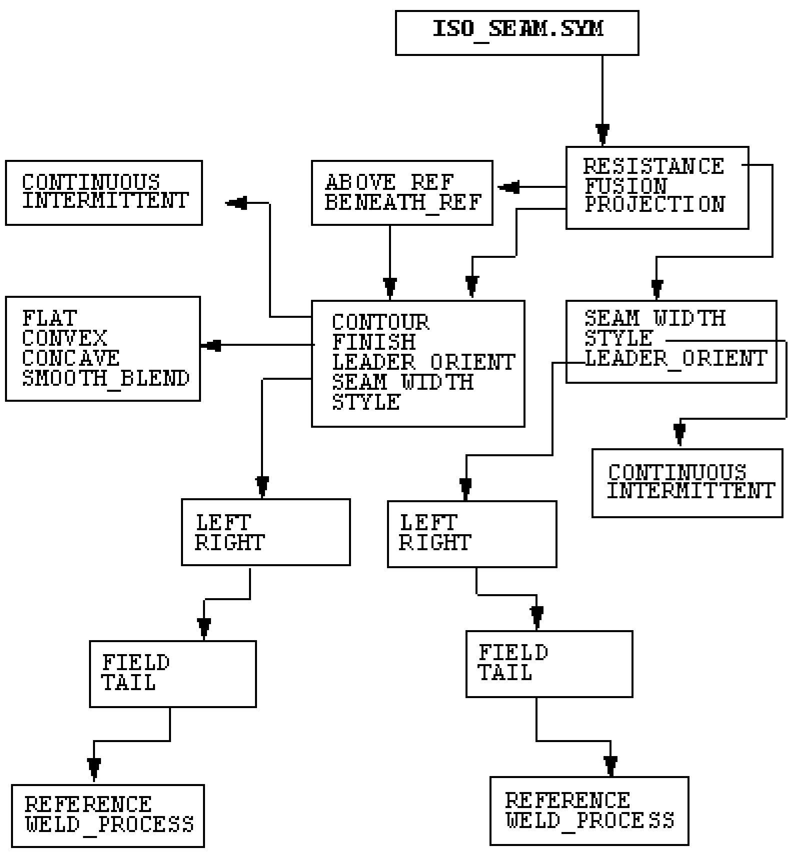
PICKS

1. resistance
2. seam_width
3. style
4. leader_orient
5. continuous
6. left
7. field
8. tail
9. reference





Symbol Definition Structure

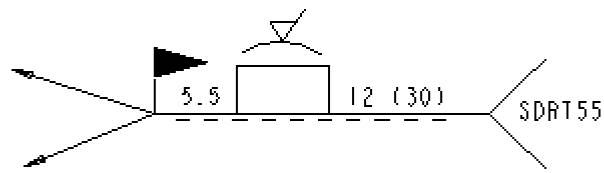


Plug Symbol: Iso_Plug.sym

EXAMPLE 1

PICKS

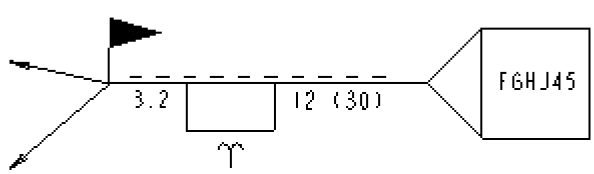
1. arrow_side
2. hole_dia
3. number_space
4. contour
5. finish
6. leader_orient
7. convex
8. left
9. field
10. tail
11. weld_process



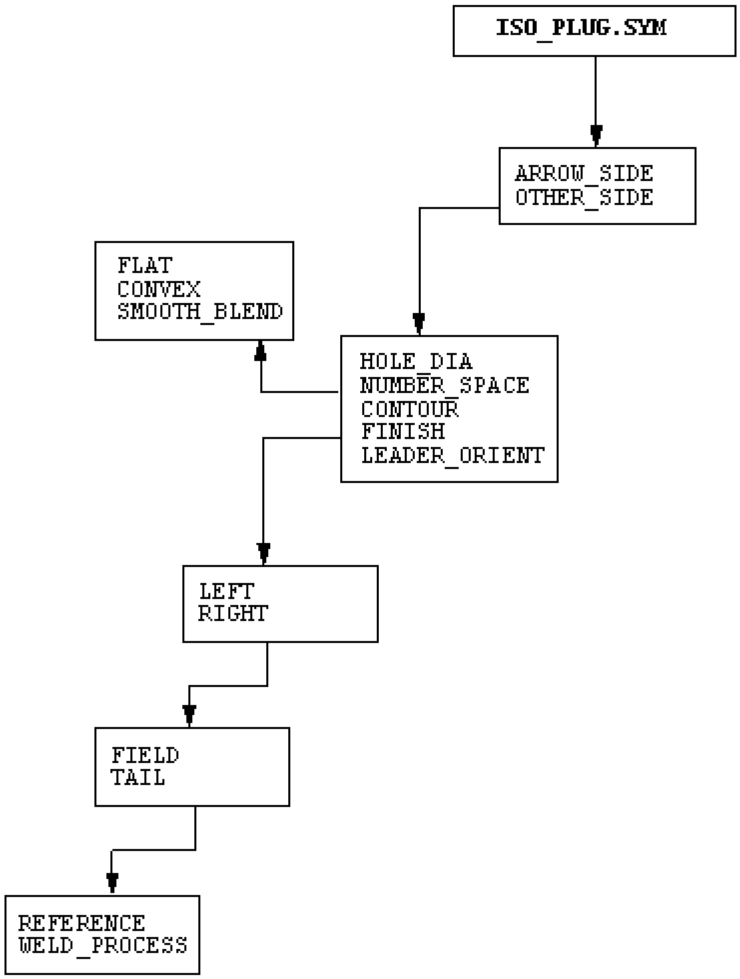
EXAMPLE 2

PICKS

1. other_side
2. hole_dia
3. contour
4. number_space
5. leader_orient
6. smooth_blend
7. left
8. field
9. tail
10. reference



Symbol Definition Structure

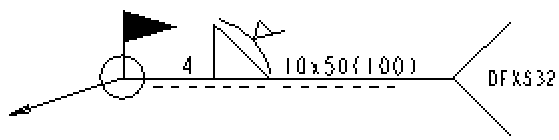


Fillet Symbol: Iso_Fillet.sym

EXAMPLE 1

PICKS

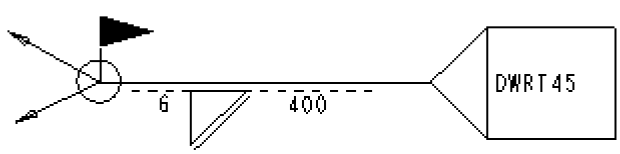
1. arrow_side
2. above_ref
3. fillet_size
4. style
5. contour
6. finish
7. leader_orient
8. intermittent
9. convex
10. left
11. field
12. all_around
13. tail
14. weld_process



EXAMPLE 2

PICKS

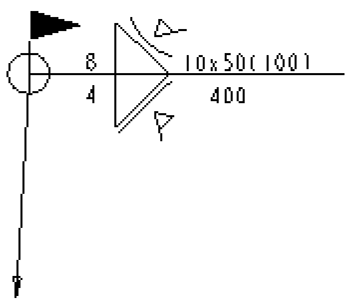
1. other_side
2. beneath_ref
3. fillet_size
4. style
5. contour
6. leader_orient
7. continuous
8. flat
9. left
10. field
11. all_around
12. tail
13. reference



EXAMPLE 3

PICKS

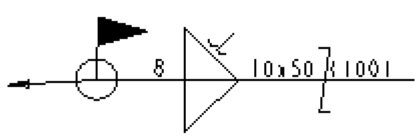
1. symmetrical
2. fil_size_as
3. fil_size_os
4. style_as
5. style_os
6. contour_as
7. contour_os
8. finish_as
9. finish_os
10. leader_orient
11. intermittent
12. continuous
13. concave
14. flat
15. left
16. all_around
17. field



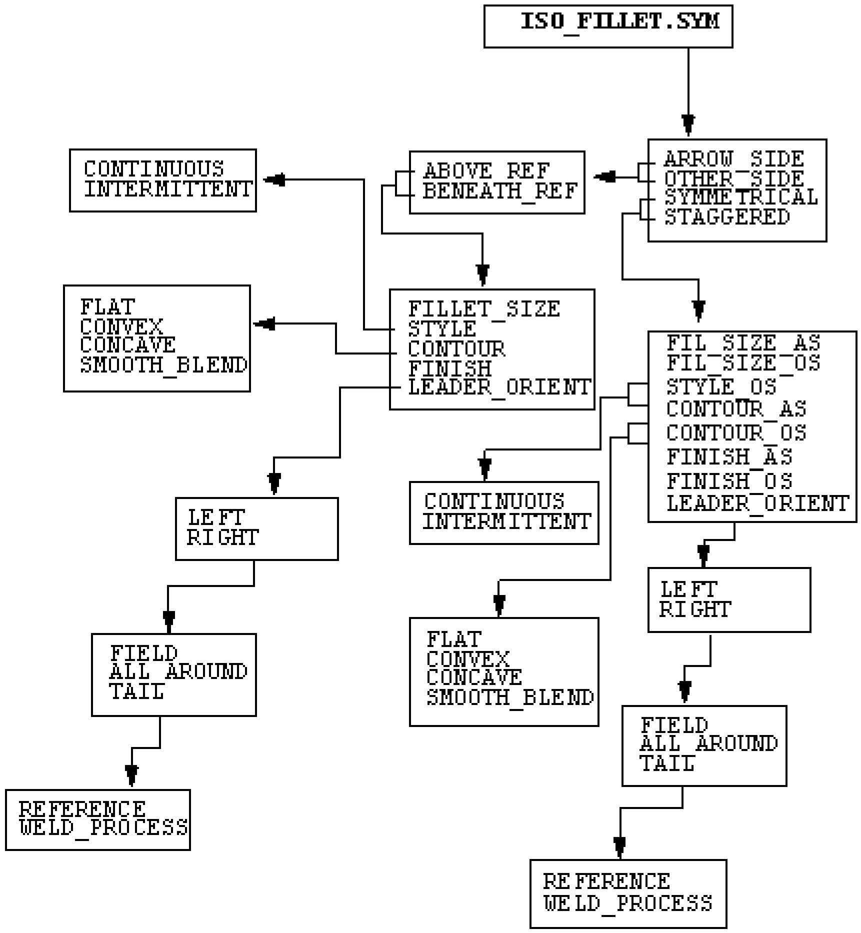
EXAMPLE 4

PICKS

1. staggered
2. fil_size_as
3. contour_as
4. leader_orient
5. smooth_blend
6. left
7. all_around
8. field



Symbol Definition Structure

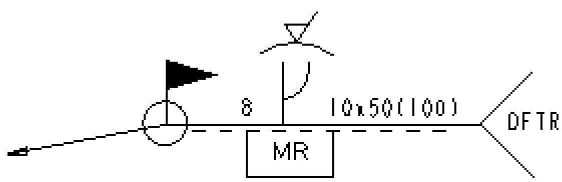


J Butt Symbol: J_Butt.sym

EXAMPLE 1

PICKS

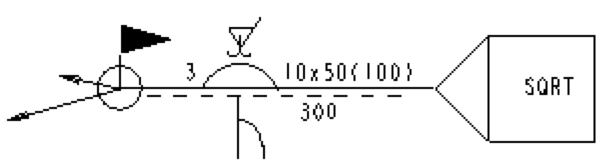
1. arrow_side
2. above_ref
3. weld_size
4. style
5. contour
6. back_type
7. finish
8. leader_orient
9. intermittent
10. convex
11. strip
12. left
13. removable
14. field
15. all_around
16. tail
17. weld_process



EXAMPLE 2

PICKS

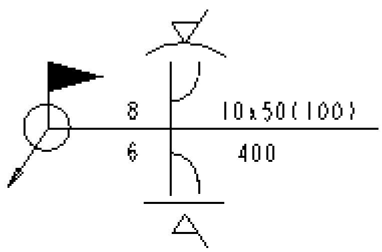
1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference



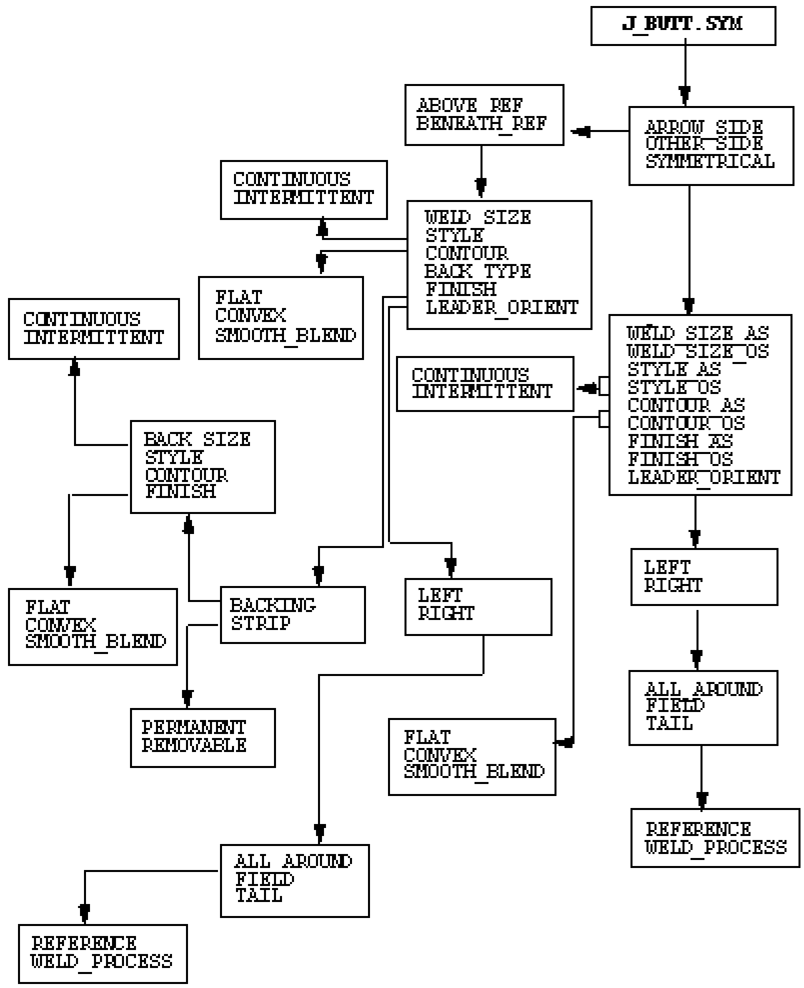
EXAMPLE 3

PICKS

1. symmetrical
2. weld_size_as
3. weld_size_os
4. style_as
5. style_os
6. contour_as
7. contour_os
8. finish_as
9. finish_os
10. leader_orient
11. intermittent
12. continuous
13. convex
14. flat
15. left
16. all_around
17. field



Symbol Definition Structure

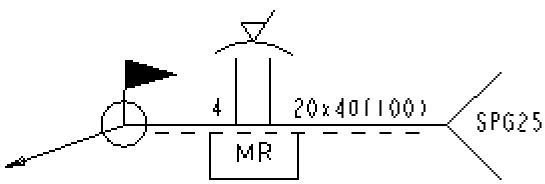


Square Butt Symbol: Iso_Square.sym

EXAMPLE 1

PICKS

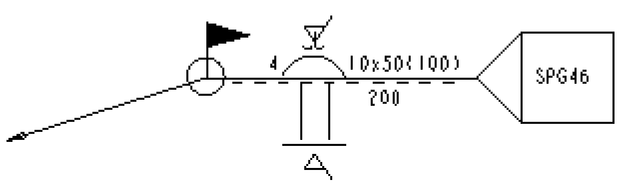
1. arrow_side
2. above_ref
3. weld_size
4. style
5. contour
6. back_type
7. finish
8. leader_orient
9. intermittent
10. convex
11. strip
12. left
13. removable
14. field
15. all_around
16. tail
17. weld_process



EXAMPLE 2

PICKS

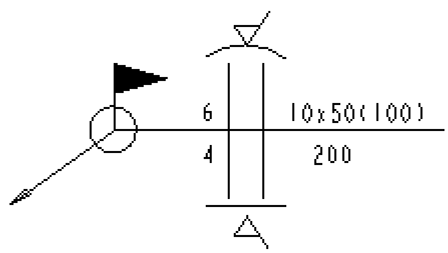
1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference



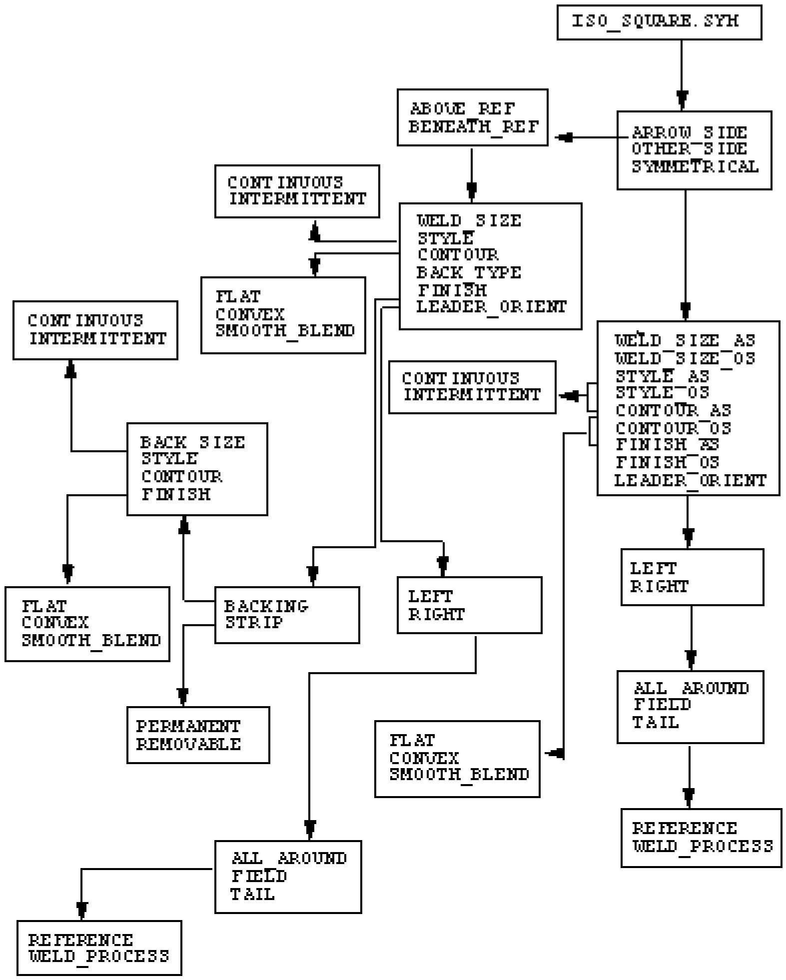
EXAMPLE 3

PICKS

1. symmetrical
2. weld_size_as
3. weld_size_os
4. style_as
5. style_os
6. contour_as
7. contour_os
8. finish_as
9. finish_os
10. leader_orient
11. intermittent
12. continuous
13. convex
14. flat
15. left
16. all_around
17. field



Symbol Definition Structure

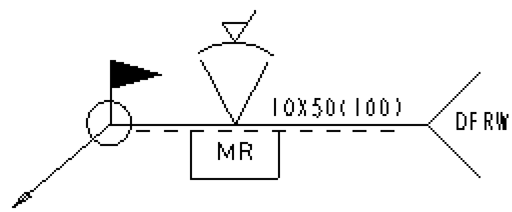


V Butt Symbol: V_Butt.sym

EXAMPLE 1

PICKS

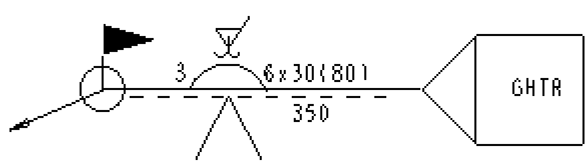
1. arrow_side
2. above_ref
3. style
4. contour
5. back_type
6. finish
7. leader_orient
8. intermittent
9. convex
10. strip
11. left
12. removable
13. field
14. all_around
15. tail
16. weld_process



EXAMPLE 2

PICKS

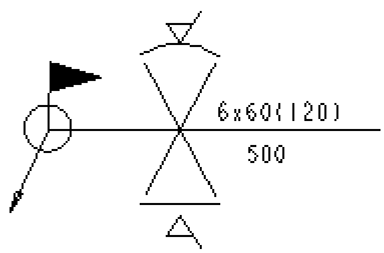
1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference



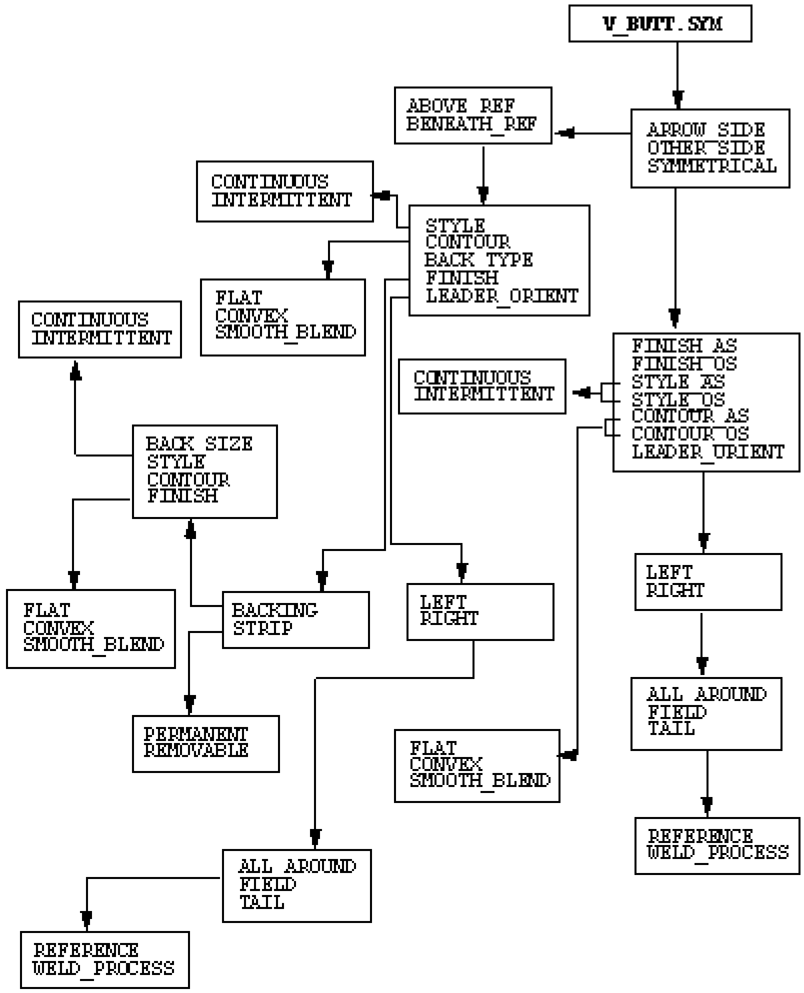
EXAMPLE 3

PICKS

1. symmetrical
2. style_as
3. style_os
4. contour_as
5. contour_os
6. finish_as
7. finish_os
8. leader_orient
9. intermittent
10. continuous
11. convex
12. flat
13. left
14. all_around
15. field



Symbol Definition Structure

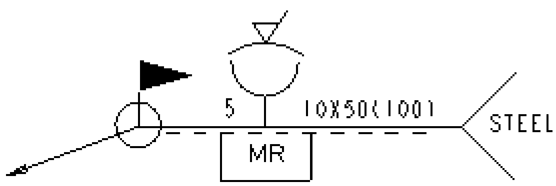


U Butt Symbol: U_Butt.sym

EXAMPLE 1

PICKS

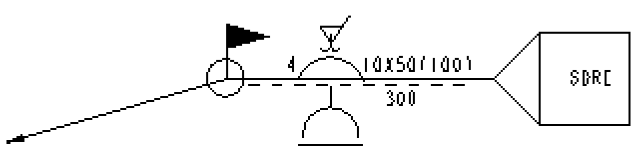
1. arrow_side
2. above_ref
3. weld_size
4. style
5. contour
6. back_type
7. finish
8. leader_orient
9. intermittent
10. convex
11. strip
12. left
13. removable
14. field
15. all_around
16. tail
17. weld_process



EXAMPLE 2

PICKS

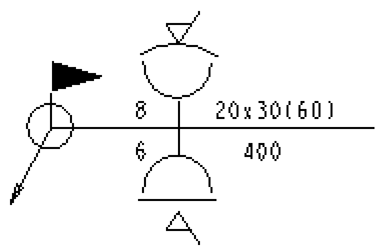
1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference



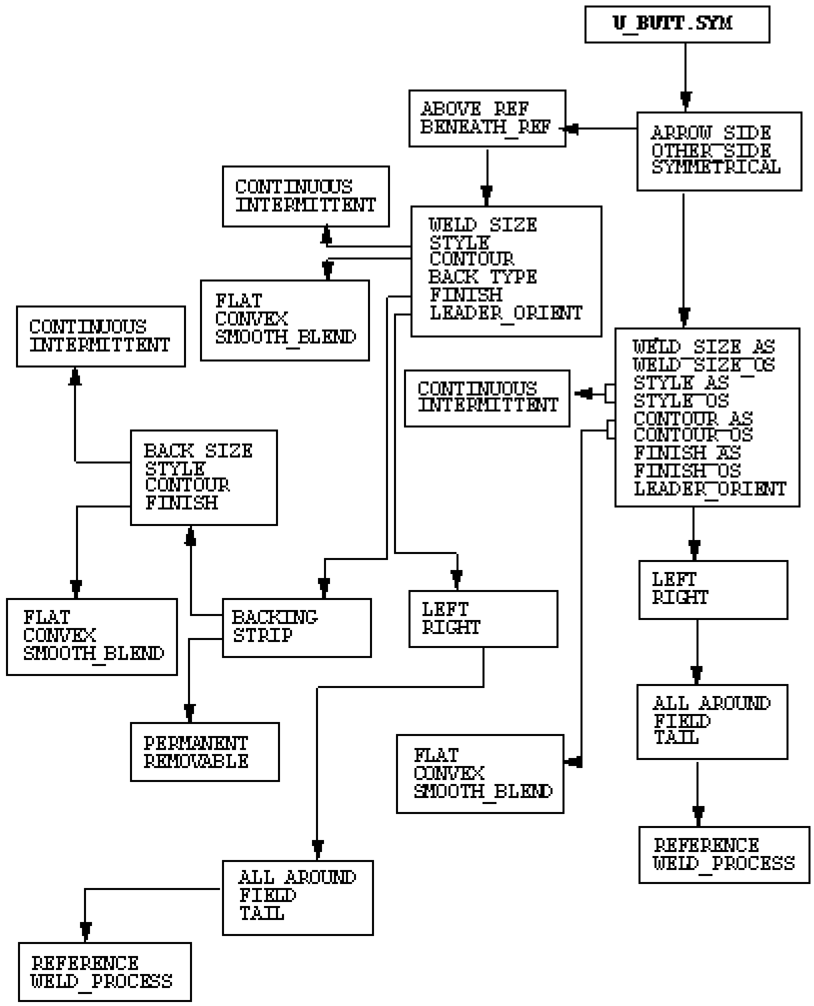
EXAMPLE 3

PICKS

1. symmetrical
2. weld_size_as
3. weld_size_os
4. style_as
5. style_os
6. contour_as
7. contour_os
8. finish_as
9. finish_os
10. leader_orient
11. intermittent
12. continuous
13. convex
14. flat
15. left
16. all_around
17. field



Symbol Definition Structure

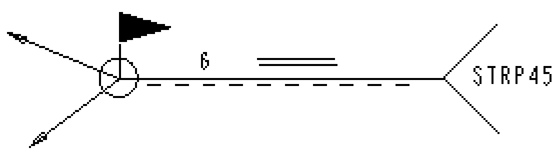


Surface Joint Symbol: Surface_Joint.sym

EXAMPLE 1

PICKS

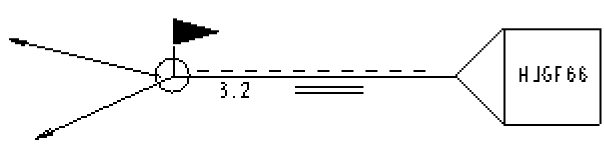
1. arrow_side
2. weld_size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. weld_process



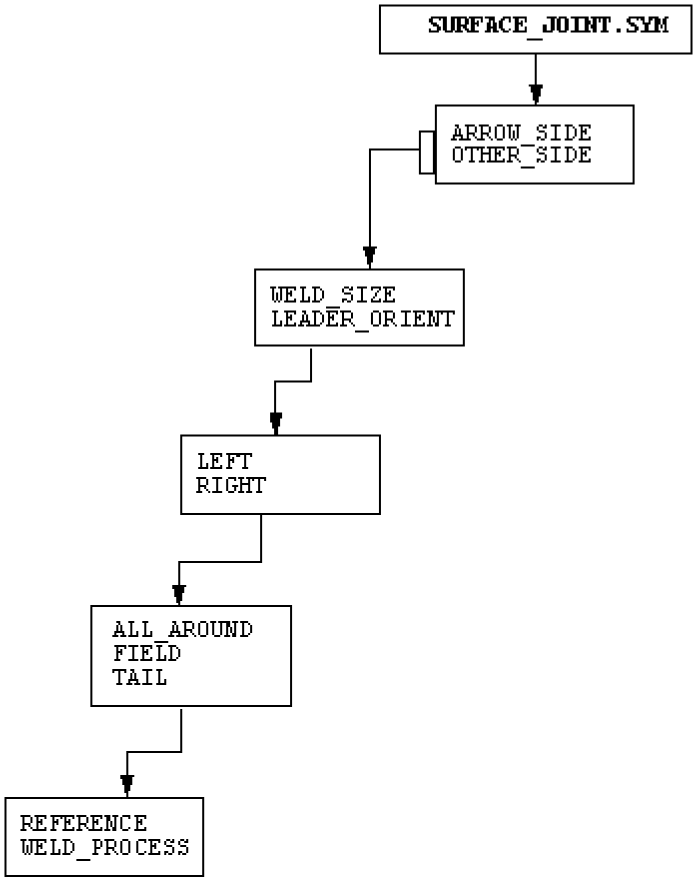
EXAMPLE 2

PICKS

1. other_side
2. weld_size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. reference



Symbol Definition Structure

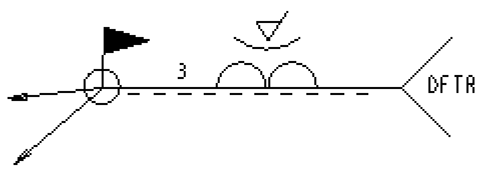


Surfacing Symbol: Iso_Surfacing.sym

EXAMPLE 1

PICKS

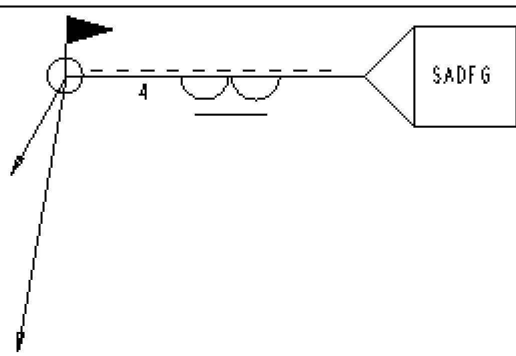
1. arrow_side
2. weld_size
3. contour
4. finish
5. leader_orient
6. concave
7. left
8. field
9. all_around
10. tail
11. weld_process



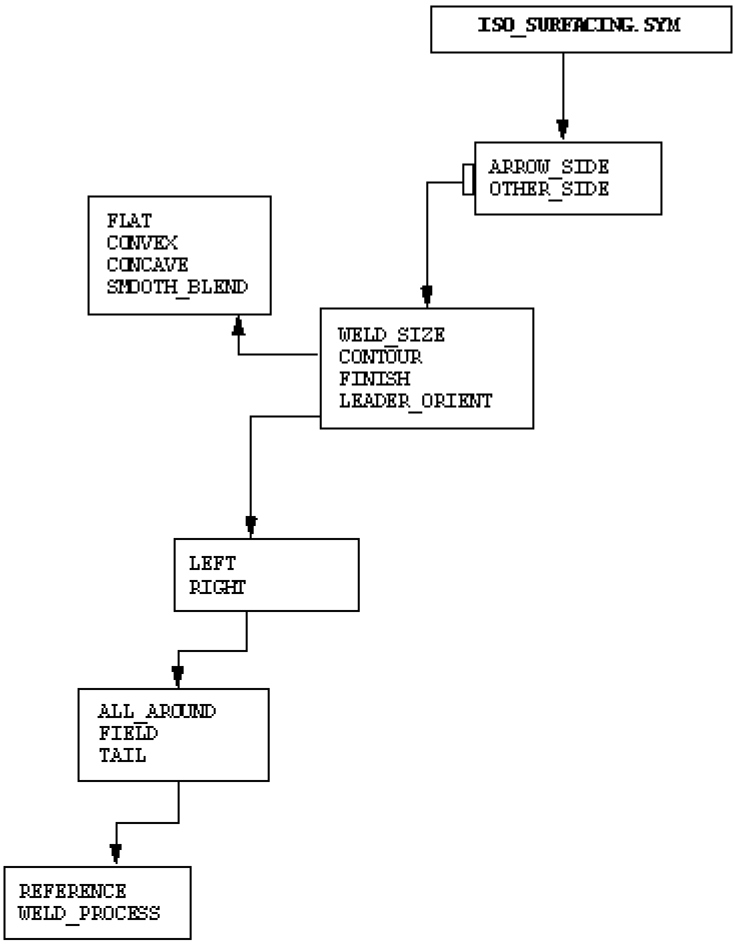
EXAMPLE 2

PICKS

1. other_side
2. weld_size
3. contour
4. leader_orient
5. flat
6. left
7. field
8. all_around
9. tail
10. reference



Symbol Definition Structure

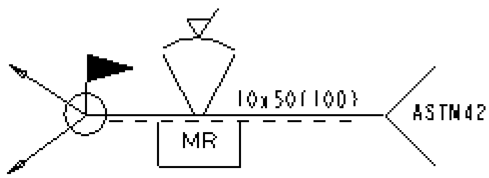


Steep-flanked V-butt Symbol: Steep_V.sym

EXAMPLE 1

PICKS

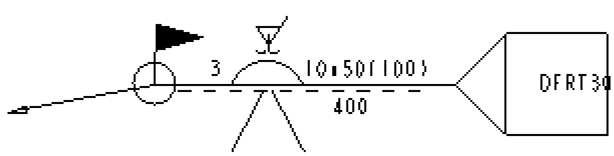
1. arrow_side
2. above_ref
3. style
4. contour
5. back_type
6. finish
7. leader_orient
8. intermittent
9. convex
10. strip
11. left
12. removable
13. field
14. all_around
15. tail
16. weld_process



EXAMPLE 2

PICKS

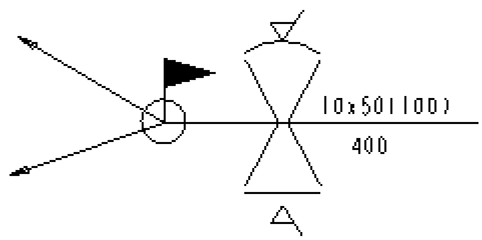
1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference



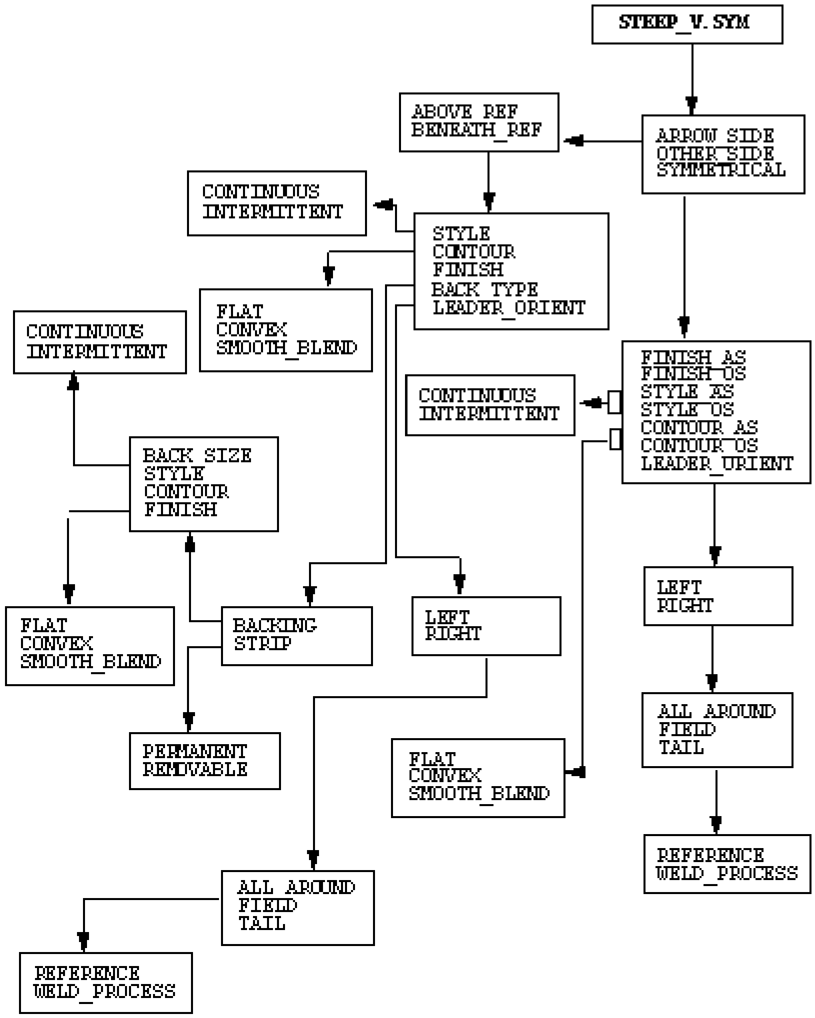
EXAMPLE 3

PICKS

1. symmetrical
2. style_as
3. style_os
4. contour_as
5. contour_os
6. finish_as
7. finish_os
8. leader_orient
9. intermittent
10. continuous
11. convex
12. flat
13. left
14. all_around
15. field



Symbol Definition Structure

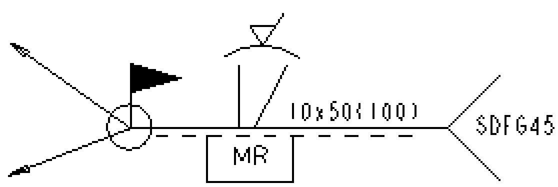


Steep-flanked Bevel Butt Symbol: Steep_Bevel.sym

EXAMPLE 1

PICKS

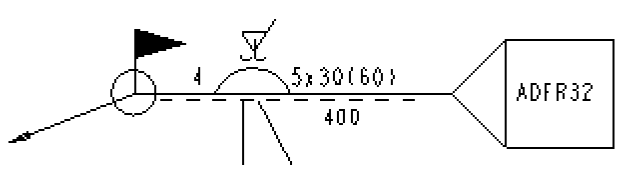
1. arrow_side
2. above_ref
3. style
4. contour
5. back_type
6. finish
7. leader_orient
8. intermittent
9. convex
10. strip
11. left
12. removable
13. field
14. all_around
15. tail
16. weld_process



EXAMPLE 2

PICKS

1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference

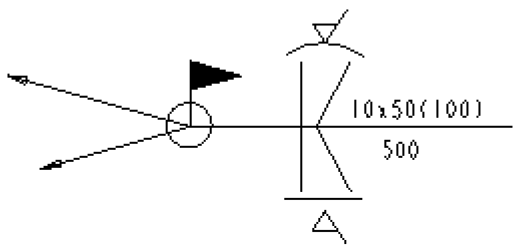


EXAMPLE 3

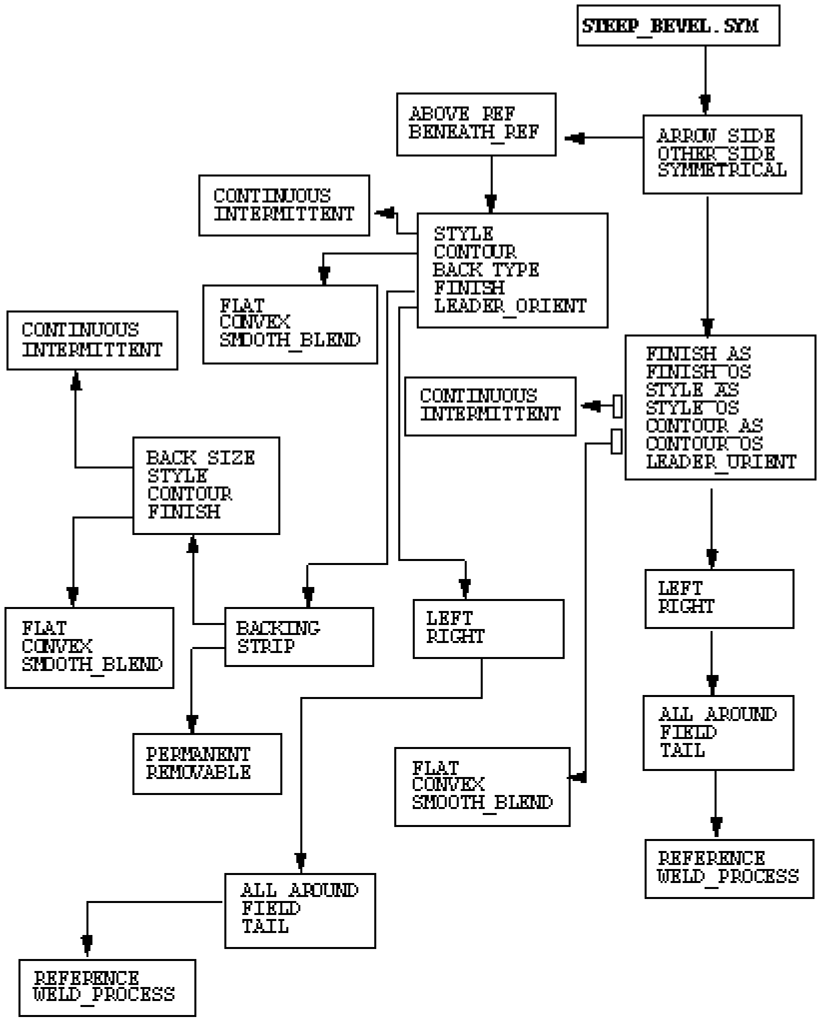
PICKS

1. symmetrical
2. style_as
3. style_os
4. contour_as
5. contour_os
6. finish_as
7. finish_os
8. leader_orient
9. intermittent

11. convex
12. flat
13. left
14. all_around
15. field



Symbol Definition Structure

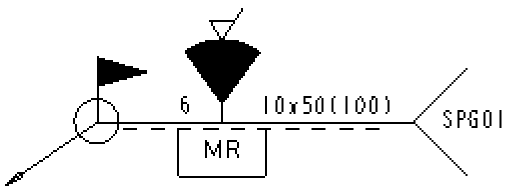


V-butt with Broad Root Face Symbol: Broad_Root_V.sym

EXAMPLE 1

PICKS

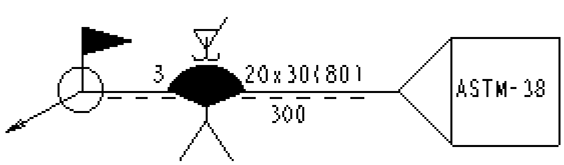
1. arrow_side
2. above_ref
3. weld_size
4. style
5. contour
6. back_type
7. finish
8. leader_orient
9. intermittent
10. convex
11. strip
12. left
13. removable
14. field
15. all_around
16. tail
17. weld_process



EXAMPLE 2

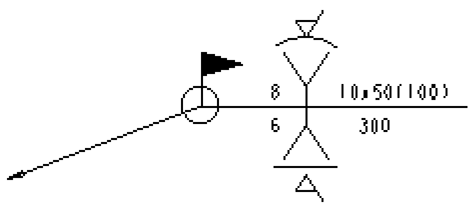
PICKS

1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. field
18. all_around
19. tail
20. reference

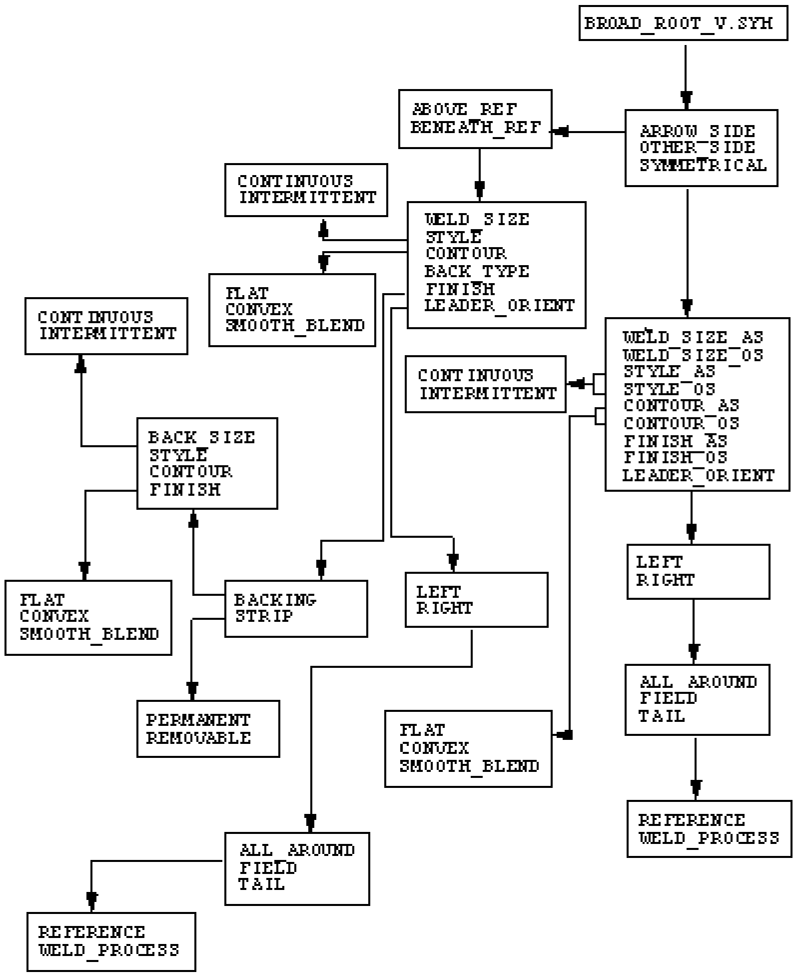


EXAMPLE 3

PICKS	
1. symmetrical	11. intermittent
2. weld_size_as	12. continuous
3. weld_size_os	13. convex
4. style_as	14. flat
5. style_os	15. left
6. contour_as	16. all_around
7. contour_os	17. field
8. finish_as	
9. finish_os	
10. leader_orient	



Symbol Definition Structure

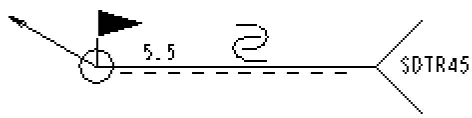


Fold Joint Symbol: Fold_Joint.sym

EXAMPLE 1

PICKS

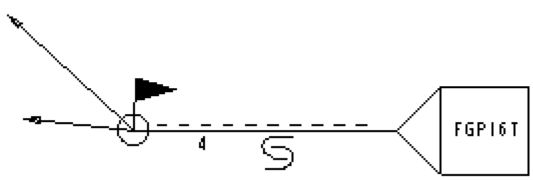
1. arrow_side
2. weld_size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. weld_process



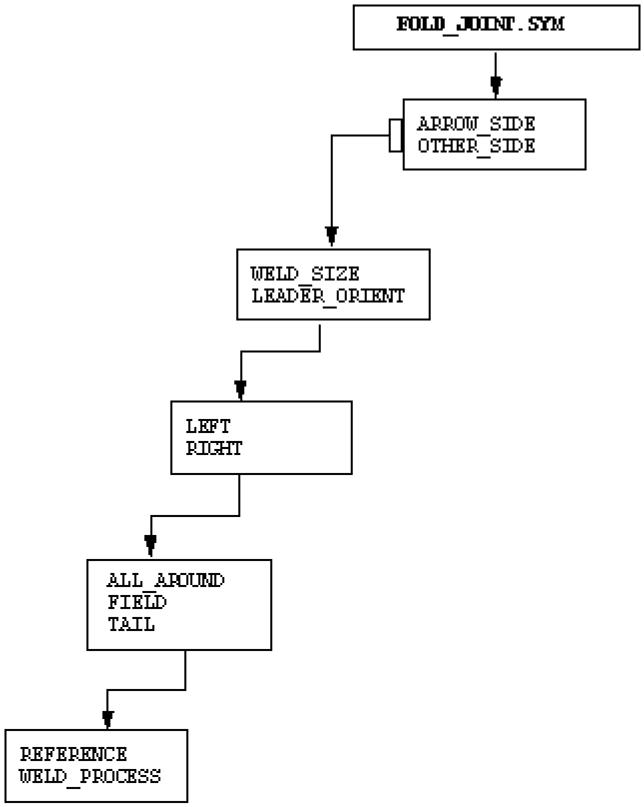
EXAMPLE 2

PICKS

1. other_side
2. weld_size
3. leader_orient
4. left
5. field
6. all_around
7. tail
8. reference



Symbol Definition Structure

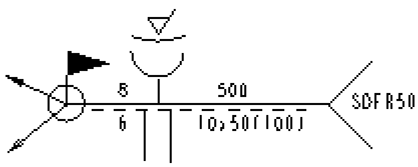


Combine Symbol: Combine.sym

EXAMPLE 1

PICKS

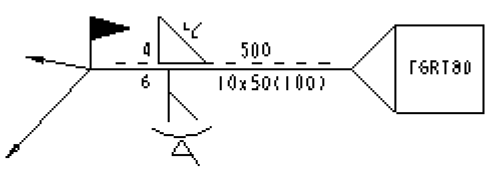
1. as_above_ref
2. weld_type_as
3. weld_type_os
4. weld_size_as
5. weld_size_os
6. weld_length_as
7. weld_length_os
8. leader_orient
9. u_weld_as
10. contour_as
11. machine_as
12. concave
13. square_os
14. contour_os
15. flat
16. continuous
17. intermittent
18. left
19. all_around
20. field
21. tail
22. weld_process



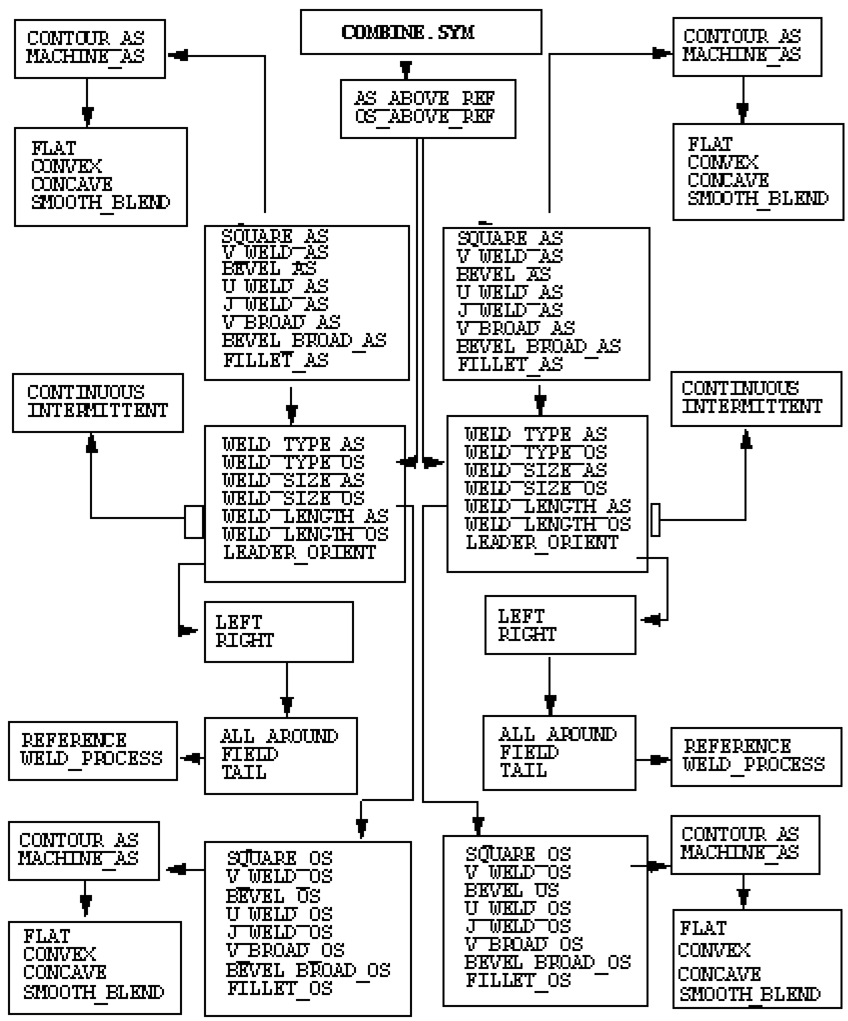
EXAMPLE 2

PICKS

1. os_above_ref
2. weld_type_as
3. weld_type_os
4. weld_size_as
5. weld_size_os
6. weld_length_as
7. weld_length_os
8. leader_orient
9. bevel_broad_as
10. contour_as
11. machine_as
12. convex
13. fillet_os
14. contour_os
15. smooth_blend
16. intermittent
17. continuous
18. left
19. field
20. tail
21. reference



Symbol Definition Structure

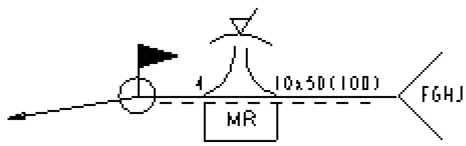


Edge Flanged Symbol: Iso_Edge_Flange.sym

EXAMPLE 1

PICKS

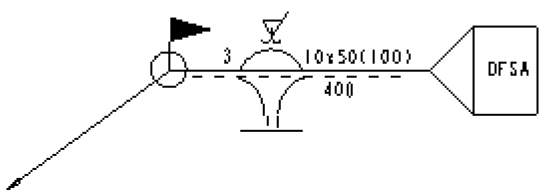
1. arrow_side
2. above_ref
3. weld_size
4. style
5. contour
6. back_type
7. finish
8. leader_orient
9. intermittent
10. convex
11. strip
12. left
13. removable
14. field
15. all_around
16. tail
17. weld_process



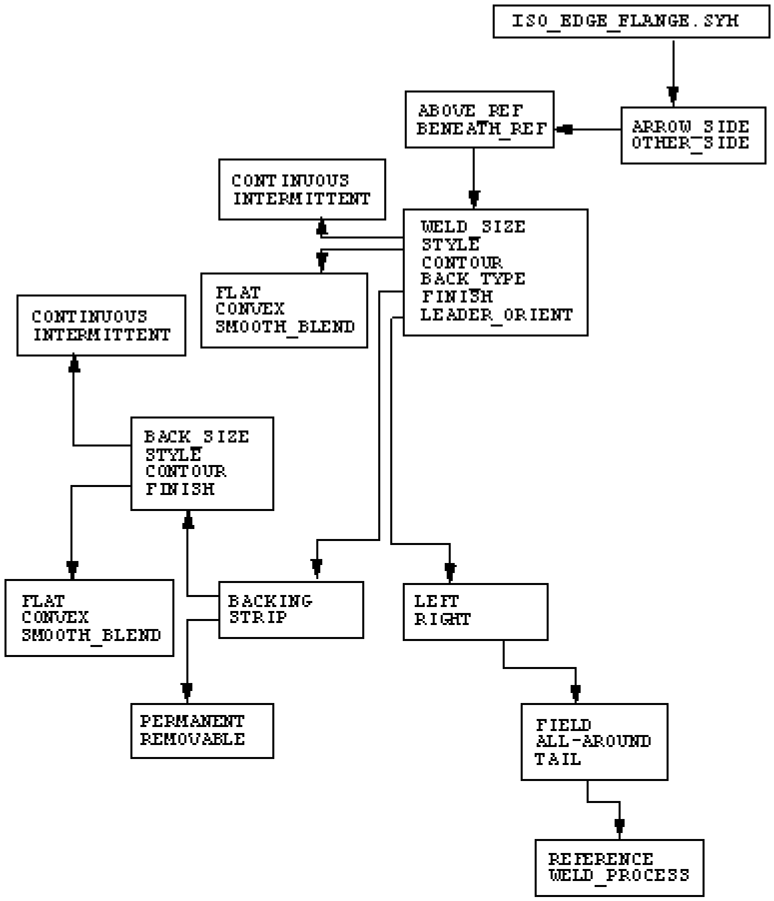
EXAMPLE 2

PICKS

1. other_side
2. beneath_ref
3. style
4. contour
5. back_type
6. leader_orient
7. continuous
8. flat
9. backing
10. back_size
11. style
12. contour
13. finish
14. intermittent
15. smooth_blend
16. left
17. all_around
18. field
19. tail
20. reference



Symbol Definition Structure

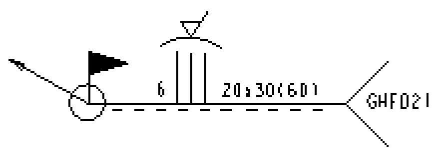


Edge Symbol: Edge_Weld.sym

EXAMPLE 1

PICKS

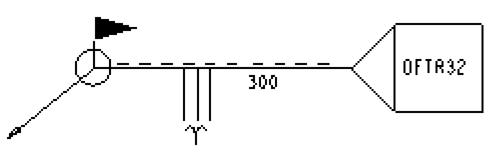
1. arrow_side
2. weld_size
3. style
4. contour
5. finish
6. leader_orient
7. intermittent
8. convex
9. left
10. field
11. all_around
12. tail
13. weld_process



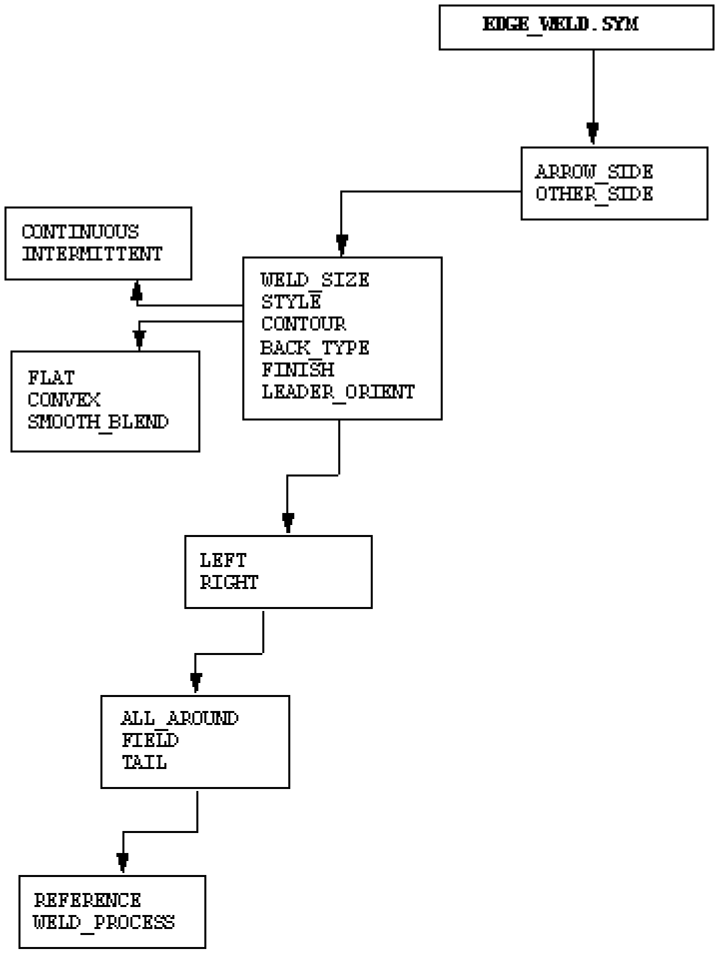
EXAMPLE 2

PICKS

1. other_side
2. style
3. contour
4. leader_orient
5. continuous
6. smooth_blend
7. left
8. field
9. all_around
10. tail
11. reference



Symbol Definition Structure



To Create a User-Defined Weld Symbol

Redefine an existing system weld symbol by doing one or all of the following:

- Add as many copies of variable texts as you want.
- Change the default values of variable texts.
- Add and delete as many notes and entities as you want, and place new ones in any group (or in no group at all).
- Redefine the cosmetics of existing notes and entities.

1. Save the symbol by assigning a new temporary name.
2. Move the original (system-supplied) symbol from the system weld library to another directory or rename it. The system weld symbol libraries are located in the installation directory path <install_dir>/symbols/library_syms/weldsymlib.
3. Move the new user-defined symbol into the system weld library and assign it the same name as the original.

Surface Finish Symbols

About Surface Finish Symbols

You can add surface finish symbols to a drawing using standard surface finish symbols available in the *//symbols/surffins* directory, or you can create and save your own surface finish symbols. The standard symbols are shown in the following table.

	Generic	**Machined**	**Unmachined**
no_value			
standard	 value	 value	 value

Surface Finishes for Drawings

Pro/ENGINEER reflects drawing surface finishes only in the drawing. It supplies a generic surface finish symbol that consists of building blocks, or groups. To create a desired instance, you must select any groups that you want to include in the symbol and specify required information. The generic surface finish symbol is located in the system symbols area.

Surface finishes are associated with surfaces in the part, not the entities or views in the drawing. Each surface symbol applies to the entire surface. When you specify a surface finish for a surface that already has one, Pro/ENGINEER redefines the surface finish information in the part and replaces the old symbol with the new one. Just as you cannot display the same dimension in two different views, in Pro/ENGINEER, you cannot display the same surface finish in two views.

If you create and add your own surface finish symbols, specify their location by setting the configuration file option `pro_surface_finish_dir`.

Note: you cannot attach surface finish symbols to parts intersected by an assembly feature.

The INST ATTACH Menu

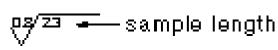
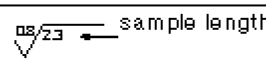
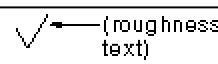
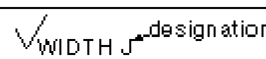
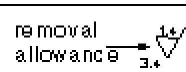
- **Leader**—Creates the symbol with a leader. Click commands from the ATTACH TYPE menu .
- **Entity**—Attaches the symbol to an entity (model edge or draft geometry).
- **Normal**—Attaches the symbol to an edge, entity, or dimension. Positions it with the vertical reference pointing up. When attached to an edge at an angle, a yellow arrow indicates To position it normal to the selected entity.
- **No Leader**—Creates a symbol that is unattached.

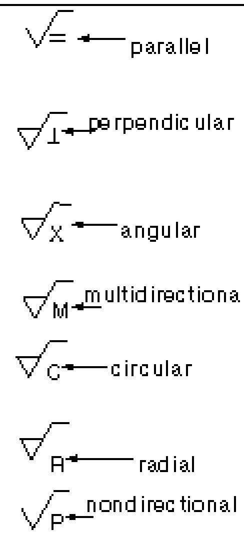
- **Offset**—Creates a symbol without leaders that is placed relative to a detail entity.

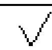
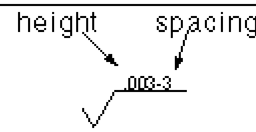
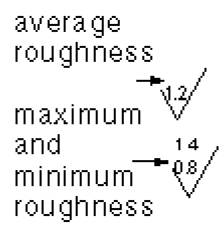
Valid Surface Finish Symbol Groups

The following table describes the groups that you can use to create a surface finish symbol. The first column indicates whether it is an ANSI or ISO symbol, or both.

Creating a Surface Finish Symbol

ISO/ ANSI	GROUPS	DESCRIPTION	ILLUSTRATION
ANSI ISO	LEADER	Used to create a symbol leader.	
ISO ANSI	MACHINED	Material removal by machining is required.	
ISO ANSI	NO_REMOVAL	Material removal prohibited.	
ISO only	PROD_METHOD	Text used for specifying production method.	
ANSI only	ROUGH_ SPACE	Required maximum roughness spacing (mm or inch).	
ISO ANSI	SAMPLE_LEG	Roughness sampling length or cutoff rating (mm or inch).	
ISO only	OTHER_ROUGH	Text used for specifying other roughness.	
ANSI only	DESIGNATION	Text used for special designations.	
ISO ANSI	REMOVAL ALLOW	Material removal by machining that is required to produce the surface (mm or inch).	

ISO/ ANSI	GROUPS	DESCRIPTION	ILLUSTRATION
ISO ANSI	LAY	Designates direction of lay. The system supports the following types: PARALLEL—Approximately parallel to the line representing the surface to which the symbol is applied. PERP—Approximately perpendicular to the line representing the surface to which the symbol is applied. ANGULAR—Angular in both directions to the line representing the surface to which the symbol is applied. MULTI_DIR—Multidirectional. CIRCULAR—Approximately circular relative to the center of the surface to which the symbol is applied. RADIAL—Approximately radial to the center of the surface to which the symbol is applied. NON_DIR—Nondirectional, or protuberant.	

ISO/ ANSI	**GROUPS**	**DESCRIPTION**	**ILLUSTRATION**
ISO ANSI	UNSPECIFIED	Basic surface texture symbol. You can produce the surface by any method, except when the bar or circle is specified.	
ANSI only	WAVINESS	Specified waviness (mm or inch): WAVE_HEIGHT— Maximum waviness height. WAVE_SPACE—Maximum waviness spacing.	
ISO ANSI	ROUGHNESS	Roughness rating indicates permissible roughness range (mm or inch). The system supports these types: AVERAGE—Roughness average rating. MAX_MIN—Maximum and minimum roughness average values.	

Defining Symbol Attributes

Controlling Symbol Instance Height

To relate the height of a symbol to a model view that was defined using the **On Item** or **Free Note** placement attributes, select **Variable - Model Units** from **Symbol Definition Attributes** dialog box. The system then remembers the height of instances of the symbol in model units. If you change the view scale of the model units, it automatically adjusts the symbol's visible size to maintain a constant proportional relationship with the model.

You can proportion symbols, such as diameter symbols, to the text that follows. In such cases, define the symbol attributes by selecting **Variable - Text Related** from the Symbol Instance dialog box. However, if a symbol (for example, a diameter symbol) does not contain any text to which you can make it proportional, you can create symbol text with a blank line by doing one of the following:

- Create a one-line note in the symbol and change whatever text you had in the note to a single blank line (make sure to retain the braces and other special characters). To include this note when defining the symbol, choose **Pick Box** and select **Variable - Text Related** from the Symbol Instance dialog box to define the symbol.
- Include a variable text note when defining the symbol. Select **Variable - Text Related** from the Symbol Instance dialog box to define the symbol. When you create an instance of this symbol, backspace over the default text in the **Var Text** box so that text does not appear in the symbol.

Note: If you change the drawing scale, symbols with a fixed size will not change. To change the symbol size you should redefine the symbol attributes.

To Control Mirror Properties of a Simple Symbol

You can control the way in which a symbol and its notes appear and reorient themselves upon mirroring and rotation by using one of the following methods:

- Set the drawing setup file option `sym_rotate_note_center` to `yes` (the default value). The system rotates the symbol note as if its origin were in the middle of the height of the text, rather than at the bottom of the text. If you set it to "no," the system rotates the text by rotating its origin point, as it is. Changing the value of this setup option changes the position of existing rotated texts when you repaint.
- A symbol, when mirrored, may
  - retain its original orientation in the mirror image,
  - mirror its geometry only
  - mirror its text only, or,
  - mirror both
- These properties are retained in the symbol definition. Use the **Symbol Definition Attributes** dialog box (when defining or redefining the symbol) to set them.

About Variable Text Parameters

Normally, if you include an ampersand (&) before a parameter name in a note, the note searches for the parameter in the drawing, and if found, displays the parameter value. Notes in symbols also do this, but you must set up the note in the symbol definition differently than you would a note in the drawing. You must enter the parameter as a value for a string of variable text.

If you insert text notes with a backslash before and after them (for example `\notetext\`) in the symbol definition, these notes become "variable text" notes. They appear in the **Var Text** tab of the **Symbol Definition Attributes** dialog box.

Variable text means that you may enter a selection of values in the preset values text box for each variable text note. For example, if your note had the text

"`Drafter Name \names\`,"

"names" would appear in the left window of the Var Text tab. You could enter a list of employees names in the right side area to select from when the instance is placed. When you place the instance, you can select a name to follow "Drafter Name" from a drop down list in the **Custom Drawing Symbol** dialog box. (Select the symbol and click **Properties** on the right mouse button shortcut menu to show the dialog box.)

You may also enter a parameter as the preset value for the variable text, for example "&draftername". If there is a "draftername" parameter in the drawing, the symbol note will return its value.

Setting Symbol Parameters

About Symbol Parameters

You can include two types of parameters in a symbol:

- Node parameters identifying nodes (if nodes are present in the symbol).
- Symbol definition parameters, or the top parameter set, identifying the symbol. The system replaces them with the corresponding information when you add the symbol to the drawing. Fixed text appears the same for any instance of the symbol. You can use the following parameters in a symbol definition:
  - Any of the system parameters for drawings
  - Any user-defined parameters
  - &dwg_name
  - &model_name
  - &scale

  Note: You cannot use a note parameter for the value of variable text.

Depending on the application, a symbol includes the following categories of parameters:

- Required
- Optional, system-defined
- Optional, user-defined

If an application uses a symbol with required parameters, but they are missing, the system issues an error message and you can edit the symbol parameter file.

Pro/ENGINEER uses symbols containing an individual parameter set to create electrical diagrams. You can define Pro/DIAGRAM symbols in Drawing mode and Diagram mode. However, if you want to use them in Pro/DIAGRAM, you must define them as components or connectors by providing the parameter set appropriate for the type of object the symbol represents (component or connector).

To Generate a Parameter Set

To create a parameter set, you can do one of the following:

- Generate a set of default parameters by reading in data (system- or user-defined).
- Type parameter names and their values in a table using the Pro/TABLE environment.

To Generate a Set of Default Parameters

You can generate a set of default connector or component parameters to symbols. This is a Pro/Diagram specific function.

1. Click **Format > Symbol Gallery**. The **SYM Gallery** menu appears.
2. Click **Symbol > Refine** to locate the symbol to assign default parameters.
3. Click **SYMBOL EDIT > Parameters** > **Read**.
4. Using the READ SYM PRM menu, do one of the following:
    - Click **Comp Default** and retrieve a system set of default parameters for components.
    - Click **Conn Default** and retrieve a system set of default parameters for connectors.
    - Click **Other** and retrieve a user-specified file (with the extension ".spm") containing the appropriate parameters.

To Edit Symbol Parameters

Once you have created a parameter set, you can modify symbol and node parameters by editing the parameter file using the Pro/TABLE editor. You can change the parameter value, add user-defined parameters, or delete parameters. If the system encounters errors after you have edited the parameter file, it displays an Information Window indicating the type of errors. You can use commands in the REEDIT menu to resolve the discrepancies.

To View Symbol Parameters

Using the **Show** command in the SYM PARAMS menu, you can view the symbol definition parameter set.

1. Click **Format > Symbol Gallery**. The **SYMBOL** menu appears.
2. Click **symbol > Refine** to locate the symbol to view parameters.
3. Click **SYMBOL EDIT > Parameters > Show.**

To Store a Symbol Parameter File

While working with a symbol definition, you can save the parameter set to a file in your working directory, and read it in later to create similar parameter sets for other symbol definitions.

1. Click **Format > Symbol Gallery**. The **SYMBOL** menu appears.
2. Click **Symbol > Refine** to locate the symbol to store parameters.
3. Click **SYMBOL EDIT > Parameters > Write.**
4. Type the filename without an extension. The system stores the file with the name "filename.spm" to disk.
5. If the name that you typed for the parameter file already exists in your working directory, type [`Y`] to replace the existing file or [`N`] to avoid overwriting; then type a new filename.

Adding Nodes to Symbols

About Adding Nodes

While defining symbols or after defining them, you can add nodes (also referred to as pins) as valid wire attach points on components and connectors. You can also generate, modify, store, and view parameter sets.

If you include a node in a symbol, the system identifies it by placing its name in a note. When you create nodes during symbol definition, it automatically adds the PIN command and node name value to the symbol parameter file. The drawing setup file option `node_radius` controls the display of nodes in symbols.

To Create a Node

1. Click **Format > Symbol Gallery > Redefine** to retrieve the symbol to include a node. The symbol edit window opens.
2. In the **SYM EDIT** dialog box, click **Insert > Node**. The **SYMBOL NODE** menu appears.
3. Click **Make Node**.
4. Type the node name in numeric or alphanumeric form.
5. Select the node location on the symbol. A green dot indicating a node with the node name appears on the symbol.

To Blank Parameters in Node Notes

By default, the system shows node notes when it creates them. To blank all node notes:

1. Click **Format > Symbol Gallery > Redefine** to retrieve the symbol that includes a node.
2. Click **Insert > Node**. The **SYMBOL NODE** menu appears.

3. Click **Hide Node Notes**. To redisplay them, choose **Show Node Notes**.

Referencing Parameters in Node Notes

You can edit node notes as text to include the following references to other parameters:

- Node name, for example, `&node_name`
- Node parameters, for example, `&signal_name`
- Top-level symbol parameters, for example, `&ref_des`
- Any valid drawing parameters included as labels, for example `&scale`

  Note: You can reference only those symbols to which you have assigned values in the symbol parameter set. If you delete a parameter that a note references, "***" appears in the note instead of the missing parameter value.

Manipulating Symbol Instances

About Manipulating Symbol Instances

Just as you might modify the value or location of drawing dimensions, you can modify symbol instances in your drawings. You can alter any of the symbol attributes as necessary.

You can apply any changes to the symbols individually or you can update all instances throughout the entire drawing by redefining the symbol definition.

The following actions are typically applied to individual symbol instances:

- Adjust the symbol height
- Change the angle at which the symbol is displayed
- Reposition the origin of the symbol
- Modify the symbol group attributes
- Edit the variable text values
- Convert the instance to draft entities
- Reattach instances and add new leader lines

To ensure that the changes are available for another drawing session, you should modify and save the symbol definition within the symbol directory.

To Modify a Symbol Instance

1. Select the symbol instance in the drawing.
2. Right-click and click **Properties** from shortcut menu. The Custom Drawing Symbol dialog box opens.
3. Make any of the following modifications as necessary:

- The leader or placement type.
- The symbol height (if it is allowed in the definition).
- The rotation angle.
- The color or the origin.
- The groups displayed (If groups are set up in the definition).
- The variable text values (if variable text is set up in the definition). In the Symbol Instance dialog box, click the **Variable Text** tab. Preset values are listed in a drop down list next to each parameter. The predefined values are part of the symbol definition.

4. When you are finished, click OK to update the display of the instance.

To Edit the Symbol Leader Attachment

You can edit leader attachment references and display or add additional attachment to symbol instances in your drawings.

1. Select the symbol whose leader attachment you wish to modify. Be sure to select the symbol and not just the symbol leader.
2. Right-click and click **Edit Attachment** from the shortcut menu. The menu manager opens.

- To move the attachment point to another reference, click **Change Ref**.
- To add a new attachment reference for the symbol, click **Add Ref**.
- To maintain the same references click **Same Ref**. This maintains the current status of attachment.

3. If you are adding or changing an attachment, you need to define the attachment type. First, you should define the references for the symbol:

- On Entity
- On Surface
- Free Point
- Midpoint
- Intersect

4. Define the physical attributes for the leader at the attachment point:

- Arrow Head
- Dot
- Filled Dot
- No Arrow

- Slash
- Integral
- Box
- Filled Box
- Double Arrow

5. After defining the leader attachment, click Done. The leader is modified.

To Convert Symbols to Draft Entities

You can convert a drawing symbol to a draft entity, which enables you to...

1. Select the symbol to convert.
2. Click **Edit > Convert to draft entities**. Any leaders are removed and the symbol is reduced to independent graphic elements and text.

To Relate a Symbol Instance to Drawing Objects

You can relate a symbol to a dimension, note, or another symbol, so that the symbol follows the dimension throughout the drawing, thus making sure the symbol and related object stay together.

After relating the symbol, you can adjust the offset the distance by dragging the symbol to a new location.

To Relate Symbols Using Groups

1. After placing the symbol in the drawing, select the symbol you want to relate to another object.
2. Click **Edit > Group > Relate to Object**.
3. Select the dimension, note, or symbol to relate the symbol to. The symbol is now related.

To Relate Symbols Using Placement References

In order to relate a symbol using placement references, you must select Free as one of placement type attributes in the symbol definition.

1. After placing the symbol in the drawing, select the symbol.
2. Right-click and click **Properties**. The Custom Drawing Symbol dialog box opens.
3. Under **Placement**, select **Offset** from the **Type** box.
4. On the drawing, select the dimension to relate the symbol to. If necessary, drag the dialog box to see the dimensions on the drawing. The dimension highlights.
5. To reposition the symbol, middle-click on the desired symbol location. The symbol relocates and highlights.

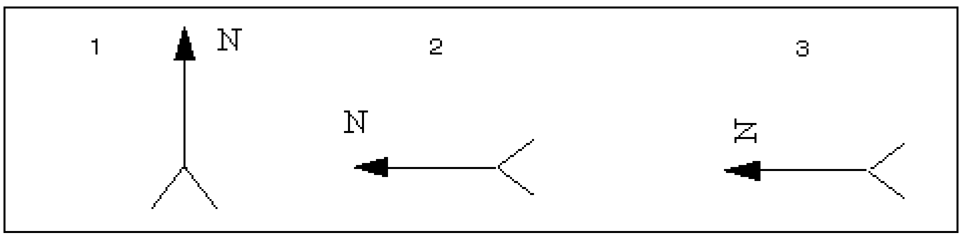
6. Click **OK** on the dialog box. The symbol is related to the dimension and will maintain the offset distance wherever the dimension is located.

To Control Text During Symbol Rotation

If you are rotating a symbol you may want to set controls to manage the orientation of any text contained within the symbol. By doing so you can ensure that the text meets the appropriate standards and is readable.

You can control the text during symbol rotation in the following ways:

- Define the text behavior within the symbol definition. By setting the **Fixed Text Angle** symbol attribute, the text will maintain its orientation no matter how the symbol is positioned.

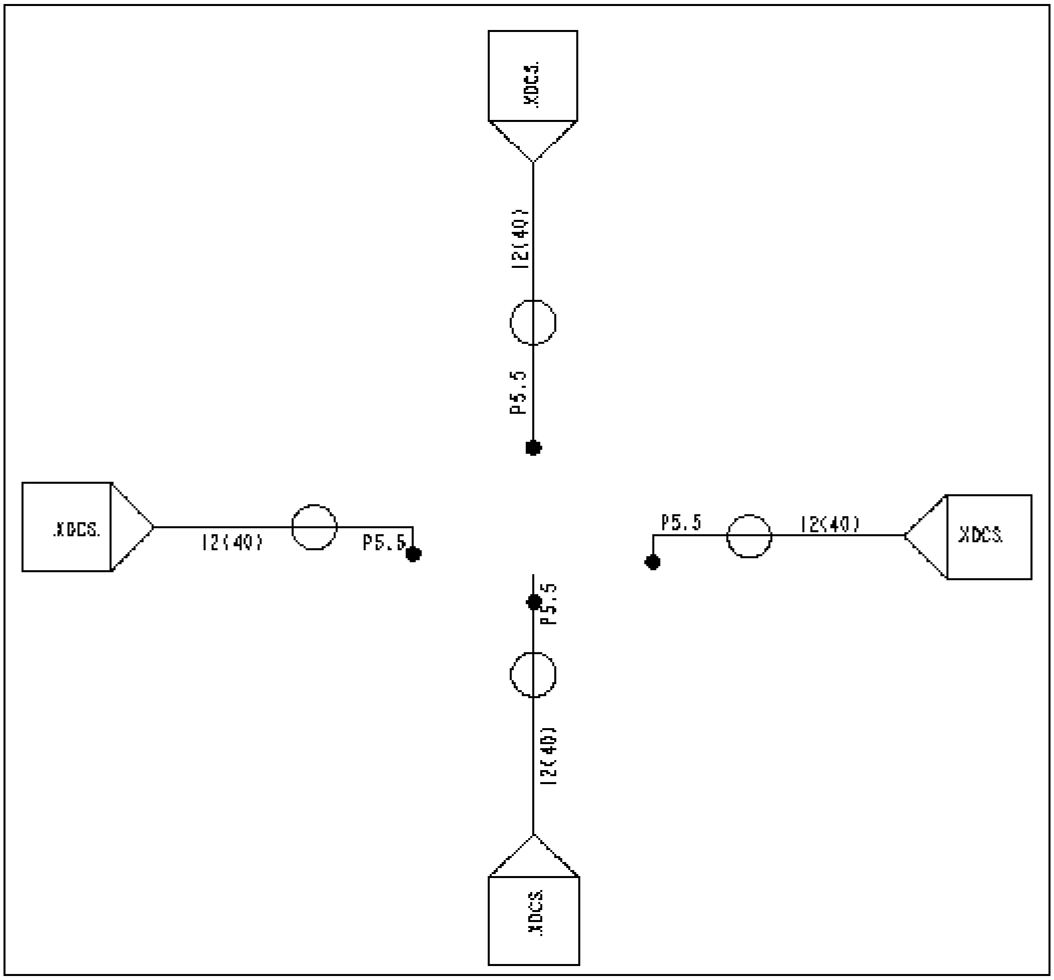


1. The symbol in its original position.

2. The symbol created when using the **Fixed Text Angle** check box.

3. The symbol created when using **Fixed Text Angle** and rotating the symbol during instance creation.

- Set the `sym_flip_rotated_text` drawing setup file option, which flips the text to the opposite side of a rotated symbol. By setting `sym_flip_rotated_text` to `yes` (the default is `no`), while the symbol orientation is +/- 90 degrees, any text defined in the **Attributes** box of the Symbol Definition Attributes dialog box menu simultaneously rotates and flips with the symbol.



Updating Symbol Instances

As you dimension and detail your models you may need to redefine or retrieve an alternate version of a symbol. And, while it is not necessary, you may also want to update all instances of that symbol to reflect the symbol change throughout the current drawing. If you do not update the symbol in the current drawing, any newly added instances will maintain the symbol's old attributes.

The process of updating symbols is largely dependent upon the relationship between the symbol and the directory in which it is stored. Similar to the way that the Windows operating system does not consider two files with the same name to be the same file if they are stored in different directories, Pro/ENGINEER does not consider two symbols with the same name but different paths to be the same. For example:

Symbol Directory	**Symbol Name**	**Description**
symbol directory (`pro_symbol_`	`ptc.sym`	If the symbol is retrieved from the symbol directory set by the `pro_symbol_dir` configuration

Symbol Directory	Symbol Name	Description
dir)		option its full name is simply <symbol_name>.sym.
C:\PTC\files\symbols	C:\PTC\files\symbols\ptc.sym	If a symbol is retrieved from a location other than the symbol directory, its full name is <full_path_to_directory>\<symbol_name>.sym.

If the existing symbol instances' full name does not match the full name of the symbol being retrieved, Pro/ENGINEER does not prompt you to update the existing symbol instances. Instead, the new symbol instance is given the name <symbol_name>(2). This symbol renaming can be avoided by retrieving new symbol instances from the same symbol directory.

If you do not remember the symbol name or storage directory, click **Format > Symbol Gallery > Show Name** and select the desired symbol instance. The full symbol name is displayed in the message area.

Note: Pro/ENGINEER is case sensitive when determining whether the symbol names match; C:\MyFiles is not the same as C:\myfiles.

To Update Symbols in Drawings

If you change or modify a symbol instance, you may also want to update all instances of that symbol to reflect the changes throughout the current drawing. In order for Pro/ENGINEER to prompt you to update the symbol instances, you need to retrieve the new symbol from the same directory as the original symbol.

If the symbols are retrieved from separate directories the new symbol is renamed.

To Update Drawing Symbols Retrieved from the Same Directory

1. Click **Format > Symbol Gallery**. The SYM GALLERY menu appears in the menu manager.
2. Click **Redefine**. The GET SYMBOL menu appears.
3. Click **Retrieve**. The Open dialog box opens.
4. Browse to and open the desired symbol. If the symbol file names are the same, you are prompted to update all instances.
5. Type [Y] to update all instances of the symbol in the current drawing to the most recent version. The symbols are updated throughout the current drawing.

To Update Drawing Symbols Retrieved from Different Directories

If the existing symbol instances were retrieved from a different directory than the new symbol, there are a few alternate techniques available:

- Manually replace the existing symbol instances with the new instance.

- Copy the new symbol to the directory specified in the full name of the existing instances. For example, if the full name of the existing instances is `C:\PTC\files\symbols\ptc.sym`, copy the new version of `ptc.sym` to `C:\PTC\files\symbols`. Ensure that the new version of the symbol has a higher revision number than the old one. The revision of each symbol is shown after the `.sym` extension. For example, `ptc.sym.2` has a higher revision than `ptc.sym.1`. Then retrieve the new symbol instance.
- Write the existing symbol instances to disk in the symbol directory. Click **Format > Symbol Gallery > Write > Pick Inst** and then select an existing symbol instance. This will change the full name of the existing symbol instances to `<symbol_name>.sym`. Copy the new version of the symbol to the symbol directory, ensure that the new version of the symbol has a higher revision than the old one, and retrieve the new symbol instance.
- If the existing symbols were retrieved from the symbol directory and the new symbol is located elsewhere, set the symbol directory to the location of the new symbol before retrieving it.

About Dragging Symbols and Surface Finishes

When you drag a symbol or surface finish away from the edge of an entity to which they are attached, an extension line is created. However, the extension line is created only if the symbol, surface finish, or an arrowhead of a leader is attached normal to any of the following entities.

- Straight model edges (solid or surface)
- Straight draft entities
- Datum curves that are straight in 3D

Note: You cannot drag a symbol or surface finish away from an edge of an entity that is not straight.

When you drag the symbol or surface finish away from the edge of the entity, only the following placement types for a symbol, surface finish, and arrowhead of the leader support the creation of extension lines:

- On Entity
- Normal to Entity
- With Leader (On Entity)
- Tangent Leader (On Entity)
- Normal Leader (On Entity)

If you have a surface finish from a non-straight model curve that appears straight in the drawing, you can drag the surface finish only up to the boundary of the curve.

You can modify the line style of extension lines by setting the `leader_extension_font` drawing setup file option.

Datum Targets

About Drawing Datum Targets

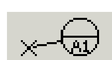
Use datum targets to indicate critical measurement points on the plot. The target is drawn inside a scalable circle with a leader. You can create a datum target specifying any set datum point, except one that has been created using **Offset Csys**. **Point** datum targets reference a selected set datum point on a surface or edge. **Diameter** datum targets contain a required diameter. **Box** datum targets contain the dimensions of the selected area while **Line** datum targets contain the datum name and a leader line pointing to the selected datum curve.

When a target is added, the lower part of the circle references the datum name, and adds an integer to note the instance of the target symbol; for example, DTM5 becomes DTM51, DTM52, etc.

Draft targets do not resequence themselves if you delete one of them. For example, if there are three datum targets (B1, B2, and B3), deleting B2 leaves B1 and B3, not B1 and B2.

Using the **Show/Erase** dialog box, you can show datum targets in multiple views.

To Show Datum Targets

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.
3. Under **Type**, click .
4. Use the following options in the **Options** tab to filter the datum targets you want to show in the drawing:
    - **Erased**—Show previously erased items.
    - **Never Shown**—Show items that are yet to be displayed in the drawing.
5. Under **Show By**, use the following options to define where you want to display the datum targets:
    - **Feature**—Show the datum targets for a particular feature on the drawing.
    - **Part**—Show the datum targets for a particular part on the drawing.
    - **View**—Show all the datum targets for the features and parts within a particular drawing view.
    - **Feature and View**—Show the datum targets for a feature that appears in several views in one selected view. For example, if you have a feature with datum targets that appears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to display it.
    - **Part and View**—Show the datum targets for a part that appears in several views in one selected view.

- **Show All**—Show all the datum targets within the drawing.

6. Select the appropriate feature, part, or view to display the datum targets. You may need to repaint the drawing for the datum targets to displayed. When the datum targets are displayed the **Preview** tab becomes available.

7. Of the datum targets previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:

   - **Sel to Keep**—Select the individual datum targets to show in the drawing. Any datum target not selected will be erased.
   - **Sel to Remove**—Select datum targets to remove from the drawing. Any datum target not selected will remain in the drawing.
   - **Accept All**—Keep all the previewed datum targets.
   - **Erase All**—Erase all the previewed datum targets.

   While you are showing the datum targets, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and click **Resume Show and Erase**.

   If you need to select multiple datum targets to keep or remove, you can do this by either holding down the `CTRL` key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.

8. When you are finished with the preview, click **OK** in the **Select** dialog box. The datum targets are displayed in the drawing.

9. Click **Close** to close the **Show/Erase** dialog box.

Note: The `show_preview_default` drawing setup file option lets you set the preview tab default to either select to keep or select to remove.

To Insert Datum Targets

1. Click **Insert** > **Draft Datum** > **Target**.

2. In the drawing, select a set reference datum. You can select draft datum planes and axes as well. The **TARGET** menu opens in the menu manager.

3. Do one of the following:

   - Click **Point** to put the referenced datum name in the lower half of the datum target symbol.
   - Click **Diameter** to select a dimension point for each target datum. The datum name is displayed in the lower half of the datum target symbol and

the dimension with the diameter symbol is displayed on the top half of the datum target symbol.

- Click **Box** to create a datum target with the datum name and the values of the selected dimension. The datum name is displayed in the lower half of the datum target symbol and the dimension values are shown on the top half of the datum target symbol.
- Click **Line** and select a datum curve to create a datum target with the datum name and a leader line pointing to the datum curve. This datum curve must be perpendicular to the set datum plane. The datum name is displayed in the lower half of the datum target symbol.

4. Using the **GET DTM SEL** menu, do one of the following:
   - Click **Select Point** to select a datum point in the drawing.
   - Click **Create Point** to create a new datum point.
   - Click **Select Curve** to select a datum curve in the drawing.
5. Click on the location point for the target.

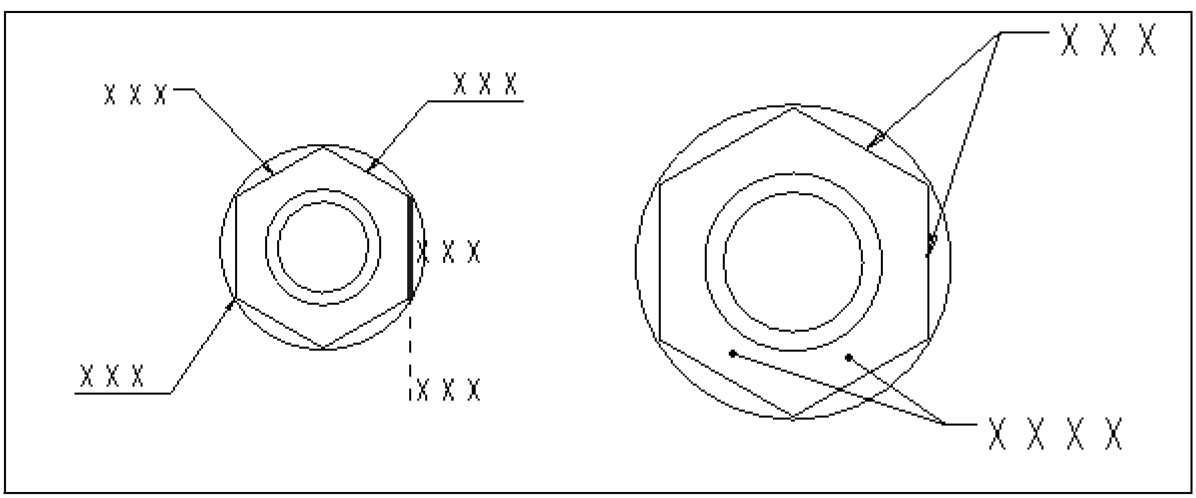
The target and leader are added to the drawing. You can move or resize the target symbol.

Text and Notes

About Drawing Notes

A text note can be grouped with a dimension, attached with or without a leader to one or more edges on the model, or located "free."

Pro/ENGINEER creates note text using the default values (such as height, font) specified in the **Format > Text Style Gallery** dialog box. To format the text and style of a selected note, click **Properties** from the right mouse button shortcut menu.



Above left: Some note types - clockwise from left: Leader on entity, ISO leader on entity, no leader on entity (highlighted face) no leader on entity and moved from entity, ISO leader on vertex. Above right clockwise: double leaders on item, double leaders on surface.

When you insert a note, you specify the following characteristics in the Note Types menu manager:

- Leader type (or no leader)
- Note angle, (horizontal, vertical, angular, normal to the leader or tangent to the leader)
- Justification (Left, right, center.)
- Attachment location (on entity, on surface, free point, midpoint or intersection)
- Arrowhead style (dot, arrow, box etc.)

After you create your first note, Pro/ENGINEER creates subsequent notes using the attributes that you specified previously.

To Show Model Notes in a Drawing

Using the **Show/Erase** dialog box or the Model Tree, you can show model notes in drawings.

Note: Location within drawings is dependent upon their actual placement in the model. Unplaced notes appear in a default location when you show model notes in the drawing. You can use the text of a model note to create a drawing note by reading in a model note from a file.

1. Click **View** > **Show and Erase**. The **Show/Erase** dialog box opens.
2. Click **Show**.
3. Under **Type**, click 

.
4. Use the following options in the **Options** tab to filter the notes that you want to show in the drawing:
   - **Erased**—Show previously erased items.
   - **Never Shown**—Show items that are yet to be displayed in the drawing.
5. Under **Show By**, use the following options to define where you want to display the notes:
   - **Feature**—Show the notes for a particular feature on the drawing.
   - **Part**—Show the notes for a particular part on the drawing.
   - **View**—Show all the notes for the features and parts within a particular drawing view.
   - **Feature and View**—Show the notes for a feature that appears in several views in one selected view. For example, if you have a feature with a note

that appears in several projection views, you can use **Feature and View** to select the feature in the view in which you want to display it.

  - **Part and View**—Show the note for a part that appears in several views in one selected view.
  - **Show All**—Show all the notes within the drawing.

6. Select the appropriate feature, part, or view to display the notes. You may need to repaint the drawing for the notes to displayed. When the notes are displayed the **Preview** tab becomes available.
7. Of the notes previewed in the drawing, there may be some that you do not want to show. Use the following options in the **Preview** tab to filter them:
   - **Sel to Keep**—Select the individual notes to show in the drawing. Any note not selected will be erased.
   - **Sel to Remove**—Select notes to remove from the drawing. Any note not selected will remain in the drawing.
   - **Accept All**—Keep all the previewed notes.
   - **Erase All**—Erase all the previewed notes.

   While you are showing the notes, items may overlap on the drawing. If this occurs, you can pause the show and erase tool and move the drawing items to enable you to see and correctly select the items to show or erase. To do this, right-click anywhere on the drawing and click **Pause Show and Erase**. Then move the drawing items within the view as necessary. Before continuing with the show and erase process, you must right-click and click **Resume Show and Erase**.

   If you need to select multiple notes to keep or remove, you can do this by either holding down the CTRL key while selecting or using region selection. If you do not select all the necessary items at this time, you will have to repeat the show and erase process.
8. When you are finished with the preview, click **OK** in the **Select** dialog box. The notes are displayed in the drawing.
9. Click **Close** to close the **Show/Erase** dialog box.

Note: The `show_preview_default` drawing setup file option lets you set the preview tab default to either select to keep or select to remove.

To Insert a Drawing Note

When you insert a note, you click through a series of options to set up all the note properties such as attachment type and location, leader style, and text position. Your selections are retained as defaults for subsequent notes, until you change them.

1. Click **Insert** > **Note**. The **Note Types** menu appears on the Menu Manager. Use the **Note Types** menu to specify the note appearance and contents source.

2. Select one of the following option for the note leader:

- **No Leader**—Bypass any leader setup options and are prompted only for the note text and position on the sheet.
- **With Leader**— Leader attached to a specified point. You are prompted for the attachment style, arrow style, and so on.
- **ISO Leader**—ISO style leader (underlines text). Same as above, with ISO style.
- **On Item**—Note is directly attached to the selected item. You bypass any leader setup options and are prompted only for the note text and an item to which to attach the note directly.
- **Offset**—A note is grouped with a selected entity. You can bypass the leader setup options and are prompted only for the note text and a reference entity to offset the text. You can select one of the following reference entities to offset the note text:
    - Draft datum point or shown 3D datum point
    - Perpendicular axis center point
    - Non-perpendicular axis end point
    - Shown axis and draft axis
    - Dimension
    - Dimension arrow
    - Geometric tolerance
    - Note
    - Symbol instance
    - Reference dimension

  When you move the reference entity, the offset note associated with it also moves. When you move a note independent of the reference entity, the offset value is reset.

3. Select **Enter** to enter the note text from the keyboard or select **File** to open from a file.
4. If you have selected the With Leader option, select an orientation for the leader from the following options:

- **Standard**—Uses the default leader type.
- **Normal Leader**—Makes the leader normal to the entity. In this case, the note can have only one leader.
- **Tangent Leader**—Makes the leader tangent to the entity. In this case, the note can have only one leader.

5. Set the alignment of the note text using one of the following options:

- **Left**
- **Center**
- **Right**
- **Default**

6. Select a style for the note text. By default the note is created with the current style, or the last style that you used. Use **Cur Style** to select a new current style from a list of defined styles, or click **Style Lib** to define a new style, or select one from the library.
7. When finished with the commands on the **Note Types** menu, click **Make Note**.

- If you used one of the no-leader types, you are prompted for the note location, followed by the note text.
- If you selected **File** for the source, a browser opens and you can select the file.
- If you selected **Enter** from the Menu Manager, the prompt line activates and the text symbol palette appears.
- If you use one of the leader note types, two things happen at the same time:
    - The **Attach Type** menu appears on the Menu Manager, and
    - You are prompted to select an attachment point for the "arrow end" of the leader.

      Use the **Attach Type** menu to set up the type of attachment point, the position of the attachment point or points, and the graphic style of the "arrow side" of the leader.

      To determine the type of attachment point, set the Menu Manager to one of **On Entity**, **On Surface**, **Free Point**, **Mid Point**, or **Intersect**. **Mid Point** attaches to the center of a circle or arc, or the midpoint of a line. **Intersect** attaches to the intersection of two lines.

      For multiple leaders, hold down the CTRL key to select multiple points. The selected points are highlighted. Each point gets a leader when the note is placed. You can combine different attachment points and arrow end styles if necessary.

8. Select the location for the note on the sheet. When finished with the selection of one or more attachment points, click **OK** in the **Select** dialog box, click **Done** on the Menu Manager. You are prompted for the position of the note on the sheet.
9. Click the position for the note on the sheet. You are prompted for the note text.

   Enter the note text, either from the keyboard of from a selected file. If you typed a line, press ENTER. The line blanks again so you can enter a second line if necessary. If there is no second line, press ENTER again to place the note.

   The note is placed. You are returned to the Menu Manager to set up and place another note. If you want to adjust the placement of the note you just created, click **Done Return** on the Menu Manager. When you exit the note routines your last note is selected for movement.

Note: The text symbol palette, which appears when you enter text for the note from the keyboard, is usually used to enter symbols in note text. You can also enter symbols directly from the keyboard. To enter keyboard symbol mode, press Ctrl+A/a before you start typing. The keystrokes are displayed in symbol font. Press Ctrl+B/b to return to text mode.

Creating and Saving Drawing Notes

To Create and Edit Custom Text Styles

Use the **Style Lib** dialog box to create new text styles, as well as modify and delete existing ones.

1. Click **Format > Text Style Gallery**. The **Text Style Gallery** dialog box opens.
2. Click **New...**. In the New Text Style dialog box, type a name fin the Style Name box, select an existing style from the **Style Name** list, or click **Select Text...**.
3. Type a name in the **Style Name** box, select an existing style from the **Style Name** list, or click **Select Text...**.
4. After you have changed all attributes, click **OK**.
5. To close the dialog box and save the changes, click **Close** (**Cancel** changes to **Close** after you make a change).

To Specify the Default Font

By default, Pro/ENGINEER creates text in regular ASCII font. A thickened version of the system default font (the filled font) is also automatically available to you as an alternative font. You can specify the ISO font as the default font by setting the drawing setup file option `default_font` to `isofont`, or you can set any of the Pro/ENGINEER-supported fonts as the default for a particular drawing.

In addition to those fonts, Pro/ENGINEER supplies the four auxiliary fonts listed in the following table and 41 TrueType™ fonts.

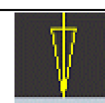
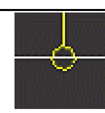
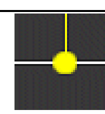
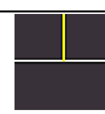
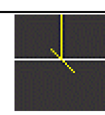
Leroy font set	`leroy`
Calcomp Alphabetical font set	`cal_alf`
Calcomp Greek and Mathematical Symbol set	`cal_grek`
ISO font set	`isofont`

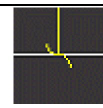
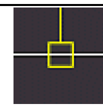
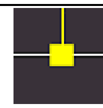
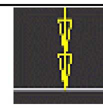
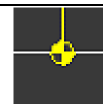
The TrueType fonts are located in the Pro/ENGINEER loadpoint directory. Additional TrueType fonts can be purchased and placed into the loadpoint directory. TrueType fonts can take more time to repaint. Typically, any TrueType Font can be used by Pro/ENGINEER by placing it in the directory specified in the setup option `pro_font_dir`.

By default, any fonts placed in the loadpoint directory appear in the pull-down list in the **Text Style** dialog box. You can use the Apply button on the dialog box to preview any font for selected text.

The ATTACH TYPE Menu In Detailed Drawings

The commands on the **Attach Type** menu determine how the "arrow end" of a leader is attached to an object, and what graphic is used at the leader end.

On Entity	Creates a leader that is attached to a model edge or draft geometry. Selecting the entity near its vertex locates the leader at the vertex.	
On Surface	Locates the leader on model geometry or surfaces. You can select model geometry, threads, and datum surfaces. Once you make an attachment on a surface, if you reorient the view, the attachment remains. If the size of the surface changes, Pro/ENGINEER updates the leader point accordingly.	
Free Point	Locates the leader anywhere in the drawing. You can create a note using this command only in Detailed Drawings.	
Midpoint	Locates the leader at the midpoint of a model edge or draft geometry.	
Intersect	Locates the leader at the intersection of two model edges or two draft entities.	
Arrow Head	Creates a leader with an arrow head.	
Dot	Creates a leader with a dot.	
Filled Dot	Creates a leader with a filled dot.	
No Arrow	Creates a leader with no arrow.	
Slash	Creates a leader with a slash.	

Integral	Creates a leader with an integral.	
Box	Creates a leader with a box.	
Filled Box	Creates a leader with a filled box.	
Double Arrow	Creates a leader with a double arrow head.	
Target	Creates a leader with a target.	

To Relate a Note to Dimension Text

When you relate a note to a dimension, the note moves with the dimension, retaining its position.

Relating a Note at Time of Creation

Follow the instructions in the topic To Add a Drawing Note, and use the **Offset** note type. You are prompted to select a dimension or other detail object from which to offset the note.

Relating an Existing Note

If the note is already placed use the following procedure to relate a note to a dimension:

1. Select the note or notes to relate.
2. Click **Edit > Group > Relate to Object.**
3. Click the dimension to which to relate the note. The note is related to the dimension.

To Enter a Note from a File

You can keep note source text files in the working directory or you can set up a separate directory for them using the setup option "pro_note_dir."

You can enter notes from a file that contain dimensions, parameters, special symbols, and superscripted or subscripted text.

To use a text file as a note source, follow the instructions in the topic *To Add a Drawing Note*, and select **File** rather than **Enter** for the source choice. You are prompted to select a file from a browser to supply the note text.

Creating and Saving Drawing Notes

Using the Pro/ENGINEER **Info** menu, you can write the following information to a file:

- Annotations to views.
- Notes with or without a leader, containing any number of lines.
- Parameters (the system substitutes parameters with their values after copying them to a file).

Notes written to disk using **Info** > **Save Note** are automatically appended with a .txt extension and a version increment extension. This causes notes to appear automatically in the menus when you retrieve them when you are creating notes.

The system saves each selected note in a separate file. When specifying the name of a destination file, type the file name without an extension. The system then automatically appends the default .txt extension and a version increment extension (for example, note.txt.1). You can save notes with another extension (for example, note.ascii.1). The system uses your extension instead of .txt and appends a version increment extension (note.ascii.1). If you type the same file name as an existing file name, the system updates the version increment extension automatically so as not to overwrite an existing file (note.txt.2).

Note: During the process, Pro/ENGINEER loses boxes and underlining information, and does not save special symbols. To save them, bring the note into a text editor and save it from within the editor.

To Write Notes to a File

1. From the Pro/ENGINEER menu bar, choose **Info** > **Save Note**.
2. Select a note to save.
3. Type the name of a destination file in the prompt line.

To Add a Balloon Note

Balloon notes consist of text enclosed in a circle. You can add them freely anywhere on the drawing, or attach them to any number of edges. When you enter multiple lines of text for the balloon note, Pro/ENGINEER encloses each line within a balloon, stacking them horizontally or vertically next to each other, depending upon whether you choose **Horizontal** or **Vertical** on the **NOTE TYPES** menu. The size of these related balloons is equal to the largest one necessary to enclose the longest text.

1. Click **Insert** > **Balloon**.
2. Click the required commands on the **NOTE TYPES** menu and click **Make Note**. The **ATTACH TYPE** menu appears.
3. Select the leader type, if necessary.
4. Select an entity, surface, or a free point as the attachment location for the balloon and click **OK** in the **Select** dialog box. The **GET POINT** menu appears.

5. Specify the point where you want to attach the balloon note.
6. Type the note text. Press ENTER to start a new line, or press ENTER twice to complete the note. The balloon note is placed on the drawing.
7. Click **Done/Return** to close the **NOTE TYPES** menu manager.

Note: To restrict the size of a balloon, use the `max_balloon_radius` or `min_balloon_radius` drawing setup file options.

Modifying Note Text

Text Strings

When you are editing text, dimension values appear as follows:

- @D (if displayed as a numerical value).
- @S (if displayed as a symbolic value).
- @O (if the dimension does not have a value, @O locates the text origin).
- Special characters appear between the control characters ^A and ^B.

When you edit a note, the system preserves all of the attributes (font, height, width, or slant angle) applied to a portion of the text. However, the note appears much different from how it does on the drawing. The system breaks up a text string into portions wherever there is a new line of text or a parameter (such as dimensions), and encloses each portion of the text in braces ({}), giving it an integer label. Labels identify the initial order of the text, and any attributes for that portion. When editing text, or adding more lines, you can copy the attributes of a portion of text by using the same integer label.

Note: If you delete the special control characters when editing text, the text string changes. If you remove ^A, all special characters become their ASCII equivalent; if you remove ^B, all text that follows becomes a special character.

You can edit view-related notes, such as section, detail, and scale notes; however, be sure not to delete or otherwise alter symbols that represent the name or scale of the view. If you edit these symbols, the system ignores the change, and displays the original note:

- `&view_name`—name of the view
- `&view_scale`—name of a general scaled view
- `&det_scale`—scale of a detailed view

If you set the `view_note` option in the drawing setup file to `std_din`, you can create a view-related note with the words `SECTION`, `DETAIL`, and `SEE_DETAIL` omitted. This only affects the creation of view-related notes. If you switch from `std_ansi` to `std_din`, the system does not update view notes.

To Modify Text Style

1. Select the note or text you want to modify.
2. Click **Format** > **Text Style**. The **Text Style** dialog box opens.

   Note: The **Text Style** options are also available in the **Note Properties** dialog box and the **Dimension Properties** dialog box.
3. Use the available options in the **Text Style** dialog box to edit the text properties.
4. Click **OK** to save changes and close the **Text Style** dialog box.

Note:

If the value of the note height is set to less than 0.0014, which is the minimum allowable text height, the value will automatically switch to the default height specified in the `drawing_text_height` drawing option.

If you change the style assigned to note text, you can change other text strings to the same style using . Select the text or note you want to modify and click the icon. The last selected style is applied to the selection.

To Modify a Style Definition

To modify a style definition,

1. Click **Format > Text Style Gallery**. The **Text Style Gallery** dialog box opens.
2. Select the style to modify and click **Modify**. The Modify Text Style dialog box opens.
3. Use the dialog box to make changes to the style definition. Click **OK** when finished.

To Modify Selected Text Color

1. Click **Format > Text Style**.
2. Select text; then click the middle mouse button to complete. The Text Style dialog box opens.
3. In the **Text Style** dialog box, select the **Color** box.
4. In the **Color** dialog box, select a system color, then click **OK**. The system displays the selected text in the specified color and closes the Color dialog box.
5. To reset the text to the old style, click **Reset**. To reset selected text to its intrinsic color, click **Intrinsic** in the Color dialog box.

To Wrap Note Text

When a note is created the text string is entered as one line. Use this procedure to wrap the text to a desired width.

1. Select the note. The text is highlighted and "handles" appear on either side of the string.
2. Use the cursor to drag the right side text area handle to the desired width. The text wraps according to the width of the text area.

If you add text to a note, the new text is not wrapped. To wrap it, you can either

- Re-drag the text box to the desired width, or
- Select the note and from the right mouse button pop up menu, click Wrap Text. The note is wrapped to the longest line.

About the Text Style Dialog Box

You can change the font, width, and color of an existing text style using the **Text Style** dialog box. However, Pro/ENGINEER does not automatically update any text to which you previously assigned that text style, that is, before you modified it. The changes that you make using this dialog box only affect text to which you assign a text style after modifying the text style.

You can access the **Text Style** dialog box in the following ways:

- By clicking **Format** > **Text Style**.
- By clicking the **Text Style** tab in the **Note Properties** or the **Dimension Properties** dialog box. The **Note Properties** or the **Dimension Properties** dialog boxes open when you select a note or a dimension text, right-click, and click **Properties** on the shortcut menu.
- When you modify the text style of the additional text associated with a geometric tolerance, by selecting the additional text, right-clicking, and clicking **Text Style** on the shortcut menu.

Using the **Text Style** dialog box, you can:

- Change the font by selecting a font name in the **Font** list.

  Note: To make Open Type and True Type fonts available on UNIX, ensure that the filename extension for the font is in lowercase.

- Change the text height by clearing the **Default** check box and typing a value in the **Height** box.
- Change the text width by clearing the **Default** check box and typing a value in the **Width Factor** box.
- Change the text thickness by clearing the **Default** check box and typing a value in the **Thickness** box.
- Change the slant angle of the text by typing a value in the **Slant Angle** box.
- Underline the text by clicking the **Underline** check box.
- Enable font kerning for the text string by clicking the **Kerning** check box. This reduces the space between certain pairs of characters, improving the appearance

of the text string. Alternatively, control the font kerning by setting the `default_font_kerning_in_drawing` configuration option.

- Change the line spacing by clearing the **Default** check box and typing a value in the **Line Spacing** box.
- Specify the horizontal positioning of the text by selecting **Left**, **Center**, **Right**, or **Default** in the **Horizontal** list.
- Specify the vertical positioning of table cell text in individual table cells by selecting **Top**, **Middle**, **Bottom** or **Bottom of Origin Line** in the **Vertical** list.

  Note: The **Vertical** option for text positioning is available for selection only for the **On Item** notes.
- Rotate the text at an angle by typing a value in the **Angle** box. The default rotation point of the text is determined by the horizontal and vertical positioning of the text. For example, if the horizontal positioning is **Right** and the vertical positioning is **Bottom**, the text is rotated about the point where the horizontal and vertical positions intersect by an angle that you specify in **Text Style** dialog box.
- Flip the text so that it reads backwards by clicking the **Mirror** check box.
- Use **Break Crosshatching** to set an offset around a note if it is within a crosshatched area.
- Change the color of the note text by clicking **Color** and selecting the required color from the **Color** palette.

Note: You cannot change the font, height, thickness, and width of the additional text associated with a geometric tolerance.

To Enclose Notes in Text Boxes

To enclose words or characters in a box, type "`@[`" before and "`@]`" after the text.

For example:

`@[Notes in drawings@]` puts the box around the entire text string.

`@[Notes@]` in drawings puts the box around the word "Notes."

To Create Superscripted and Subscripted Text

To create superscripted text, type `[@+text@#]`and to create subscripted text, type `[@-text@#]`,where `text` is the note text that is superscripted or subscripted.

You can superscript or subscript plain text and special symbols; however, you cannot super- or subscript dimensions, instance numbers, other parameter values, or geometric tolerances. You can create superscripted and subscripted text as a separate note or include it in a text line with regular text on either side of it, and then modify it independently of the rest of the text line in which it appears. The system positions the text in reference to the closest line of regular text, whether that text is part of another note, or part of the note currently being created.

To Specify an Existing Text Style as the Current Style

As you create a note, you can apply an existing text style by choosing **Cur Style** from the NOTE TYPES menu manager. When you change the current style from the default to another text style, it only affects new notes, balloon text, symbols, and table text.

If you choose **Default** from the SEL STYLE menu, the system uses the style that is defined by the drawing setup file options `drawing_text_height`, `text_width_factor`, and `text_thickness`.

To Change Text Strings

1. Select the text string to modify. The text is highlighted.
2. Right-click and click **Properties** from the shortcut menu. The Note Properties dialog box opens.
3. Click the **Text** tab.
4. Edit the text string as necessary. When you have completed the changes, either save the note or click **OK** to return to the drawing.

To Place Draft and Reference Dimensions in Notes and Tables

You can place draft (add and dd) dimensions and reference (rd) dimensions parametrically in drawing notes and tables using the format &add or &dd. Draft (driven) dimensions and reference dimensions created in the drawing are updated when the model is regenerated.

Adding Special Text Characters to Notes

When typing text strings into notes, you can use special text characters, special symbols, and drawing symbols.

You can add special symbols to text strings in notes by typing them from the keyboard, the palette window, or a text file. For a table listing all of the special symbols available with Pro/ENGINEER, as well as their definitions and ASCII character representations, see the *Pro/ENGINEER Installation and Administration Guide*.

Whenever you want to display the characters & or @ in the text of a note, you must type them twice. For example, type [`See Views 1 && 2 @@ Sheet 3`] for the note to read `See Views 1 & 2 @ Sheet 3`.

Adding Drawing Symbols to Notes

You can include a drawing symbol in a note if its instance is present in the drawing. To include a symbol, use the following format:

```
&sym(<symbolname>)
```

Type the symbol filename without an extension.

> **Note:** You cannot call out drawing symbols in dimensions that are stored with the model (those of type d or ad).

When you include an instance of a generic symbol, after you type the symbol name, select groups that compose the instance (this is similar to creating instances of a generic symbol).

For example, if a drawing contains an instance of the symbol bevel, to include the symbol in a note, type [&sym(bevel)].

When you edit a note, the system represents the symbol in the following format:

```
n:&sym(sym_path)
```

where n is the number of the text element, and sym_path is the name or pathname of the symbol.

Note: Pro/ENGINEER displays a drawing symbol in the text note to which it is added, but it does not display it in the dimension text line. Dimension text resides in Part mode; therefore, it may not acquire drawing symbols that reside in Drawing mode.

To Include the Symbol:

1. Click **Insert** > **Note**.
2. Type the note in the format described earlier.
3. Specify the symbol height.
4. For an instance of a generic symbol, select groups to compose the instance. To complete the instance description, type variable text, if necessary.
5. To finish the note, press the ENTER key twice.

Including Parameter Information in Notes

When you set the drawing set up file option yes_no_parameter_display to yes_no, parameters can have a yes or no value in drawing notes. When you set it to true_false (the default value), they can have a true or false value.

To specify parameter information in a note, use the following format:

- Dimensions—&d# or &ad#, where # is the dimension ID. Examples: &d12. &ad5.
- Reference dimensions—&rd#, where # is the dimension ID. Example: &rd2.
- Instance number—&p#, where # is the parameter ID. Example: &p8
- User-defined parameters—&xxxx where xxxx is a symbol defined in a relation.
- Datum names—&dtm_name, where name is the name of a datum plane.
- Drawing labels—You can add the following drawing labels to a drawing:
    - &todays_date—Adds the date as of the note's creation in the form dd-mm-yy (for example, 2-Jan-92). If you include this symbol in a format table, the system evaluates it when it copies the format into the drawing.
    - &model_name—Adds the model used for the drawing.

  - &dwg_name—Adds the name of the drawing.
  - &scale—Adds the scale of the drawing.
  - &type—Adds the drawing model type.
  - &format—Adds the format size.
  - &linear_tol_0_0 through &linear_tol_0_000000—Adds the linear tolerance values for one to six decimal places.
  - &angular_tol_0_0 through &angular_tol_0_000000—Adds the angular tolerance values for one to six decimal places.
  - &current_sheet—Adds the sheet number for the sheet on which the note is located.
  - &total_sheets—Adds the total number of sheets for the drawing.
- Drawing parameters—&parameter:d, where parameter is the parameter name.

Erasing a Note

Erasing a note does not delete it from the drawing. The system also hides any dimensions or parameters that are included in the erased note, and does not return them to their original locations elsewhere in the drawing.

1. Select the note to erase.
2. Click Erase from the right mouse button shortcut menu. The note is erased.
3. To show erased notes, click **View** > **Show and Erase.** Use the **Show** tab, and the notes icon.

Modifying Note Parameters

About Note Parameters

In addition to text and special symbols, notes can include model dimensions, pattern instance parameters, symbols, drawing labels, and drawing parameters. The drawing setup file option yes_no_parameter_display controls the display of yes/no parameters in drawing notes and tables.

When specifying parameter information in a note, keep in mind the following restrictions:

- When you include dimensions in notes, the system removes them from their originally-assigned views.
- You cannot add draft dimensions (associative—add # or nonassociative -dd#) to a note.
- A datum name in the note is read-only, so you cannot modify it; unlike dimensions, a datum name does not disappear from the model view if it is

included in a note. The system encloses its name in a rectangle, as if it were a set datum.

- You cannot use drawing labels in drawing relations; you can only use them in drawing notes and tables.
- When the note appears, the system replaces dimensions and parameters by the corresponding numerical values. If you set the configuration file option `switch_dims_for_notes` to `yes`, dimensions appear as their symbolic values during note creation. If you set it to `no`, they remain as numerical values.
- Before referencing a model in a parametric note, make sure that you add this model to the drawing. Otherwise, the system does not evaluate the model parameter referenced in the note.
- You can include a gtol symbol in a note by entering it as a parameter; however, you cannot include global parameters (that is, parameters created in a layout) and draft dimensions in notes.

To Control the Format of the Date

The configuration file option `todays_date_note_format` controls the initial format of the date displayed in a drawing. The format for the setting is a string consisting of three portions: the year, the month, and the date. You can enter the portions in any order. The default value is `%dd-%Mmm-%yy`.

- Year
  - %yy, for 97
  - %yyyy, for 1997
- Month (if the month contains two digits (for example, 10),% mm, % m, or % m all produce the same result)
  - %Mmm, for Jan
  - %MMM, for JAN
  - %Month, for January
  - %MONTH, for JANUARY
  - %mm, for 01
  - %m, for 1
  - % m, for <space>1
- Date (if 2 digits are needed to represent the date, all three are the same. Therefore, "%dd %mm %yy" produces "01 01 97," and "%MMM %d %yyyy" produces "JAN 1 1997")
  - %dd, for 01
  - %d, for 1

  - % d, for <space>1.

The following formats are also valid:

- %dd-%Mmm-%yy (= 01-Jan-97)
- %mm/%dd/%yy (= 01/01/97)
- %Mmm %dd,%yyyy (= Jan 01, 1997)

To Include a Feature Parameter in a Note

To include a feature parameter in a note, use this format:

```
&<param_name>:FID_<feat_ID>
```

or

```
&<param_name>:FID_<FEAT_NAME>
```

To Include a Model Note in a Drawing Note

To include a model note in a drawing note, use the format &<param_name>.

To Display Pro/PDM Data

You can display the name of a model's product database on a drawing sheet, as well as the model's revision in that product database and its release level, by entering the following in a note:

- To display the model's product database of origin, enter the parameter &PDMDB.
- To display the model's revision, enter the parameter &PDMREV.
- To display the drawing's revision number, enter the parameter &PDMREV:D.
- To display the model's release level, enter the parameter &PDMRL.

To Reference Parameters Assigned to Objects

In model and drawing notes, to reference parameters that you assigned to objects in Part or Assembly mode, use this format:

- For edges:

```
&<param_name>:EID_<edge_name>
```

- For surfaces:

```
&<param_name>:SID_<surface_name>
```

- For other objects:

```
&<param_name>
```

To Reference a Mass Properties Symbol in a Note

You can create a parametric note that references a mass properties symbol. After the geometry changes, you can update the note to reflect the latest value of the mass properties parameter.

1. Set a user-defined parameter. For example: [`volume`].
2. Add a relation, assigning this parameter to a mass properties symbol that you want to reference in a note. For example: `volume = mp_volume("")`.
3. Add a note containing the user-defined parameter. For example: [`&volume`].

To Update a Parametric Note

1. Regenerate the model.
2. Do one of the following:
   - Manually recompute mass properties by choosing **Analysis** > **Model Analysis...**.
   - Include a MASSPROP statement in the program of the model using Pro/PROGRAM.

Controlling the Number of Decimal Places in Parameters

When you append certain parameter symbols with the characters "[.#]", the system displays those parameters with the number of decimal places specified by #, which is an integer. The system rounds the number, but uses the same value. This applies to the following parameters:

- User-defined model parameters or those defined in model relations
- Drawing labels, such as `&scale` (drawing scale) and `&det_scale` (detail view scale)

When adding the note that contains the parameter, append the parameter symbol with "[.#]", where # is the number of decimal spaces to appear. For example, if a detailed view scale is 1.125, and you want to display only two decimal places, change the drawing label to `&det_scale[.2]` (no spaces). This displays the scale note as 1.13.

To edit the scale note, select it, right-click and click **Properties** on the shortcut menu.

Restrictions with Dimensions and Other Model Parameters

You cannot use this functionality on dimensions or other model parameters, such as pattern parameters. However, if you make a user-defined parameter equal to the model parameter, and then add the user-defined parameter to the note, you can control the display of the model parameter decimal places.

To Show the Scale of an Individual View

The drawing parameter `scale_of_view_x` evaluates the drawing scale of whichever view has name X.

For example, to call out the scale of a view named `DETAILED_BAR`, type `&scale_of_view_detailed_bar`.

Example: Parameters in Notes

The following example illustrates the parameter's usage in notes:

1. In Part mode, assign the name "A" to a surface.
2. Create a parameter with a name "RAD," of type Number.
3. Assign a value ".03" to this parameter.
4. In Drawing mode, create a note by typing [`Break sharp edge with R = &RAD:SID_A`]. This note appears: "Break sharp edge with R =.03".

If a drawing note has a single attachment (defined using **On Item** or one leader line), you can show the parameters of the entity to which it is attached by typing the following string into the note: `&param_name:att`. The system then interprets the `param_name` in a series of contexts until it is able to evaluate it successfully. First, it searches the immediate entity to which the note is attached, such as an edge. Next, it searches the feature that owns the entity. If it still cannot evaluate the parameter, it then searches the model that owns the feature. Finally, if applicable, it searches the component that refers to the model. If you want to specify the exact context in which to interpret the parameter, specify a full "`att_`" postfix instead.

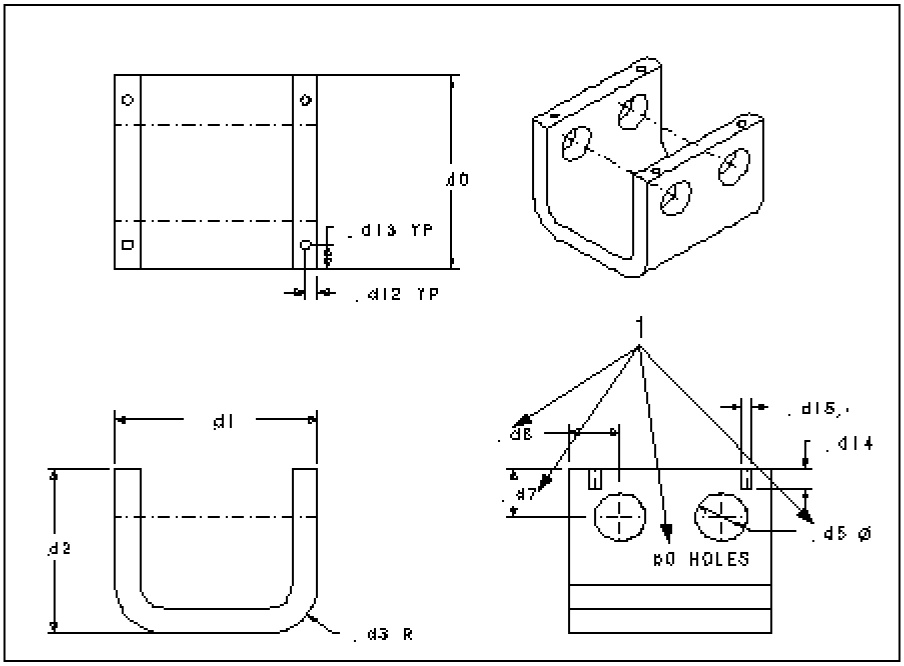
After you create the note, if you choose **Switch Dimensions** from the Pro/ENGINEER **Info** menu, the system appends the "`att_`" postfix to specify the exact context in which it interprets the parameter.

Param_name:att_edge	Edge
Param_name:att_feat	Feature
Param_name:att_mdl	Model
Param_name:att_cmp	Component

For example, if a note is attached to an edge by a single leader line (or defined using **On Item**), you can set the note to show a relational parameter of that edge by typing [`&Length:att`]. The note then appears as follows: `&Length:att_edge`.

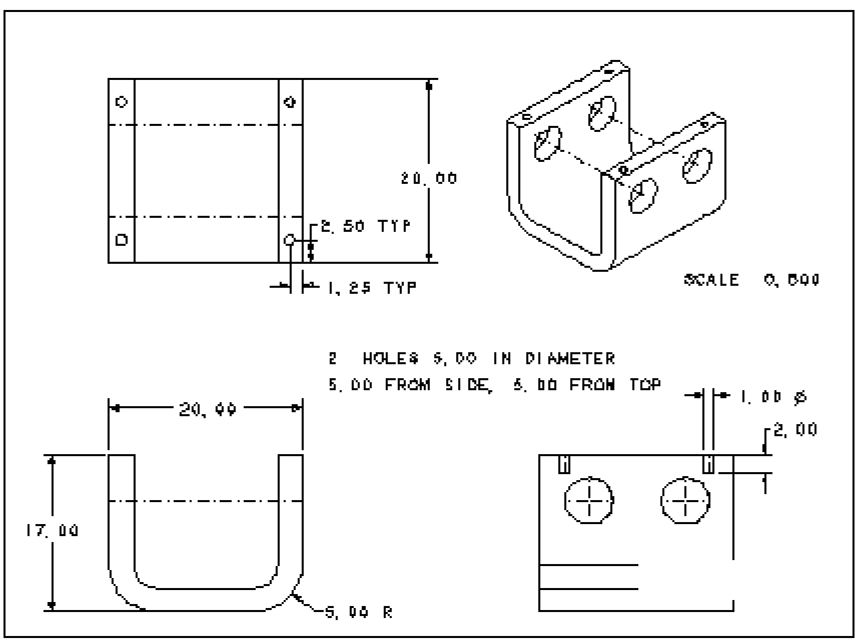
Note: You must create the drawing note after you create the parameter; otherwise, the system does not evaluate it correctly.

Example: Parameter Symbols for Notes



1. Parameter symbols.

Example: A Note with Parameters



Type the following:
&p0 HOLES &d5 IN DIAMETER <ENTER>
&d6 FROM SIDE, &d7 FROM TOP <ENTER><ENTER>

Example: Including a Model Note in a Drawing Note

1. In a model, create this note: all surfaces must be painted with ID 1.
2. Add a parameter part_note with the value of 1.
3. In Drawing mode, type this note:

   [Applies to all parts:
   &part_note]

   The following note appears:

   Applies to all parts:
   all surfaces must be painted

   Note: You cannot insert the part_note text side-by-side with any other note text. When you call out a note parameter in another note, the called out parameter is displayed on a new line.

Example: Controlling the Decimal Places in Parameters

For example, for a model parameter "d1" with the value 12.37580:

1. Type the relation as [length = d1].
2. Add the drawing note.
3. When specifying the parameter, type [&length[.3]]. The parameter appears as 12.376, but the value the system uses in calculations is still 12.37580.

You can also use the **Num Digits** command to modify the number of decimals in a scale.

Location Callouts

About Location Callouts

You can define a zone or location grid to drawing formats by creating location callouts. You define the location grid by using the existing lines of a format or by using equally spaced increments. Use this location grid to locate drawing views using parametric notes to indicate both the sheet and drawing location of a view.

Parametric location callouts allow for intelligent notes to be added to drawings to indicate the location of a detail or section view, saving time and preventing out-of-date notes.

Location callouts are created using a standard syntax:

- View location—&pos_loc:view:<view name>
- Sheet of view—&sheet:view:<view name>
- Note location in parent view—&pos_loc:parent_note:<view name>
- Sheet location of note—&sheet:parent_note:<view name>

For example, if Detail A is located in A3 on the location callout grid and on sheet 2, type the note:

```
See Detail A at &pos_loc:view:detail_a on sheet &sheet:view:detail_a
```

The note text shows as:

See Detail A at A3 on sheet 2.

The syntax for the location callouts is customizable, which allows for additional text to be added based on individual company standards. The detail setup option `pos_loc_format` is provided to specify the syntax of the `pos_loc` and sheet parameters in the callout. The default of `pos_loc_format` is %s%x%y, and appears as 2D4.

%r added to `pos_loc_format` will repeat the syntax string for the same item appearing several times in different locations. This applies to items such as symbols and connectors, but not to views because views have unique names. The syntax will be %s%x%y, %r where %r means to repeat the substring from the first special character to the last.

`&pos_loc`

Behavior:

The location note updates upon regeneration if the view is moved on the drawing. If the view is erased or moved outside of the grid, the value of the location callout changes to ***. The callout updates when the view is resumed or the view is moved back onto the grid. If the view is deleted, then the location callout is no longer parametric and the value changes to ***.

`&sheet`

Behavior:

If you switch the view to a different sheet, the location callout automatically updates to the new sheet number. If the view is erased, the value of the location callout changes to ***. The callout updates when the view is resumed. If the view is moved outside of the grid, the value remains as the correct sheet number. If the view is deleted, then the location callout is no longer parametric and the value changes to ***.

You specify which corner of the location grid you want as the origin. The grid origin is shown as a yellow `X`. The default grid origin is the lower right corner of the location grid.

To Define the Location Callout Grid

Define a location grid on the format file (.frm) your drawing is referencing.

1. In a Format file click **File > Page Setup**. The **Location Grid** menu opens on the menu manager.
2. Click **Define**.
3. Click one of the following to define:

- **Grid Outline**—Specify the grid outline by using the rubberband box or by papersize (equal to the size of the page).
- **Grid Origin**—Pick a point to indicate the grid origin. The lower right-hand corner is the default origin.
- **Columns**—Specify the amount and location of grid columns by number or pick points, and then specify the grid labels by letter, number, or custom.
- **Rows**— Specify the amount and location of grid rows by number or pick points, and then specify the grid labels by letter, number, or custom.

4. Click **Done/Return**.

To Show the Location Callout in a New Drawing

1. In Drawing mode, click **Insert > Note**.
2. Select the location for the location callout.
3. Type a note, and then type the following:
   - To show the view location on the grid: &pos_loc:view:<view name>
   - To show just the sheet location: &sheet:view:<view name>
   - In a detail view, to show the note location in the parent view: &pos_loc:parent_note:<view name>
   - In a detail view, to show the sheet location of the note: &sheet:parent_note:<view name>
4. Click **Done/Return**. The location callout is created.

Example: A Location Callout

To show the location of a view named Main_view, type the following in the **Enter NOTE** text box:

```
Main_view is located in &pos_loc:view:main_view
```

The note shows as:

```
Main_view is located in 3B
```

According to the location callout grid you specified, Main_view is located in section 3B.

Annotation Features

About Associating Annotations of a 3D Model in Drawings

When you create a 3D model with 3D annotations and show the annotations in a drawing, you can define whether associativity of the attachment and position references of the 3D model in the drawing is to be maintained with their relative text placement position and attachment references in the 3D model.

Note: Cosmetics of annotations, such as, text properties, jogs, breaks, skew, witness line clipping, and flip arrow states are not associative.

About Associating the Position of a 3D Annotation Shown in a Drawing

Associativity of position is maintained only for annotations that meet the following criteria:

- Annotations placed with reference to their model coordinates. This includes flat to screen annotation planes that use model units.
- Annotations attached to geometry.
- Annotations not placed within another annotation.

Note: Flat to screen annotations created prior to Pro/ENGINEER Wildfire 2.0 and model notes placed flat to screen using the **Note Properties** dialog box do not support associativity of position.

If you maintain the associativity of the position and change the placement position of the annotation in the 3D model, shown annotations in all the associated drawings are updated to the new placement position.

You can associate the text placement position of an annotation with its 3D model in a drawing only if the text placement position is independent of the attachment point. For example, you can associate the text placement position of a leader type annotation with its 3D model because the leader type annotation is independent of the attachment point. However, you cannot associate the text placement position of a note placed on an entity with its 3D model because this type of annotation is not independent of its attachment point.

You can also disassociate the position references of an annotation with its 3D model in a drawing. The shown annotation in a drawing is automatically disassociated with its 3D model when you:

- Move the annotation in a drawing to a different location.
- Clean up the placement of drawing dimensions.

Remember the following points while modifying annotations:

- You can insert jogs on leaders of annotations such as notes, geometric tolerances, symbols, and surface finishes without changing the associativity of position. However, if you insert a jog on a dimension witness line, the associativity of position of the dimension witness line with the 3D model is automatically lost.
- You can insert a break on a dimension witness line without changing the associativity of position of the dimension witness line with the 3D model.
- Jogs, breaks, and skews in 3D dimensions are not associative and are not displayed in the drawing when the dimension is shown.
- If you skew a dimension, the dimension automatically loses its associativity of position with the 3D model.

About Associating the Attachment of a 3D Annotation Shown in a Drawing

Associativity of attachment is maintained only for annotations that meet the following criteria:

- Annotations must be attached to geometry.
- Annotations must use an annotation plane that is not flat to screen.

Some model annotations, such as symbols, surface finishes, set datum tags, and dimensions that are shown in a drawing have bidirectional associativity of attachment references. That is, when you change the attachment references of 3D annotations in a model, the references of the shown annotations are updated in the drawing. Similarly, when you change the attachment references of the shown annotations in a drawing, their attachment references are updated in the model as well. For such annotations, the **References driven by model** check box in the **Dependencies** tab of the appropriate dialog boxes is always selected.

Note: You cannot modify the attachment references of certain shown annotations in a drawing.

Some model annotations, such as notes and geometric tolerances shown in a drawing can have attachment references that can be made independent of their 3D attachments. For such annotations, you can decide whether or not to retain the reference associativity.

When you add, delete, or change the attachment references of a shown annotation that supports independent attachment references in a drawing, a confirmation dialog box appears indicating that the attachment reference associativity of the shown annotation with its associated 3D model will be broken.

To Modify the Associativity of Position and Attachment References

1. Select a shown annotation in a drawing.
2. Right-click and click **Properties** on the shortcut menu. Depending on the type of the selected shown annotation, an appropriate dialog box opens.
3. Click the **Dependencies** tab.
4. Under **Placement Position**, clear the **Driven by Model** check box to break the associativity of position.
5. Under **Attachment**, clear the **References driven by model** and **Attach point(s) driven by model** check boxes to break the attachment associativity.
6. Click **OK** to save the changes and close the dialog box.

Note:

- The associativity of position is automatically disabled when you dynamically move a shown model annotation in a drawing.

- The **Attach point(s) driven by model** check box is automatically cleared when you dynamically move the attachment point of the shown annotation with respect to its current reference.
- The **References driven by model** check box is cleared when you click **Change Ref** on the **MOD OPTIONS** menu and select a different attachment reference. A confirmation dialog box appears indicating that the attachment references of the shown annotation with its associated 3D model will be broken. Click **Yes** to modify the attachment references of the shown annotation in the drawing and break its associativity with the 3D model.

To Restore 3D Dependencies in a Drawing

1. Select one or more annotations in a drawing that you have disassociated from the 3D model.
2. Right-click and click **Restore 3D Dependencies** on the shortcut menu. Alternatively, right-click and click **Properties** on the shortcut menu and select the options in the **Dependencies** tabbed page to restore 3D dependencies.

Cleaning Up Dimension and Detail Display

About Cleaning Up Dimensions

You can cleanup the placement of drawing dimensions to meet industry standards and enable easier reading of your model detailing. There are numerous ways to adjust the locations of dimensions, including:

- Manually moving the dimension to the desired location on the drawing sheet.
- Aligning selected dimensions with a specific dimension.
- Automatically arranging the display of selected dimensions automatically by setting controls for dimension placement and cosmetic attributes, like flip arrow direction.

You can also adjust the display of the dimensions by:

- Moving the dimensions to a different views
- Toggling the Leader-to-Text Style
- Modifying the witness lines.

Note:

Cleaning up the dimension display can also be accomplished by customizing the dimension display, including controlling what values appear and how the text is formatted.

To Erase Dimensions and Detailing Items

You can remove both driving and driven dimensions from the display by erasing them. This does not delete them from the model.

If you are only deleting a small number of dimensions, you may want to right-click on the dimension and select **Erase**. Otherwise, continue with the following:

1. Click **View > Show and Erase**. The Show/Erase dialog box opens.
2. Click **Erase**.
3. Under **Type**, indicate which type of dimension or detailing item you wish to erase. You can select multiple items.
4. Under **Erase By**, where you want to erase the dimensions:

- **Selected Items**—Erase individual dimensions from any location on the drawing.
- **Feature**—Erase dimensions for a selected feature.
- **Feature and View**—Erase dimensions for a selected feature within a view.
- **Part**—Erase dimensions for a part at the selected assembly location.
- **Part & View**—Erase dimensions for a selected part within a view.
- **View**—Erase all dimensions in the selected view.
- **Erase All**—Erase all dimensions in the drawing.

5. Select the appropriate feature, part, or view to erase the dimensions from. You may need to repaint the drawing for the dimensions to display. When you are finished with the preview, click **OK** in the Select dialog box. The dimensions are shown.
6. Click **Close** to close the Show and Erase dialog box.

To Move Dimensions

You can move dimensions within a view using the following procedure:

1. Select the dimension you want to move. The cursor changes to a four-pointed arrow.
2. Drag the dimension to the desired location.

Note:

- You can use the `CTRL` key to select multiple dimensions. If you move one of the selected dimensions, they all move with it.
- You can move created (driven) dimensions by editing the attachment point of the dimension. Select the dimension and click **Edit** > **Attachment**. Select a new attachment point within the same view and middle click to place the dimension. Shown model dimensions can not be moved with this command, as it would require changing the reference within the model.
- You move dimensions according to their `X` and `Y` coordinates using **Edit** > **Move Special**.

To Move an Item Between Views

You can move detailing items attached to the model with leader lines (or attached directly to an edge) from one view to another view of the same model. You can switch the view that dimensions appear in using the following procedure. If Pro/ENGINEER cannot display a dimension in the selected view, it displays a warning message and does not move it.

1. Select a item to move.
2. Click **Edit > Move Item to View**.
3. Select the view to which you want to move the item. The item is attached to the new view, and activated to move for adjustment.

Note:

- When you select a patterned feature dimension, all dimensions of the pattern move.
- When you switch the view of an ordinate baseline, all ordinate dimensions that reference it also switch views.

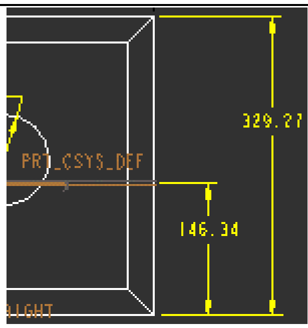
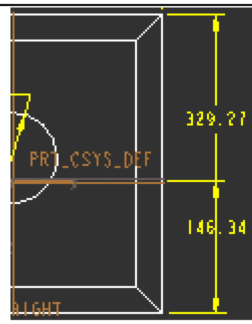
To Align Dimensions

You can clean up your drawing display by aligning linear, radial and angular dimensions. The selected dimensions align to the first dimension selected, provided that they share a parallel witness line. Any dimension that cannot be aligned with the selected dimensions will not move.

1. Select the dimension to which you wish to align other dimensions. The dimension is highlighted.
2. Hold down the CTRL key and select the remaining dimensions to align. You can select the additional dimensions individually or use region selection. You can also select non-dimensional objects, however, the alignment only applies to the selected dimensions. The selected dimensions are highlighted. .
3. Right-click and click **Align Dimensions** on the shortcut menu or click **Edit** > **Align Dimensions** on the main menu. The dimensions align to the first selected dimension.

Note:

- Each dimension can move independently to a new location. Aligned dimensions do not maintain their alignment if one of the dimensions is moved.
- You can also align dimensions using snap lines. Moving the snap line moves all the entities aligned to it.
- The order in which you select dimensions is important. The first dimension you select establishes the target leader space. The leaders of subsequent dimensions that you select will attempt to align themselves to the previous leaders.

Unaligned Linear Dimensions	Aligned Linear Dimensions
	

To Automatically Cleanup Dimensions

Use this procedure to simultaneously:

- Clean dimensions by view, or by individual dimensions.
- Center dimensions between witness lines (including the entire text box with gtols, diameter symbols, tolerances, and so on).
- Create breaks in witness lines where they intersect other witness lines or draft entities.
- Place all dimensions on one side of a model edge, datum plane, view edge, axis, or snap line.
- Flip arrows.
- Offset dimensions from edges or view boundaries.

1. Click **Edit > Cleanup > Dimensions**. The Clean Dimensions dialog box opens but is not active.
2. Select individual or multiple dimensions or an entire view; then click **OK**. The Clean Dimensions dialog box activates.
3. Click the **Placement** tab to modify the placement of dimensions and detailing items:
    - Type an initial uniform offset value in the **Offset** box. To place the selected dimensions, type an incremental offset value in the **Increment** box.
    - In the **Placement** page, do one of the following:

      Select **View Outline** to offset the dimensions with respect to their view outlines.

      Select **Baseline** to reposition only dimensions in the same view whose leaders are parallel to the selected baseline.

- Select one of the entities in the drop down list to use as a cleaning baseline. An arrow appears to indicate the offset direction. To change the direction click **Flip Arrow**.
- Check **Create Snap Lines** to add dashed snap lines to the dimensions.
- Check **Break Witness Lines** to break witness lines where they intersect other draft entities.
- Click **Apply**. The system applies cosmetic cleanups to all dimensions. Click **Undo** to return them to their pre-cleaned state and try again without reselecting dimensions.

4. Click the **Cosmetic** tab.
   - By default, the system selects all of the check boxes. Clear any or all of them, as necessary; then click **Apply**:

     Flip Arrows flips the arrows inside the witness lines if they fit (without overlapping the text); if they do not fit or they would be overlapping the text, they flip outside the witness lines.

     Center Text centers the text of each dimension between its witness lines. If it does not fit, the system moves the text outside the witness lines in the specified direction:

     Horizontal moves the text to the left or right.

     Vertical moves text of vertical dimensions up or down.
   - **Create snaplines** creates snap lines under all moved dimensions (if the destination does not already have a snap line in the vicinity). They appear only under dimensions that are parallel to baselines (if you have selected a baseline) or parallel to a view border (if the view outline has been selected). Once you clear this check box, it remains cleared for the remainder of your Pro/ENGINEER session.

To Toggle Leader-to-Text Style

You can change the leader to text or symbol attachment for notes or symbols to the options shown below.

1. Select a note or symbol with a leader.
2. Click **Format > Toggle Leader Type**.

To Reroute Dimensions with Lost References

Lost dimension references may occur when 3D parts or features are suppressed, or during feature redefinition. Lost references in a drawing are highlighted in a unique color.

1. Click the lost dimension to reroute and then use the right-mouse button to use the shortcut menu.
2. Click **Edit Attachment** to reroute the dimension to new references.

To Delete Dimensions

When working with added dimensions (dimensions created with the **Insert** command.) you can either erase or delete the dimensions. Deleting the dimensions permanently removes them from the drawing.

1. Select the added dimension(s) you wish to permanently delete from the drawing.
2. Right-click and select **Delete** on the shortcut menu, or click **Edit** > **Delete**. The dimension is deleted.

Note: If you unintentionally deleted a dimension, you can undo all the deleted dimensions within that operation by clicking **Edit** > **Undo Delete**.

Formatting Dimension Display

About Formatting Dimension Display

You can customize the display of dimensions within your drawings to meet your detailing requirements. You can format these dimensions in one of the following ways:

- Individually using the **Dimension Properties** dialog box
- Globally using either drawing setup file and configuration options or the **Format** menu

Individually Formatting Dimensions

The **Dimension Properties** dialog box (**Edit** > **Properties**) enables you to format the display of individual dimensions. The dimension must already exist in your drawing if you want to access and customize its properties. The property settings only apply to the selected instances. From within the dialog box, you can:

- Modify the dimension values and set tolerances.
- Set the display of dimensions to decimal or fractional format and adjust the settings for decimal places, trailing zeros, and angular dimension units.
- Change the way the dimension is displayed (basic or inspection).
- Format the positioning and decimal places for dual dimensions.
- Show and erase dimension witness lines.

- Modify the dimension text, including adding symbols, prefixes, and postfixes.
- Define the dimension text style, including font type, height, line spacing, and color.

Note: You can modify the dimension properties by right-clicking the selected dimension and clicking **Properties** on the shortcut menu.

Globally Formatting Dimensions

You can format dimensions globally by using either the **Format** menu or drawing setup file and configuration options. After you define the dimension format setting, such as number of decimal places, the setting applies to all newly created or shown dimensions in the drawing.

Several drawing setup file options contribute to the formatting of dimensions. You can find a complete listing if you sort the drawing setup file by category in the **Options** dialog box (**File** > **Properties** > **Drawing Options**).

About Using an Exact Expression

You can use an exact expression, in the form of `=()`, to specify a value for a dimension. The expression enclosed within the parenthesis is evaluated and the exact value of the dimension is displayed. In an exact expression, the dimension value appears with the minimum number of decimal places required to display the exact value of the dimension. The maximum number of decimal places a dimension value can have depends on the number of significant digits in the dimension value, including those to the left and right of the decimal point. The maximum number of significant digits is 13.

You can use an exact expression to specify the dimension value in the **Dimension Properties** dialog box, in the embedded component in the graphics window, or on the dashboard.

Consider the following points when using an exact expression to specify the value of a dimension:

- You can use an exact expression to override the current global or local setting for the number of decimal places of the dimension value.
- If the number of decimal places required to display the exact value of a dimension is less than the default number of decimal places (as defined by the `default_dec_places` configuration option), the number of decimal places for the dimension will be set to the default value.
- Ensure that all the terms and numbers are enclosed within the parenthesis of the expression. For example, =(a/b – 12.543).
- You cannot specify a relation type expression of the form =(). For example, =(a/b) + 5, =(a/b) / (c+d), and so on.
- All parameter names and the dimension symbol appear only while you enter the dimension value in the form of an exact expression. After you finish specifying the dimension value, you will not see the exact expression, only the evaluated dimension value with the exact number of decimal places is displayed.

- When editing a dimension value specified using an exact expression, you will not see the original exact expression, but only the evaluated value of the dimension appears. If necessary, enter the required exact expression again to specify the value of the dimension.
- You can use an exact expression to specify the value of dimensions in a family table. You can use a combination of exact and non-exact expressions to specify the value of a dimension in a family table, but only for different dimensions representing different columns in a family table. You cannot use a combination of exact and non-exact expressions to specify the value within a single family table dimension (column). If you specify a non-exact expression for a dimension that used an exact expression earlier, all instance values are converted to the non-exact form of the dimension value.

To Format Existing Dual Dimensions in a Drawing

You can detail your model using dual dimension schemes, such as English and metric. The following procedure enables you to format the properties of an individual set of dimensions.

Note: Before continuing, make sure dual dimensions are enabled with the `dual_dimensioning` drawing setup file option.

1. Select the dimension to modify.
2. Click **Edit** > **Properties**. The **Dimension Properties** dialog box opens.
3. Click the **Properties** tab.
4. Under **Dual dimension**, define how to format the dimensions:

- To place the secondary dimension below the primary dimension, select **Below**.
- To place the secondary dimension on the same line as the primary dimension, select **To Right.**
- To specify the number of decimal places to use for the secondary dimension, type a value in the **Dual decimal places** box.

Note:

- If you are using metric dimensions as primary or secondary units in dual dimensions, the metric dimensions do not display as fractions if the fraction format is set.
- When you are using dual dimensioning with dimensional tolerances, the system rounds the secondary values so that they always fit within the range indicated by primary units (that is, so that they are *tighter* than primary values), and retain the desired design intent.

To Show Dimensions in Fractional Format

You can change the format of the dimensions to either decimal or fractional.

1. Select the dimensions to convert from decimal to fractional format. Press `CTRL` or use region selection for multiple selections.
2. Right-click and click **Properties** from the shortcut menu. The Dimension Properties dialog box opens.
3. Click the Properties tab.
4. Under **Format**, click **Fractional**. You can specify a denominator.
5. Click **Ok**.

To Set the Decimal Places and Trailing Zeros

You can define the number of decimal places and trailing zeros for the dimensions within your drawing. You can set the number before or after the dimension is created or shown in a drawing.

To Set the Decimal Places for Existing Dimensions

1. Select the dimension or note with numeric text. For multiple selections, use region selection or hold down the CTRL key and select them.
2. You can set the decimal places and trailing zeros in either of the following ways:

- Right-click the selected dimensions and click **Properties** or click **Edit** > **Properties**. The **Dimension Properties** dialog box opens.

  On the **Properties** tab, under **Format**, click **Decimal** and then type a value in the **Number of decimal places** box.

- Click **Format** > **Decimal Places**. You are prompted to type a number of decimal places.

  Type a value and press `ENTER`. Any dimensions selected, created or shown are displayed with the designated number of decimal places and trailing zeros.

To Set the Decimal Places Prior to Displaying Dimensions

1. Click **Format** > **Decimal Places**. You are prompted to type the number of decimal places.
2. Type a value and press ENTER. Any dimensions created or shown are displayed with the designated number of decimal places and trailing zeros.

You can also set the number of decimal places using the following options:

- To change the default number of decimal places for non-angle dimensions, set the `default_dec_places` configuration option to a value of `0` through `14` (the default is `1`).

- To change the default number of decimal places for angle dimensions, set the `default_ang_dec_places` configuration option to a value of `0` through `14` (the default is `1`).
- Remove the trailing zeros to conform to ANSI standards by setting the `draw_ang_unit_trail_zeros` drawing setup file option to `yes` (the default). This applies only if the entire seconds or seconds/minutes field is exactly zero. Select **Fractional** and type [3600] in the **Largest Denominator** box to be sure.

Note: Using exact expressions when entering a dimension value will cause the number of decimal places in the dimension to be automatically adjusted based upon the calculated value of the expression.

To Show Angles in Degrees

1. Select the dimensions to display in degrees. Press `CTRL` or use region selection for multiple selections.
2. Right-click and click **Properties** from the shortcut menu. The Dimension Properties dialog box opens.
3. Click the **Properties** tab.
4. Under **Format**, select one of the following from the **Angular Dimension Units** list:

- **Degrees**—Show the angular dimension in degrees
- **Degrees, Min**—Show the angular dimension in degrees and minutes
- **Degrees, Min, Sec**—Show the angular dimension in degrees, minutes and seconds.

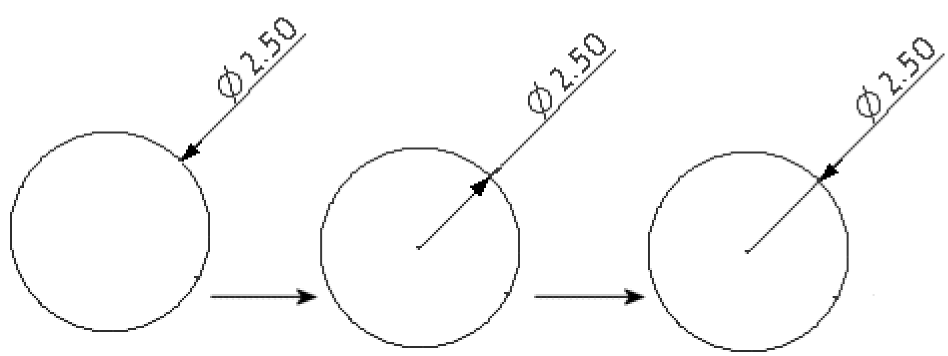
5. Click **OK**. The angular dimension is changed to the specified unit.

Note: To change the angular dimension default setting, use the `draw_ang_units` drawing setup option.

To Change the Dimension Arrow Directions

You can flip through the various arrow directions for each text orientation setting.

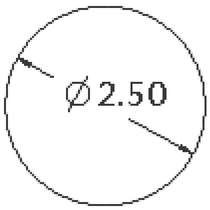
For example, when the `text_orientation` drawing setup file is set to `parallel` the **Flip Arrows** command will cycle through the following arrow displays:



You can change the arrow direction of a dimension in the following ways:

- Select the dimension, right-click, and click **Flip Arrows** on the shortcut menu.
- Select the dimension and click **Edit** > **Properties** to open the **Dimension Properties** dialog box. Under **Display**, click **Flip Arrows**.

Note: If required, you can drag diameter dimensions within the circle that you are dimensioning as shown in the following figure.



To Change the Arrow Directions of Radius Dimensions

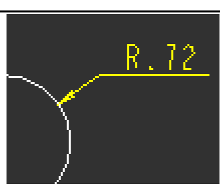
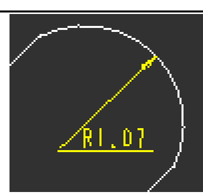
You can change the arrow direction of a radius dimension, for each text orientation setting using the **Flip Arrows** command.

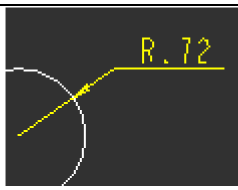
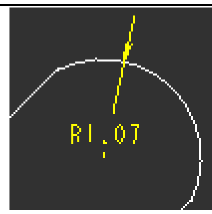
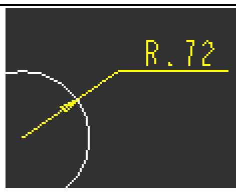
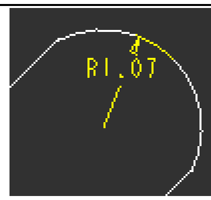
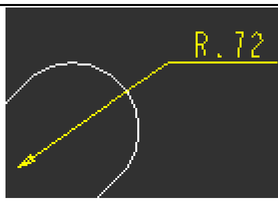
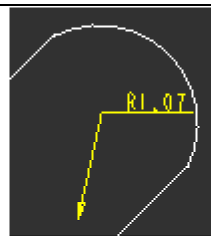
You can change the arrow direction of a dimension in the following ways:

- Select the dimension, right-click and click **Flip Arrows** on the shortcut menu.
- Select the dimension. Click **Edit** > **Properties**. The **Dimension Properties** dialog box opens. Under **Display**, click **Flip Arrows**.
- Select the dimension and drag it. Keep the dimension selected and right-click while moving it. The arrow direction changes every time you right-click.

When the text orientation of the selected radius dimension is set to either horizontal or parallel orientations, the four flip arrow states are as follows.

Note: The following table displays flip arrow states of a radius dimension with text orientation set to **Above extended elbow (ISO)**.

Dimension text placed outside the selected entity	**Dimension text placed inside the selected entity**	**State of the flip arrow**
		Default

Dimension text placed outside the selected entity	Dimension text placed inside the selected entity	State of the flip arrow
		After the first flip
		After the second flip
		After the third flip

Note: Use the **Text orientation** list in the **Dimension Properties** dialog box to change the text orientation of a dimension.

To Display Dimension Text Symbols

The following procedure provides a more permanent replacement of dimensions with symbols. You can toggle the display of all dimensions temporarily by clicking **Info > Switch Dimensions**.

1. Select the dimensions containing text to replace with symbols. Press `CTRL` or use region selection for multiple selections.
2. Right-click and click **Properties** from the shortcut menu. The Dimension Properties dialog box opens.
3. Click the **Dimension Text** tab. In the **Dimension Text** box, note the current `@D` setting.
4. Backspace over the `D` and replace it with an `S`.
- Type a prefix or a postfix if you want to add text to the value of the dimension.
5. Click **OK**. The selected dimensions now display their symbolic values rather than their dimension values.

Note: To change the dimension back to normal display, use the preceding procedure and modify @S back to @D or @O (for drawing dimensions that show text instead of a dimension value). You can replace the symbol without invalidating relations where the dimension appears.

To Convert Diameter Dimensions to Linear Dimensions

You can convert a diameter dimension to a linear dimension, or convert a linear dimension that denotes diameter of a geometry, to a diameter dimension without switching the view. When you switch views for one of these converted dimensions, Pro/ENGINEER tries to display the dimension in the format prior to the conversion, followed by the converted format type, and finally in its original displayed view.

1. Select the diameter dimension to convert to linear.
2. Click **Edit > Properties**. The **Dimension Properties** dialog box opens.
3. Click the **Show as linear dimension** check box and click **OK**.

Note: You cannot convert a linear dimension such as, the depth of an extruded slot to a diameter dimension.

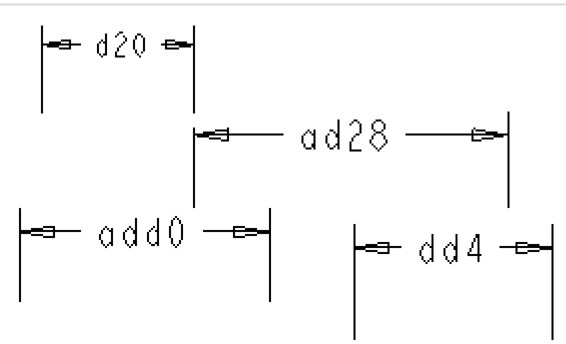
Using Driven and Reference Dimensions in Relations

You can use driven ("ad" type) and reference dimensions on the right side of relations only (to drive the value of other dimensions or parameters).

If you have used a driven dimension in a relation to drive a model dimension, the model dimension also changes until you regenerate the model in Drawing mode. You can use reference dimensions to drive model dimensions because they regenerate in Part or Assembly mode; however, you should not use driven dimensions to drive model dimensions because incorrect model dimensions might occur if you change the model in Part or Assembly mode. To add relations, use the **Tools > Relations** command.

Switching to Dimension Symbols

Use **Info > Switch Dimensions** to show the dimension symbol, and identify the symbol type. The figure below shows dimensions that have been switched to show their symbol displays. The letter combinations define the symbol type, and the number is a unique identifier for the dimension. The letter and number combination create a unique reference designator for the dimension that is used in advanced techniques such as defining relational formulas between dimensions.



- **d**—Shown (driving) dimension created during feature creation in the model. These dimensions are associative bidirectionaly between the model and the drawing.
- **ad**—Created (driven) dimension which is associative to the model and stored with the model but cannot be modified in the drawing. These dimensions update in the drawing only when model geometry is modified. "ad" dimensions result when the config.pro options "create_drawing_dims_only" and "draw_models_read_only" have values of "no", which are the default values.
- **add**—Created draft dimension which is still associative to the geometry or draft entity which it references, but gets stored with the drawing. "add" dimensions result when the config.pro options "create_drawing_dims_only" or "draw_models_read_only" have values of "yes". To create "add" dimensions for referencing draft geometry, the Drawing Setup file option "associative_dimensioning" must be set to "yes".
- **dd**—Draft dimension which is not associative to the draft entity which it references. To create "dd" dimensions, the Drawing Setup File option "associative_dimensioning" must be set to "no", which is the default value.

To Modify the Value of Dimension Symbols

1. Select the dimension (Hold down the CTRL key for multiple selections.)
2. From the right mouse button shortcut menu, click **Properties.**
3. In the **Dimension Properties** dialog box, the value in the **Name** box is the current dimension symbol. Type new text to change the symbol value.
4. Click **OK**. The system stores the new dimension symbol as modified in the model and updates any relations using the symbol.

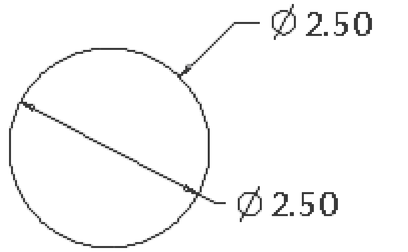
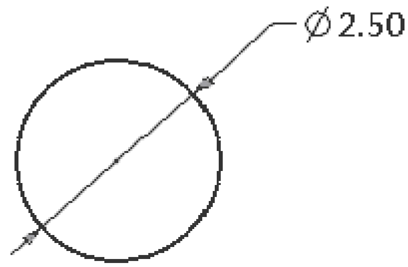
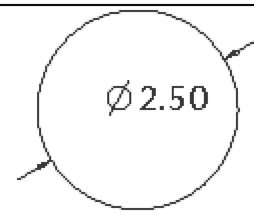
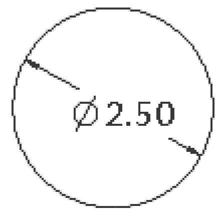
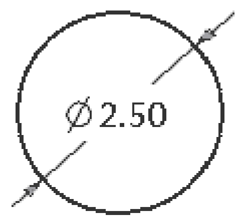
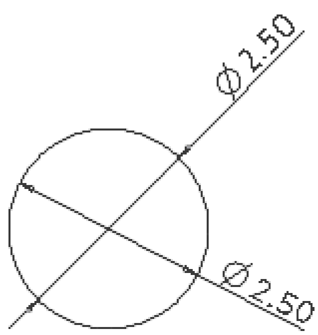
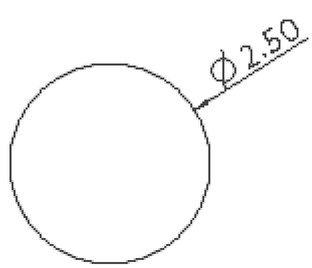
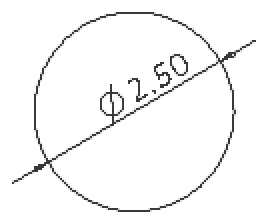
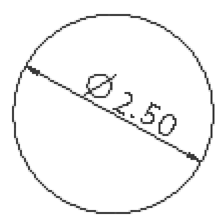
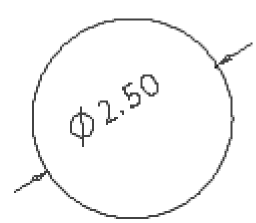
Controlling Diameter Dimension Orientation

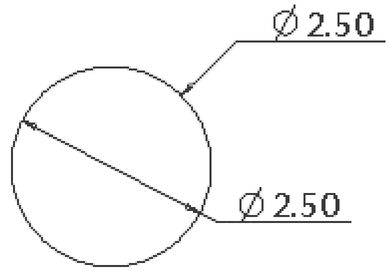
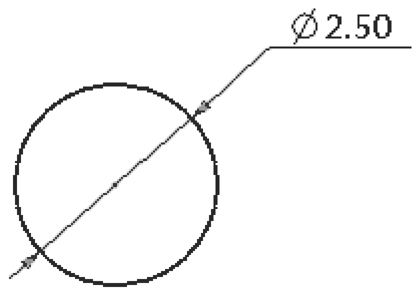
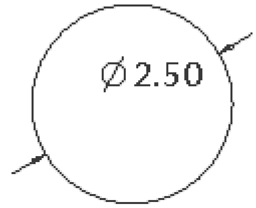
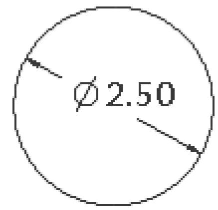
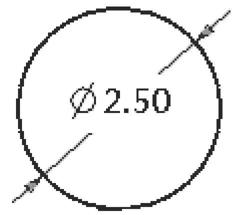
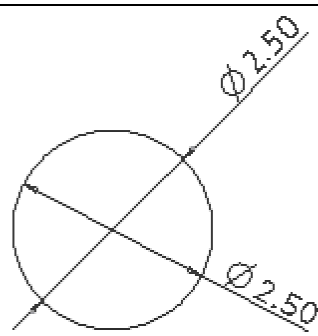
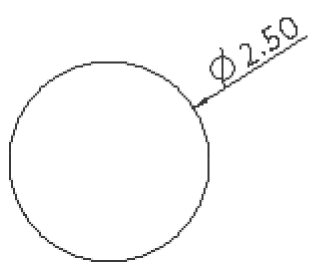
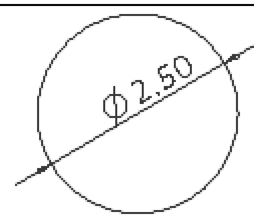
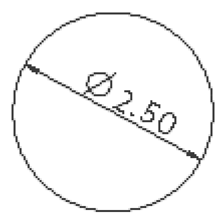
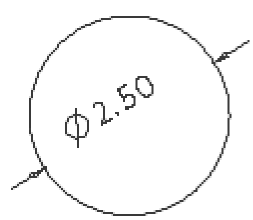
You can control the display of diameter dimensions by:

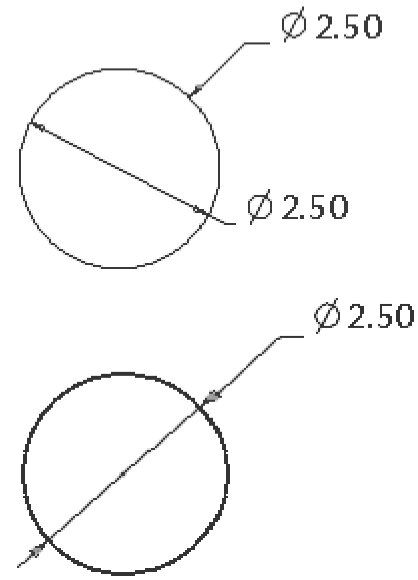
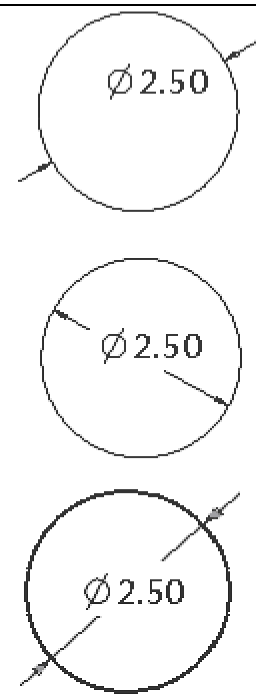
- Setting the `text_orientation` drawing setup file option.
- Right-clicking the dimension and clicking **Flip Arrows** on the shortcut menu. This switches between the three options when repeated. The order of cycling is as follows:

  One arrow inward > Then two arrows inward > Then two arrows outward.
- Right-clicking the dimension and clicking **Properties** on the shortcut menu. You can then select the appropriate display from the **Text orientation** list.

The following orientations are available for diameter dimensions:

Orientation Type	**Flip Arrow**	**Arrows Dragged Within**
horizontal	 	  
parallel	 	  

Orientation Type	**Flip Arrow**	**Arrows Dragged Within**
parallel_diam_horiz	 	  
iso_parallel	 	  

Orientation Type	Flip Arrow	Arrows Dragged Within
iso_parallel_diam_horiz		

Working with Dimension Text

About Automatic Placement of Dimension Text

When you create linear and angular dimensions, including dimensions with geometric tolerances, within the zone of the dimensioned edges or entities, the dimension text is automatically centered between the witness lines of the dimensions. Automatic placement of dimension text is possible only when you create linear and angular dimensions in the 2D mode and not for created and modified diameter, radius, true arc length, ordinate dimensions, and all driving dimensions.

- If the dimension text overlaps the arrows, the arrows are automatically flipped.
- If the dimension text does not fit between the witness lines, it is automatically displayed outside the witness lines, on the side closest to the selected placement point.
- If the arrows of such dimensions do not fit between the witness lines, they automatically flip to the outside.
- If the dimension is placed outside the zone and the arrows overlap, the arrows flip automatically.

While moving driven linear and angular dimensions, the dimension text automatically snaps to the center about the witness lines of the dimensions.

Note:

- Pro/ENGINEER uses the value as calculated by the `dim_text_gap` drawing setup file option to determine the distance for calculation of overlap.
- The gap size, or overlap, is calculated using the following formula.

  gap size = value of `dim_text_gap` * value of `drawing_text_height`

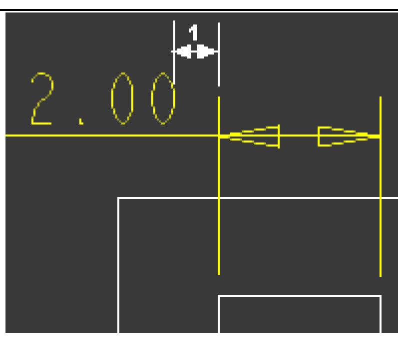
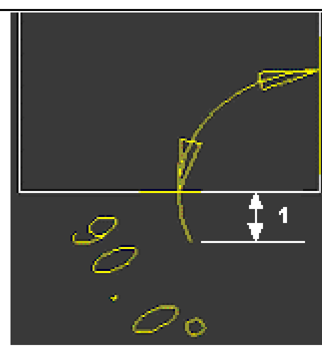
- The zone is a theoretical area, the boundaries of which are defined by the boundaries of the edges or entities selected for dimensioning. If you place the dimension outside the zone, the dimension text is not automatically centered.

About Automatic Placement of Linear and Angular Dimensions

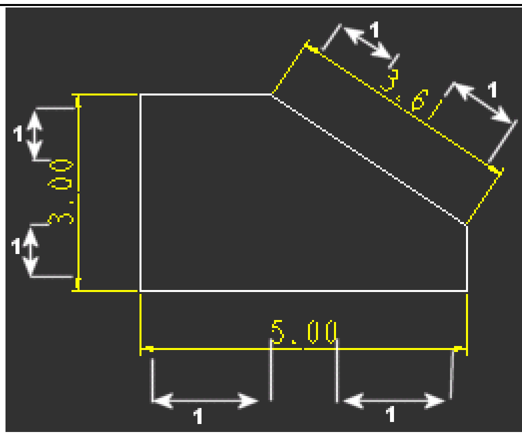
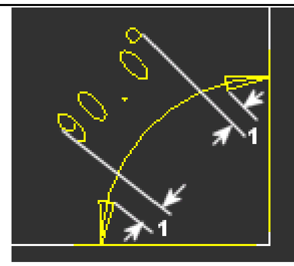
By default, the dimension text of a created linear or angular dimension is centered about its distance between the witness lines if the text fits between the witness lines and is placed within the zone. The perpendicular distance of the dimension text from the edge it dimensions is defined by where you middle-click to initially place the dimension while creating it.

The following rules apply to automatic placement of dimension text for linear and angular dimensions:

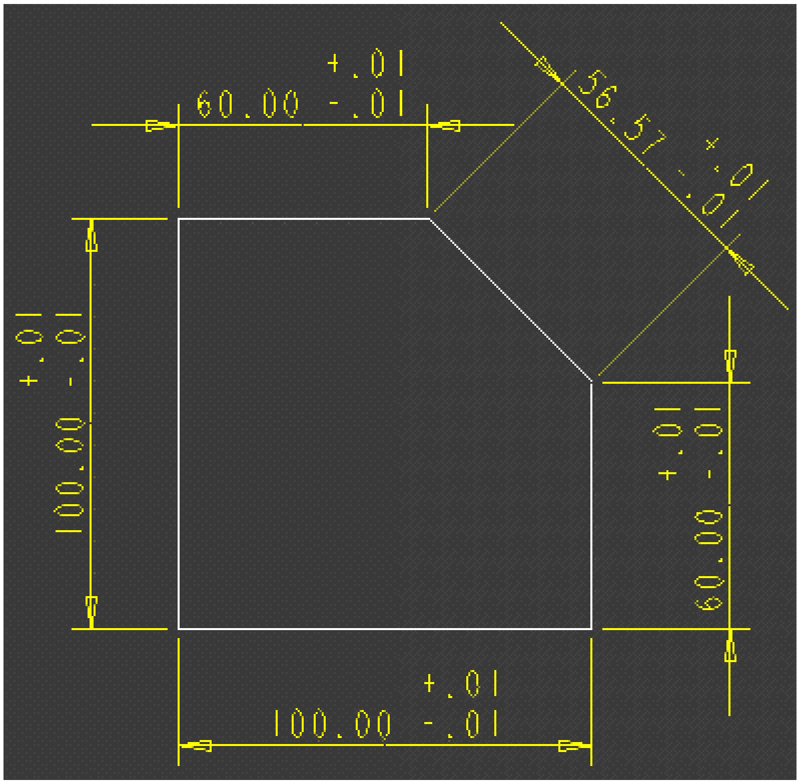
- Dimension text is considered to fit between witness lines if both the first and the last character of the dimension text, including the tolerance value, do not overlap or lie outside a witness line when you create a dimension. If the dimension text fits, it is centered between the witness lines.
- If either the first or the last character of the dimension text overlaps a witness line when you create a dimension, the dimension text is placed outside the witness lines. The distance at which the dimension is placed away from the closest witness line is the value as calculated by the `dim_text_gap` drawing setup file option.

Linear dimension	**Angular dimension**
	
1 The distance at which the dimension is placed away from the closest witness line as calculated by the `dim_text_gap` drawing setup file option	

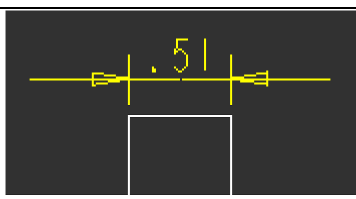
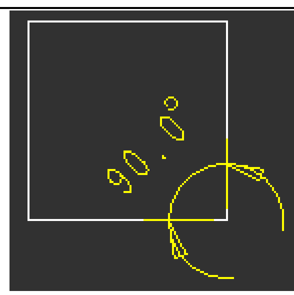
- Dimension arrows are considered to fit between witness lines, and remain inside the witness lines, if the distance between the arrows is at least the value as calculated by the `dim_text_gap` drawing setup file option.
- The distance between the first character of the dimension text and its closest arrow and the distance between the last character of the dimension text and its closest arrow should be at least the value as calculated by the `dim_text_gap` drawing setup file option. In case of dimension text with a single character, the distance between the character and the arrows on either side should be at least the value as calculated by the `dim_text_gap` drawing setup file option.

Linear dimension	**Angular dimension**
	
1 Distance between the dimension text character and the closest arrow	

- If the distance between the first or the last character of the dimension text or geometric tolerance, and the closest witness line is less than the value as calculated by the `dim_text_gap` drawing setup file option, the arrows are flipped.



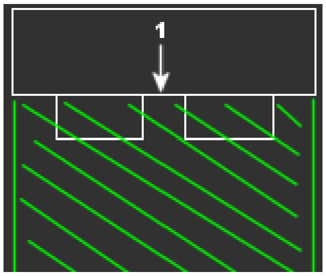
- If the distance between the arrows of a dimension is less than the value specified by the `dim_text_gap` drawing setup file option, the arrows are automatically flipped to the outside.

Linear dimensions	Angular dimensions
	

About Zones for Linear and Angular Dimensions

The zone is a theoretical area defined by the boundaries of the entities that you select for dimensioning. By default, the dimension text of a created dimension, linear or angular, is centered about its distance between the witness lines if it fits between its witness lines and is placed within the zone.

Zone for Linear Dimensions



1. Selected reference

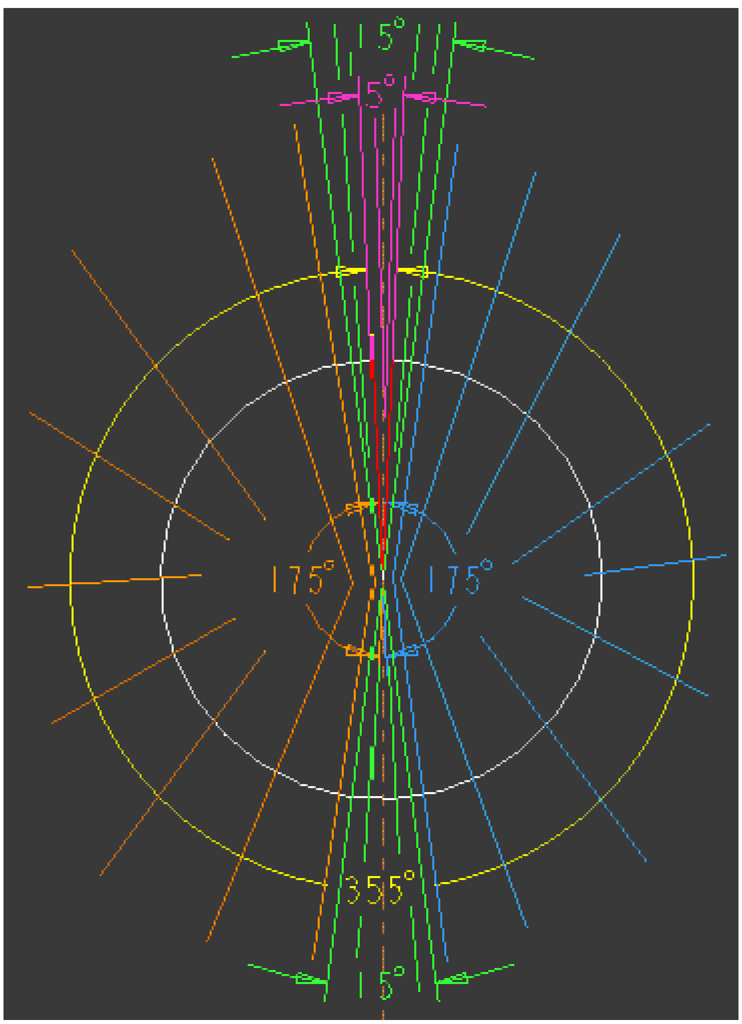
Zone

Zones for Angular Dimensions

For angular dimensions, the dimension text is always within one of the four zones,

based on the location you select. The minimum selection zone for an acute angle dimension is 15 degrees, centered about the theoretical centerline of the angle.

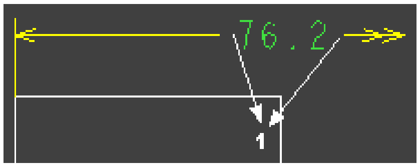
The figure below shows the different zones for angular dimensions.



- In the figure, a 5-degree angular dimension is created by the 2 red lines.
- If you select a placement point within the selected entities, (the area in magenta in the figure), the 5-degree dimension is created and placed within the entities based on the rules for automatic placement of dimensions.
- If you select a placement point outside the 5-degree zone but within the 15-degree zone as indicated by area in green in the upper part of the figure, then the 5-degree dimension is created and placed at the selected point.
- If you select the placement point outside the 15-degree zone as indicated by the areas in orange and blue, then 175-degree dimensions are created and placed based on the rules for automatic placement of dimensions.
- If you select the placement point within the 15-degree zone opposite the selected entities, as indicated by the area in green in the lower part of the figure, then a 355-degree dimension is created and placed based on the rules for automatic placement of dimensions.

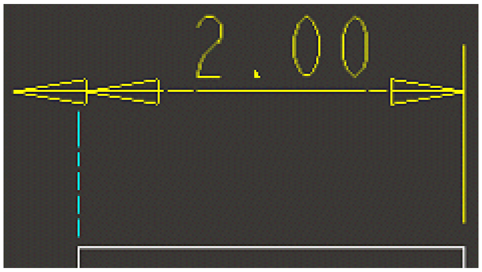
About Clipped Dimensions

During creation of clipped dimensions, arrowheads do not overlap. There is at least a small gap, equal to the value as calculated by the `dim_text_gap` drawing setup file option, between the dimension text and each arrowhead as shown in the image below.



1. Gap on each side of the dimension text

The arrowhead of the clipped portion of the dimension does not align with the view border when you create a clipped dimension as shown in the image below.



In case of clipped dimensions, the dimension text is not automatically placed outside the witness lines.

About Moving a Linear or Angular Dimension

When you move a linear dimension within the zone, it automatically snaps to the midpoint of the witness lines.

When you move an angular dimension, it automatically snaps to the center of the angle being dimensioned. If you move the dimension to the opposite side of the angle, it snaps to the center of the angle, on the opposite side.

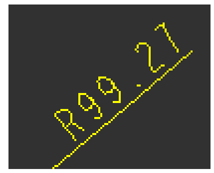
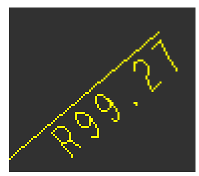
Note: Snapping occurs for both driving and driven dimensions, except for clipped dimensions.

To Control Dimension Text Orientation

You can globally control the positioning of dimension text, relative to the leader lines, by setting the `text_orientation` drawing setup file option. Individual text orientation modification is available only for diameter, radius, and chamfer dimensions. To modify the text orientation of individually selected dimensions, use the **Dimension Properties** dialog box.

1. Open the **Dimension Properties** dialog box, using one of the following methods:
    - Select the dimension that you want to modify and double-click.
    - Select the dimension and click **Edit** > **Properties**.
    - Select the dimension, right-click, and click **Properties** from the shortcut menu.
2. Under the **Properties** tab, select from the following options in the **Text Orientation** list:

Default	Uses the default value from the `text_orientation` drawing setup file option.
Next to and centered about elbow (ASME)	Positions dimension text horizontally. Text is vertically centered about the elbow line (ASME standard). This option is not available for chamfer dimensions. 
Next to and above elbow (ISO)	Positions dimension text horizontally. Text is above elbow line (ISO standard). 
Above extended elbow (ISO)	Positions dimension text horizontally. Elbow is below text and extends to width of

	text (ISO standard). 
Parallel to and above leader (JIS)	Positions dimension text parallel to and above leader line (JIS standard). 
Parallel to and below leader (JIS)	Positions dimension text parallel to and below leader line (JIS standard). 
(As Is)	Does not change orientation text style of selected dimensions.

Depending on the text orientation style you select, the text is displayed on the screen.

3. Click **OK** to close the **Dimension Properties** dialog box.

Note: The `angdim_text_orientation` drawing setup file option controls the display of angular dimensions.

To Add Text to a Dimension

Use this procedure to add prefix or "postfix" text to a dimension.

1. Select the dimension to modify.
2. Click **Edit** > **Properties**. The **Dimension Properties** dialog box opens.
3. Click the **Dimension Text** tab.
4. Use the **Name**, **Prefix** and **Postfix** boxes to surround the dimension text.
5. Click **OK.** The system stores the dimension text with the model, and displays it in either Part mode or Assembly mode when you select the feature the dimension is driving.

To Add Parameters to Dimension Text

Use this procedure to add prefix or "postfix" text to a dimension.

1. Select the dimension to modify.
2. Click **Edit** > **Properties**. The **Dimension Properties** dialog box opens.
3. Click the **Dimension Text** tab.
4. Type the parameter callout in the **Prefix** or **Postfix** field.
5. Click **OK**.

Note: You can place dimensions and parameter text in driven dimensions or associative dimensions owned by the same model; however, you cannot place them in draft dimensions or associative draft dimensions.

To Overwrite Dimension Value Display with a Text String

Use this procedure to replace a driven dimension value with a text string. Driven dimensions update with changes made in the 3D-model, but they cannot be used to modify the 3D-model from the drawing interface.

1. Select the dimension to modify.
2. Click **Edit** > **Properties**. The **Dimension Properties** dialog box opens.
3. Click the **Dimension Text** tab.
4. In the text field, replace the symbol `@D` with `@O` and type the text you want. The symbol `O` represents overwrite; anything you type following this text overwrites the dimension value. The drawing now displays the text string instead of the dimension value.

Note:

- Dimensions must always have a displayed value. If you do not type any text after replacing `@D` with `@O`, Pro/ENGINEER retains the dimension value of the selected dimension. In case of a reference dimension, if you do not type any text after replacing `@D` with `@O`, Pro/ENGINEER retains the value of the selected reference dimension and does not display `REF`.
- You cannot overwrite dimension values for driving dimensions. If you replace `@D` with `@O` and type the text you want for a driving dimension, the dimension value is not overwritten. Instead, the text is appended to the displayed dimension value.

To Set The Default Chamfer Dimension Text Display

By setting the configuration file option `chamfer_45deg_dim_text`, you can control the display of chamfer dimension text without affecting the leader. This only affects the text of *newly created* dimensions. To modify dimensions that you created before changing the setting, you must manually edit the text. Using the drawing setup file option `chamfer_45deg_leader_style`, you can control the leader type of a chamfer dimension without affecting the text. You must use `chamfer_45deg_dim_text` and `chamfer_45deg_leader_style` together to meet the appropriate standard. To control the use of leading or trailing zeros in the dimension value, use the drawing setup file option `lead_trail_zeros`.

To Relate Detail Items and Dimension Text

You can directly relate a note, draft geometric tolerance, or symbol to dimension text so that the detail item moves with the dimension when the dimension changes location.

To relate an existing detail item to dimension text,

1. Select all items you want to relate to the host object.
2. Click **Edit > Group > Relate to Object**.
3. Select the "host" dimension. The object or objects are grouped and selected as one.

To remove objects from the group, use **Edit > Group > Unrelate**.

Note: When creating a *new* symbol, note, or free draft geometric tolerance, you can relate it to a dimension when specifying its placement.

Working with Dimensional Tolerances

About Dimensional Tolerances in Detailed Drawings

You can set the tolerance standard as ANSI or ISO, and set the tolerance display on or off. You can drive dimensional tolerances using a set of tolerance tables. The system assigns each model a tolerance standard of either ANSI or ISO.

- When you switch from ISO to ANSI, the system assigns the ANSI tolerances based on the nominal dimension’s number of digits and deletes the tolerance table reference.
- When you switch from ANSI to ISO, a set of tolerance tables drives the ISO-standard tolerances.

Loading the System and User-Supplied Tables

The configuration file option `tolerance_table_dir` sets the default directory for a user-defined tolerance table. All Holes and Shafts tables overwrite existing tables when loaded. When loading General and Broken Edge tables, keep in mind the following:

- If you load one table that has the same set of class names as the model’s, the system accepts the new table.
- If you load a table that contains class names that conflict with those already loaded in the system, the system does not load those class names.

- If you load two tables with class names that do not conflict with those in the system, but that are different from them, you overwrite the ones in the system.
- If the default class of the model does not exist in the new names, you must specify a new class.

After you load the new tables, the system assigns the new dimension tolerances and you can regenerate the model.

When you regenerate the model the system goes through all of the dimensions and reassigns their tolerances from the tolerance tables. If you modify a dimension tolerance, the system deletes the tolerance table reference for that dimension, and the tolerance value remains the same until you modify it again or reassign the tolerance table.

If the regeneration fails, all relevant dimensions acquire the backup tolerance values. However, the new tolerance tables remain.

Changing the Tolerance Table Reference

All instances in a family share the same set of tolerance tables, the same tolerance standard, and the same class. When changing the tolerance table reference, keep in mind the following:

- If you modify model units, but keep all of the dimension values the same, the system updates the tolerance values to reflect the change in the overall model size.
- If a Holes or Shafts tolerance table drives a dimension’s tolerances, you cannot show it in a plus-minus symmetric format. The system assumes that the General and Broken Edge tables have symmetric values.
- If you place a dimension tolerance in a family table, the system deletes its tolerance table reference. Also, if you switch a model from ANSI to ISO, or vice versa, it preserves the tolerances in the family tables and does not assign table references to those dimensions.
- If a dimension value falls outside ranges specified in the table, the system uses the closest range to obtain tolerances (that is, it uses the last range in the system table (2000–4000) to determine tolerances for dimension values of 2000 or greater).

Setting the Tolerance Display

To set the tolerance display on and off, use the drawing setup file option `tol_display`. You can set the default display for dimension tolerances using the configuration file option `tol_mode`.

Using the configuration file option `maintain_limit_tol_nominal`, you can maintain the nominal value of a dimension regardless of the changes that you make to the tolerance values. If you set it to `yes`, the system does not modify the **Nominal Value** of a dimension with a **Limits** tolerance format when you set the format to **Limits** or change the value of the upper or lower tolerance.

To Set the Tolerance Display for Individual Dimensions

1. Select a dimension.
2. Right-click and click **Properties** on the shortcut menu. The **Dimension Properties** dialog box opens.
3. Select a tolerance format from the **Tolerance Mode** list and specify values as follows:
    - For the **Limits** format, specify values in the **Upper Tolerance** and **Lower Tolerance** boxes.
    - For the **Nominal** and **Plus-Minus** formats, specify values in the **Nominal Value**, **Upper Tolerance**, and **Lower Tolerance** boxes.
    - For the **+-Symmetric** and **+-Symmetric (Superscript)** formats, specify values in the **Nominal Value** and **Tolerance** boxes.

      Note:

      If you select the **+-Symmetric (Superscript)** format, the dimension tolerance value is aligned to the dimension text in such a way that it begins at the half-way point of the dimension text extending up.

      Dimensions that are below dimension leader lines automatically adjust such that the corresponding text does not overlap with their dimension leader lines.
4. Click **OK**. When tolerances appear, Pro/ENGINEER lists the default tolerance values in the lower-right corner of the graphics screen.

To Create an ISO-standard Model in Drawing Mode

1. Click **File > Properties**. The Menu Manager opens.
2. Click **Tolerance Standards > TOL Setup > Standard**.
3. Click **ISO/DIN** or **ANSI**.

Pro/ENGINEER loads the system and user-supplied tables, and the General table drives all dimensions.

Note:

The system loads the tolerance tables into the model when you create it as an ISO-standard model or switch the tolerance standard from ANSI to ISO. To create the model as an ISO-standard model, set the configuration file option `tolerance_standard` to `ISO`. Since the tables determine how the model regenerates, the system stores them permanently with the model, and you can only use them with driving dimensions. Four types of tolerance tables are available:

- General (one per model)
- Broken Edge (one per model)
- Holes (several per model)

- Shafts (several per model)

When you create a dimension, the system assigns it to the General table. When you assign the dimension to the tolerance table, the tolerance table and its dimension value now govern the tolerance values of the dimension. You can switch the tolerance table reference of the dimension to any other table. Dimensions in ISO models, which are driven by Holes or Shafts tables, appear as follows:

PLUS MINUS—5.69f7(+0.01)
NOMINAL—30f7(5.00)
LIMITS—5.69f7

To Change the Tolerance Class

Each ISO-standard model has an extra attribute called the *tolerance class* which determines the general coarseness of the model. The configuration file option `tolerance_class` sets the default tolerance class for ISO models (the default is `medium`). The system uses the tolerance class together with the dimension value when retrieving tolerances for General or Broken Edge dimensions.

1. Click **File > Properties**. The Menu Manager opens.
2. Click **Tolerance Standard > TOL SETUP** > **Model Class**.
3. From the **TOL CLASSES** menu, select a class name.
4. All dimensions driven by the General or Broken Edge table obtain new tolerance values. Regenerate the model.

To Change the Tolerance Table Reference

1. Click **File > Properties**. The Menu Manager opens.
2. Click **Tolerance Standards** > **Tol Tables**. The **TOL TBL ACT** menu displays these commands:

- **Modify Value**—Displays the tables in the Tol Tables menu.
- **Retrieve**—Retrieves a set of tables into the model.
- **Save**—Saves the tolerance table.
- **Show**—Displays the tolerance table, as shown next.

TABLE_TYPE	GENERAL
TABLE_NAME	DEFAULTS
TABLE_UNIT	MILLIMETER
RANGE_UNIT	MILLIMETER
DESCRIPTION	0.05–3
FINE	0.05–

MEDIUM	0.1
COARSE	0.2
VERY COARSE	0.5

3. Do one of the following:
    - Click **Modify Value**, select the table by choosing **General Dims** or **Broken Edges**, and select the dimensions.
    - Click **Holes** or **Shafts**; then type the table name and class number.

To Modify Dimensional Tolerances in a Note

You can create a note in which you substitute a dimensional tolerance, entered as a parameter, by its value. Use the format:

`&linear_tol_0_0`

to display the tolerance at one decimal place. Add a "0" to the string for each decimal place you want to the note to read. For example, the above format reads as "0.1."

`&linear_tol_0_00 reads as` "0.01."

You can use a similar format to include an angular tolerance value in a note. For example, type [`&angular_tol_0_0`] to display "0.5" (the default for one decimal). [`&angular_tol_0_00`] to display 0.05 etc.

Because the system includes tolerances in a note as parameters, associativity exists between the tolerance value in a note and a model. When you change tolerance values in a note, the system updates the default tolerance table in Part mode.

To modify the tolerance value in a note

1. Select the note.
2. From the right mouse button pop up menu, click **Properties.**
3. Type a new value for the tolerance. Click OK.

Note: Tolerance tables do not affect thread dimensions. Thread dimensions come into the model having the same tolerances with which the system stored them.

Example: A Tolerance Table

TABLE_TYPE	HOLES				
TABLE_NAME	A				
TABLE_UNIT	MICROMETER				
RANGE_UNIT	MILLIMETER				
BASIC SIZE	9	10	11	12	13

—3	295/270	310/270	330/270	370/270	410/270
3 - 6	300/270	318/270	345/270	390/270	450/270
6 - 10	316/280	338/280	370/280	430/280	500/280
10 - 18	333/290	360/290	400/290	470/290	630/300
18 - 30	352/300	384/300	430/300	510/300	700/310
30 - 40	372/310	410/310	470/310	560/310	710/320
40 - 50	382/320	420/320	480/320	570/320	800/340
50 - 65	414/340	460/340	530/340	640/340	820/340

Working with Basic Dimensions

About Basic Dimensions

Basic dimensions are theoretically exact dimensions that appear with the measurement value and some associated text enclosed in a feature control frame. If the original dimension has tolerances, Pro/ENGINEER automatically removes them; you cannot add tolerances to basic dimensions. You can transform existing dimensions into basic dimensions, and set inspection dimensions according to the DIN standard.

To Set a Dimension as a Basic Dimension

1. Select the dimension to change.
2. Click **Edit** > **Properties**. The **Dimensions Properties** dialog box opens.
3. In the **Display** field, click **Basic**.
4. Click **OK**.

Note: If you set the dimension type to **Basic**, then by default, only the numerical part of the dimension value and it's symbol are enclosed in a rectangular box. Any additional text in the dimension value is not included in the box.

To Set a Dimension as an Inspection Dimension

1. Select the dimension to change.
2. Click **Edit** > **Properties**..
3. In the **Display** field, click **Inspection**. The dimension is enclosed in an oval.
4. Click OK.

Working with Witness Lines

About Modifying Witness Lines

When Pro/ENGINEER places a dimension on a model, it leaves a gap between the model and the witness line. The drawing setup file option `witness_line_offset` controls the actual size of the gap. Sometimes this gap is not visible on a drawing. However, it becomes noticeable when you plot the drawing. To see how the drawing will look when you print it, choose **File > Print > Screen**.

A witness line can have both jogs and breaks. When you add jogs to a witness line that already has breaks, the breaks that were created as simple breaks appear on the first unjogged segment of the witness line. If a witness line has dimension breaks, jogging a witness line relocates a break to a new intersection point.

You can edit witness lines in the following ways:

- Create parametric or simple breaks in dimension witness lines and in leader lines of notes and symbols.
- Resize the gap by dragging the end handles. You can simultaneously clip several dimensions that have the same orientation so that their witness line endpoints all align. Using multiple selection, you can manipulate witness lines in the following ways:
    - Clip both witness lines of a dimension at the same time.
    - Clip the witness lines of many dimensions at once. When you pick on the witness line of each dimension, the system moves each of the endpoints together.
- Erase and resume witness lines.
- Skew witness lines.
- Add jogs to and delete them from linear (standard and ordinate) and angular dimension witness lines, note leaders, and symbol leaders.
- Create and modify angular, diametrical, radial, and linear dimensions with and without an elbow.

To Shorten a Witness Line

1. Select the dimension with the witness line that you want to clip. The witness lines are highlighted, with a "handle" at each end.
2. Drag the handles to the desired positions. Note the two way arrow over the handle shows the direction it will move. Middle click to finish.

To Erase a Witness Line

You can erase only one witness line of a dimension. If you try to erase the second one after erasing the first one, the system erases the second one and restores the first one. You cannot erase both.

1. Double-click the dimension. The Dimension Properties dialog box opens.
2. Click **Erase** from **Witness line** display section.
3. Select the witness line to erase.

Note: You cannot erase the witness lines of ordinate dimensions (including ISO-standard ordinate dimensions with a cross-line between the baseline and the dimension).

You cannot blank the witness line of a clipped dimension (one that has been clipped as a result of view clipping), and you cannot use **Show** to restore a witness line that has been automatically erased (clipped due to view clipping).

To Restore an Erased Witness Line

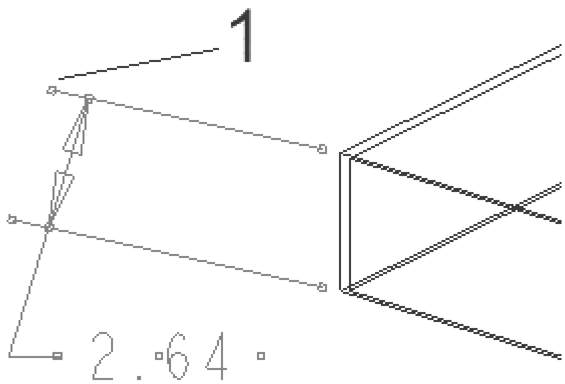
1. Select the dimension with the witness line to restore.
2. Click the right mouse button, and select **Show Wit Line** from the pop up menu. The system redraws the selected dimension and displays its witness line with a single arrowhead at the leader.

To Add a Jog to a Witness Line

1. Click **Insert > Jog**.
2. Select the dimension value. (This selects the witness lines.)
3. Click the point on a witness line to insert the jog. Move the mouse to place the jog.
4. Click again to release the jog. The command is still active and you can repeat the process to place another jog. Middle click to exit the command.

To Skew Witness Lines

1. Select the dimension with the witness line to edit.
2. Use the cursor to select a handle on the end of a witness line. The cursor changes to an arrow, perpendicular to the selected witness line.
3. Drag the handle to the desired location. Left click again to place the witness line.



Drag handle (1) up and down to skew dimension position.

To Create a Break in a Witness Line

1. Click **Insert > Break**. You are prompted to select the line to break.
2. Select the line. The **BREAK TYPE** menu manager opens.
3. Check **Parametric** or **Simple**. A parametric break prompts you to choose a witness line or snap line to break around. You are also prompted for a break size. If you move the line you selected, the break moves with it.

   A simple break lets you insert a break with one click on the line to be broken. You can adjust its size, but it is not associated with any other entity.
4. If you are inserting a parametric break, select the intersecting line. Select a break size to complete the break. You can select another witness line to break, or middle click to exit the command.

   If you are inserting a simple break, click the point on the witness line to create the break. Use the line "handles" to size the break.

Note: You cannot break a radial or diameter dimension of an arc or circle because what you see is not a witness line, but the extension of a dimension arrow.

To Create Leaders Without Elbows

For dimensions that require one leader, you can set an option to eliminate the elbow in the leader.

1. Set the drawing setup file option `default_dim_elbows` to `no`.
2. Create the dimension in the drawing. The system assigns the dimension an elbow with a default value of zero and does the following:
   - Centers the line of text containing the dimension value on the end of the straight leader line.

- Trims back the leader line to the box surrounding all of the lines of dimension text.
- Pads the character height by half.
- Displays the text center-justified about the endpoint of the dimension line.

Intersection Witness Lines

Dimensioning Scheme

To Modify the Dimensioning Scheme of Feature or Part

1. Change the drawing selection filter to `Feature`.
2. Select the feature on the drawing you want to redimension and right-click.
3. Click **Redefine Feature** from the shortcut menu. The feature opens in a new window. If you selected the base feature, only the section of the base feature and dimension values appear in the separate window. If you selected a later feature on the model, the system rolls back the model and displays it in default orientation in a separate window. The section and dimensions of the feature (if applicable) appear on the model.
4. Add and delete dimensions, or change placement constraints, as necessary. You *cannot* modify existing dimension values or add or delete features.
5. Save and close the redimensioned feature. The model regenerates to reflect the changes.

Note: If you set the `auto_regen_views` configuration file option to `no`, you must choose **View > Update > Current Sheet** or **All Sheets** to observe the change in the drawing views.

Working with Leader Lines

To Add a New Leader

1. Select an item to which you would like to add the leader.
2. From the right mouse button pop up menu, click **Edit Attachment.**
3. In the Menu Manager click **MOD OPTIONS** > **Add Ref**.
4. Select a reference point on the view for the new leader. You can use the Ctrl key to select several points if you want to.
5. Click the middle mouse button to complete. The leader or leaders are attached at the specified points.

To Attach a Leader to a New Object

1. Select a leader to modify.
2. Click **Edit > Attachment**. The Attach Type menu opens on the Menu Manager.
3. Highlight the desired options on the Menu Manager. Select a new entity at the point to attach the leader. Click the middle mouse button to finish.

Note: You can also use **Edit** > **Attachment** to move created (driven) dimensions, as well as notes and other detailing items.

To Add a Jog to a Leader

1. Click **Insert** > **Jog**.
2. Select an entity with a leader to which you want to add a jog.

   Alternatively, select the entity, right-click, and click **Insert Jog** on the shortcut menu.
3. Click on the leader at a point where you want to insert the jog and click at a location in the drawing depending upon how you want to place the jog.
4. Middle-click to finish. The jog is created in the leader. The leader remains selected and you can repeat the process to place another jog.

To Delete a Jog from a Leader

1. Select the jog to delete.
2. Right-click and click **Delete** on the shortcut menu. The jog is deleted.

Note:

- To delete all jogs from a leader, select a leader with multiple jogs, right-click, and click **Remove All Jogs** on the shortcut menu.
- If you select an entity with multiple leaders with jogs in them, you can remove all jogs from all the leaders by selecting the entity and clicking **Edit** > **Remove** > **All Jogs**.

To Change the Leader Attachment

1. Select the note.
2. Click **Edit > Attachment** from the right mouse button shortcut menu. The **Attach Type** menu opens on the menu manager.
3. Use the menu manager to change the attachment type or style.

To Change the Arrowhead Style

1. Click **Format > Arrow Style**. The **Arrow Style** menu opens on the menu manager.
2. Highlight the arrowhead style you want to apply in the menu manager, then click an arrowhead in the drawing. You can hold down the Ctrl key to select multiple arrowheads.
3. Click **OK** in the confirmation box. The arrowhead styles are changed.

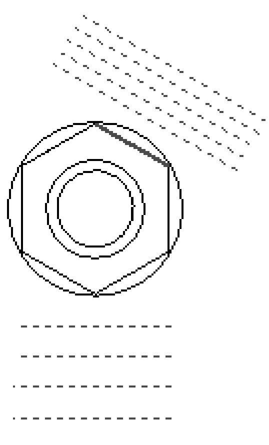
To Move Leader Attachment in a Multi Line Note

1. Double click the note. The **Note Properties** dialog box opens.
2. In the **Text** tab, type "@o" (the letter o) at the beginning of the line of text you want the leader to attach to. Do not leave a space between the symbol and the text.
3. Click **OK**. The leader is attached at the designated line.

Managing Details with Snap Lines

About Managing Drawing Details with Snap Lines

Snap lines are entities in your drawings that help locate dimensions and detailing items, including notes, geometric tolerances, symbols, and surface finishes. After you place an item on a snap line, the item moves with the line. For example, if you place several dimensions on a snap line you can simultaneously reposition all the dimensions by dragging the snap line—while maintaining the alignment of the dimensions.



You can offset snap lines from either drawing view borders, datum planes, draft entities, drawing geometry, or existing snap lines.

Snap lines follow the drawing view or object they are attached to. For example, if you attach a snap line to a view border and then rotate the view by 30 degrees, the snap lines also rotate. Additionally, consider the following:

- Moving a view on a drawing with related draft entities will cause the draft entities and their snap lines to translate the same way.
- If you move the related draft entities without moving the view, the snap lines move with the draft entities. However, if you move the drawing view after this, the draft entities and their snap lines translate with the drawing view.
- If you move draft entities that are not related to any views, the snap lines dynamically move with the draft entities.

If you delete the view borders, datum planes, drawing geometry, or existing snap lines that a snap line is attached to, the snap lines still remain in the drawing. However, if you delete a draft entity, any related snap lines are also deleted. Any items attached to the deleted snap line remain. However, they are unsnapped and will not move together.

Snap lines are positioned relative to the selected view border, datum plane, draft entity, or drawing object. The default snap line length is primarily controlled by the selected object, which means that changes to the selected object will correspondingly change the snap line length as well. After the snap lines have been created you can independently change the snap line length by dragging the handles on the end of the snap line.

- Moving a snap line attached to a view border, draft entity, or drawing object causes all other related snap lines to move accordingly.
- The ladder effect controls the movement of snap lines. For example, if you have 5 snap lines and snap line 1 is attached to a view border, draft entity, or drawing object, moving snap line 1 will cause snap lines 2 through 5 to move. Moving snap line 2 would cause snap lines 3 through 5 to move.

Note:

- Snap lines are not a graphical part of the drawing, so they do not plot.
- The drawing object filter can make selecting snap lines.

To Create a Snap Line

Snap lines enable you to locate dimensions and detailing items in your drawings. After you place an item on a snap line, the item moves with the line.

1. Click **Insert > Snap Line**. The CR SN LINE menu appears on the Menu Manager.
2. Define how you want to offset the snap line; from an object or view:

- **Offset View**—Offset snap lines from a drawing view borders.
- **Offset Object**—Offset snap lines from a drawing object, including geometry, datum plane, and other snap lines.

3. Depending on how you defined the snap line offset, select the appropriate view borders or drawing objects to offset the snap line from. If you select a non-straight drawing object, keep in mind that a straight snap line will be created tangent to the point you select. The selected items are highlighted.

4. Middle-click to confirm your selection. You are prompted for an offset distance. If you are offsetting the snap line from a drawing object, a direction arrow appears, indicating where the snap line will be created.

5. Type the offset distance for the first snap line. A negative value positions the snap line in the opposite direction of the direction arrow. Middle-click to confirm the distance.

   If you are creating multiple snap lines from the same reference, type the number of snap lines to create and then specify the space between the lines.

6. Middle-click to confirm the snap line creation. The snap line is attached to the specified view borders or drawing objects. To leave the snap line creation process, Middle-click or click **Done/Return**.

Note:

- You can change the snap line offset distances by double-clicking the snap line. You are prompted to type the new offset value.
- You can delete a snap line by right-clicking it and selecting **Delete** on the shortcut menu. You could also select the snap line and press `DEL` or click **Edit** > **Delete**.
- You cannot create snap lines off draft entities that are part of cross-hatching.
- Snap lines are not automatically created for draft entities when using **Clean Dimensions**. You must manually create and attach items to snap lines offset from draft entities.

To Place Items on Snap Lines

You can add dimensions and detailing items to snap lines. Any items that are snapped to the line will move with the line.

1. Select the dimension or detailing item you wish to place on a snap line. The item is highlighted.
2. Drag the dimension or detailing item over the desired snap line. If the item can be snapped, the snap line turns purple. After you release the mouse button, the item is snapped to the line.

Note: Under **Default Actions**, select **Snap to Snap Lines**. By default, the system selects both commands. If snap lines do not appear or you have unchecked the **Snap to Snap Lines** checkbox, the items that are already snapped to a snap line continue to move with their snap lines.

To Modify Snap Line Attachment or Spacing

1. Select a snap line and right-click the line to display the shortcut menu.
2. Click **Edit Attachment**. The MOD SNAP ATT menu appears in the Menu Manager. Click one of these commands:
   - **Keep Position**—Maintain the position of the snap line.

- **Change Orient**—Reroute the snap line, changing its orientation by assigning the offset of the existing reference to the new reference.
- **Offset View**—Reroute the snap line attachment to a view border.
- **Offset Object**—Reroute the snap line attachment to other geometry or another snap line.

3. Specify a new reference. The snap line attachment changes.

Note: You can manipulate snap lines in several other ways:

- To remove a snap line, select the line and click **Edit > Delete.** Once you delete a snap line, any items located on it are free.
- To change the position of a snap line, select the line and drag it to its new position. Pro/ENGINEER updates all items located on the moved snap line after you choose a new location.
- To control the length (position of the endpoints) of a snap line, select the line and drag an endpoint (handle).
- To change the spacing of a snap line, select the line, right-click on the line, and then choose **Properties** on the shortcut menu. Enter a new spacing value and press ENTER. Pro/ENGINEER updates all items located on the moved snap line after you choose a new location.

To Control the Display of Snap Lines

By default, snap lines are visible in drawings. However, they are only used to help locate your dimensions and drawing detailing, so they do not plot with the drawing.

1. Click **Tools > Environment**. The Environment dialog box opens.
2. Under **Display**, select **Snap Lines**. The snap lines are visible in the drawing.

Draft Entity Snap Lines and Related Views

As you work with draft entity snap lines, consider the behavior of snap lines and snapped items as you relate and unrelate draft entities to views.

Note: The term background view refers to items belonging to the drawing but not belonging to any view, such as a free note.

- The view of a snapped item will not be automatically changed or related due to a manipulation of a draft entity.
- If a draft entity is related to view X, you can snap items to snap lines if they belong to view X or to the background view.
- If a draft entity is not related to a view (just on the background view), you can snap items to its snap lines regardless of the view to which they belong, including the background view.
- If a draft entity is not related to a view (just on the background view), you can relate the draft entity to the view as long as the items on a snap line offset from

that draft entity do not belong to a different (non-background) view. For example, you can relate draft entity to a view X if all the items snapped to the snap line belong to either the background view or view X.

- At any time, you can unrelate a view (changed back to the background view). All items snapped to its snap lines remain snapped.

There are no special considerations for draft entities in draft groups. Draft Groups will exhibit the same behavior.

Using Tables, Reports, and BOM Balloons

About Drawing Tables

A drawing table is a grid of rows and columns in which you can enter text. The text in a drawing table has complete text functionality; you can modify it by double clicking the cell and entering the text in a dialog box. You can enter dimension symbols and drawing labels as well, and the system updates them as you modify the model or drawing. You can include a drawing table in drawing formats, drawings, and layouts.

You can modify the line font, color, and width of a table grid the same way you would modify geometry.

Using Pro/REPORT with Tables

Pro/REPORT lets you designate certain cells as "repeat regions" that automatically display design data that you specify through special report parameters in the cell. If you define these regions in a report file, (.rep) they are read-only. If you define them in a drawing however, you can edit values shown in the drawing table and pass the edits back to the associated parts.

See About Creating Reports for more information on this subject.

Tip: Selecting Table Cells

You can select a table using pre-highlighting. When you move the pointer to any corner of the table, the entire table is pre-highlighted. You can then click to select the table.

To select a row or column using pre-highlighting move the pointer over an outer cell border. The row or column is pre-highlighted. Click to select.

To Create a Drawing Table

1. Click **Table** > **Insert** > **Table**. The **TABLE CREATE** menu appears.
2. Click one of the following options:
    - **Descending** or **Ascending**—Lets you choose the direction in which to create rows from the starting point.

- **Rightward** or **Leftward**—Lets you choose the direction in which to add columns from the starting point.
- **By Num Chars** or **By Length**—Lets you define the length of columns and rows using drawing units (such as inches or millimeters) or by specifying the number of characters as the unit.

3. Use the commands on the **GET POINT** menu to locate the starting corner of the table on the sheet. The starting corner depends on the choices you made in the preceding step.
   - **Pick Pnt**—Lets you select a point on the sheet and specify it as the starting corner.
   - **Vertex**—Lets you select a vertex and specify it as the starting corner.
   - **On Entity**—Lets you select an entity on the format and specify it as the starting corner.
   - **Rel Coords**—Lets you specify relative coordinate values for placing the starting corner by entering relative values for the X- and Y-axes.
   - **Abs Coords**—Lets you specify absolute coordinate values for placing the starting corner by entering absolute values for the X- and Y-axes.
4. If you have selected **By Num Chars** to define the length of the columns or rows follow the steps below:
   a. Locate the starting point of the table by clicking on the drawing. A scale of numbers appears.
   b. Click on the numbers to mark the width of each column. Click **Done** when finished. The scale reappears vertically.
   c. Click on the numbers to mark the height of each row. Click **Done** when finished.

   Alternatively, if you have selected **By Length** to define the length of the columns and rows follow the steps below:

   a. At the prompt, enter the width of the first column in drawing units, and press ENTER.
   b. Continue entering the width of additional columns until you have the number of columns you want. Press ENTER again without typing another column width value.
   c. At the prompt, enter the height of the first row in drawing units, and press ENTER.
   d. Continue entering values for the height of additional rows until you have the number of rows you want. Press ENTER again without typing another row height value.

   The table is displayed on the drawing sheet.

Formatting Tables

To Copy and Paste Cells or Cell Contents

To copy cell contents from one cell to another empty table cell:

1. Select the table cell from which to copy the contents.
2. Click **Edit > Copy**.
3. Select another empty table cell where you would like to paste the contents.
4. Click **Edit > Paste**.

   Note: Copying and pasting multiple table cells is not supported.

To copy tables from one point to another:

1. Select the table to copy.
2. Click **Edit > Copy**.
3. Click empty space to come out of Table selection.
4. Click **Edit > Paste**.
5. The clipboard window opens, showing the copied object. Click to select a transition point on the object in the clipboard. (The cursor will be attached at the selected point.)
6. Click the point in the drawing to paste the table. The table is copied at the selected point.

To Enter Text in a Table Cell

To add text to or edit text in a drawing table cell,

1. Double click the cell. The **Note Properties** dialog box opens
2. Use the dialog box to enter text or text symbols. Use the **Text Style** tab in the dialog box to set text height, slant angle etc.

   Use **Format > Wrap Text** to bring the text inside the boundaries of the cell.

Note: If the cell is part of a report repeat region, the insert parameters dialog box opens instead. If you want to enter a custom parameter in the repeat region, use the **Properties** command from the right mouse button shortcut menu to enter the text.

To Merge Cells

1. Use the Ctrl key to multiple select cells that you want to merge. Select the opposite corners of a range of cells to merge across rows and columns.
2. Click **Table > Merge Cells**. The cells are converted to one cell.

Restrictions When Merging Cells

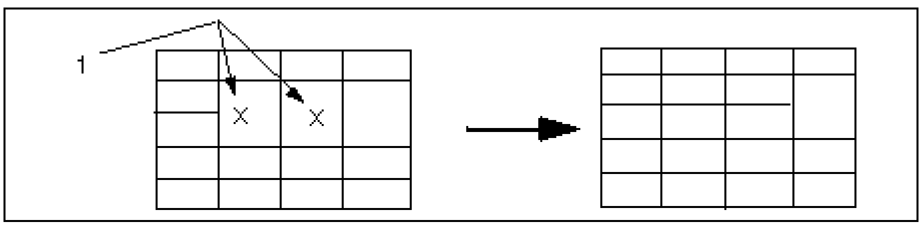
When merging cells, keep in mind the following restrictions:

- Only one cell in the range can contain text.
- The cell containing text must be positioned properly with respect to the origin of the table. For example, in a rightward, descending table, a cell with text must be at the left (upper) end of the range of cells you are going to merge, with the empty cell at the right (lower) end of the range.

To Unmerge Cells

Unmerging replaces merged cells with the original cell structure.

1. Select the cell or cells to unmerge.
2. Click **Table** > **Unmerge Cells.**



1. Pick here to get the resulting table.

To Change the Table Origin

1. Select the table.
2. Click **Table** > **Set Rotation Origin**. All the corners of the Table highlight.
3. Pick one corner to be the new origin.

To Rotate a Table 90 Degrees

1. Select the table.
2. Click **Table** > **Rotate.** Table rotates by 90 degrees counterclockwise about its origin. Each time that you select Table > Rotate, the table rotates an additional 90 degrees.

Alternately

1. Select the table.
2. From right mouse button shortcut menu, click **Rotate**.

To Blank or Display Cell Borders

You can hide selected cell lines within a table.

1. Select the entire table.

2. Click **Table** > **Line Display**.
3. In the Menu Manager, select an option, then select lines on the table:
   - **Blank**—Blanks the selected line segment.
   - **Unblank**—Redisplays the selected line segment in the table.
   - **Unblank All**—Redisplays all blanked lines in the table.

To Insert or Remove Rows or Columns

To Insert a Row or Column

1. Click **Table** > **Insert** > **Row** or **Column**.
2. Do one of the following:
   - To insert a row, select a horizontal line in the table. A new row is placed between the two rows that border the selected line.
   - To insert a column, select a vertical line in the table. A new column is placed between the two columns that border the selected line.

To Remove a Row or Column

1. Click a cell in the column or row you want to remove.
2. Click **Table > Select > Row** or **Column**. The row or column is selected.
3. Press **Del**. The column or row is deleted. (You can use **Undelete** on the right mouse button shortcut menu to abort the deletion, or click open space on the drawing to confirm it.)

To Resize Rows and Columns

You can set row height in units, or a number of characters as measurements of height.

1. Select the rows or columns to reset.
2. Click **Table** > **Height and Width**.
3. Use the dialog box to set the new dimensions for the selected rows or columns.

To Word Wrap Table Text

1. Select the table cell containing the text
2. From the right mouse button pop up menu, click **Wrap Text**. The text wraps to the column width.

Note: If you need to extend the row height, use the **Height and Width** command on the right mouse button shortcut menu.

To Justify Text

1. Select a table or a row or column from the table.
2. Click **Format** > **Text Style**. The **Properties** dialog box opens.
3. Click the **Text Style** tab. Select the justification style from the **Note / Dimension** field.
4. Click OK to finish.

To Move a Table

1. Click to select the table. Move the cursor over one of the handles on the outline, until the cursor changes to a four-way arrow.
2. Hold the left mouse button down, and drag the table to where you want it.
3. Release the left mouse button to place the table.

   Alternately, you can use **Edit** > **Move Special** to enter X and Y coordinates for the target location.

Note: To move a table to another sheet, select the table and use **Edit > Move Item to Sheet**.

To Delete a Table

1. Select the table to delete.
2. Click **Edit > Delete**.

To Save a Table

1. Select the table or a cell of a table to save.
2. Click **Table** > **Save Table** > **As Table File** or **As Text File**
3. Type a name for the table. The system saves it to a file in your current directory with a `.tbl` extension. It gives the filename a unique version number to prevent overwriting tables with the same name.

To Save a Table as Text

1. Select the table or a cell of a table to save.
2. Click **Table** > **Save Table** > **As Text File**. The **Save Drawing Table** dialog box opens.
3. Browse to the location where you want to save the table as text file.
4. Type a name for the file and click **Save**. Pro/ENGINEER saves the file in the specified location with the name `filename.txt.1`.

To Retrieve a Saved Table

1. Click **Table** > **Insert** > **Table from File**.
2. Select the name of the saved .tbl file in the file browser.
3. The table's outline appears with its upper-left corner attached to the pointer. Place the table by selecting the location for the upper-left corner.

To Save a Table as a CSV File

1. Select the table or a cell of a table to save.
2. Click **Table** > **Save Table** > **As CSV File**. The **Save Drawing Table** dialog box opens.

   Note: A CSV file is a Comma Separated Value file. You can open a CSV file in a spreadsheet application or a text editor.
3. Browse to the location where you want to save the table as a CSV file.
4. Type a name for the file and click **Save**. Pro/ENGINEER saves the file in the specified location with the name `filename.csv.1`.

Hole Charts

About Hole Charts

Hole charts organize and display detailed information for the various hole, datum point, and datum axis drawing items found within a specified drawing view. For example, a hole chart for feature holes assigns a name to each hole and lists the coordinate locations and diameter values. The actual holes are also labeled on the drawing view; numerically for ISO standards or alphanumerically for ASME standards.

Hole Chart main_view			
Hole No.	X	Y	Θ
1	0.79	5.97	0.33
2	9.18	6.14	0.52

You can create, delete, and update hole charts.

The table cells and content automatically populate when the view is selected and a general table location is chosen. You can customize the table display before creating the table, such as adding additional parameter columns or designating a sort order, or you can manually edit and format the table content after it is placed. Each drawing item type is displayed in a separate table, meaning holes are listed in one table, while datum points and axes are each listed in their own tables. You can have multiple hole charts within a drawing, but only one per view for each of the drawing item types.

When you create a hole chart pay close attention to the orientation of the coordinate system. In order for holes, datum points, and datum axes to display in a hole chart, you must reference a coplanar coordinate system. You may need to insert a new coordinate system to enable the objects for the hole chart.

Note: If you made changes to the holes, datum points, or datum axes in the model, or if you want to return any modified drawing values to the accurate model values, you should update the hole charts.

About Hole Notes

Pro/ENGINEER inserts hole notes in hole tables as offset notes by default using the axis of the selected hole as a reference. You can control the placement of these notes when you create hole tables. Hole notes created as offset notes retain their format and position even when you update a hole table. Hole notes are deleted if you delete the hole that they represent.

Note: Drawings containing hole tables created using the versions of Pro/ENGINEER earlier than Wildfire 4.0 are not updated automatically when you open them using the current version of Pro/ENGINEER. When you update such hole tables, all the previously created hole notes are deleted and are recreated as offset notes.

To Create a Generic Hole Chart

Make sure a coordinate system is coplanar with the hole, datum axis, or datum point drawing item you wish to display in a hole chart. If a coplanar coordinate system does not exist you should insert one in the 3D model.

1. Click **Tools** > **Hole Table.** The Menu Manager opens.
2. To create a hole chart using the default columns and settings, click **Create**. The **LIST TYPE** menu appears.
3. Define the type of hole chart to create:
    - **Holes**—Creates a hole chart table that lists the location of a hole in X and Y coordinates and lists its diameter.
    - **Datum Points**—Creates a table that lists the locations of datum points in X, Y, and Z coordinates.
    - **Datum Axes**—Creates a table that lists the locations of datum axes in X and Y coordinates.
4. Select a coordinate system reference for the drawing items to list in the chart. Make sure the coordinate system is coplanar with the holes, datum axes, or datum points you wish to display in a hole chart.
5. Select the location for the top left corner of the hole chart. The hole chart is displayed and populated with the appropriate data. You can move and relocate the hole chart as you would an ordinary table.

Note:

- Only drillable hole features can be listed in the hole chart.
- Only straight holes with their placement surface coplanar to the Hole coordinate system (XY plane) will show up when using the Holes command for hole chart list type.

To Create a Custom Hole Chart

You can add columns and format the hole chart display to better meet your specific detailing needs.

Make sure a coordinate system is coplanar with the hole, datum axis, or datum point drawing item you want to display in a hole chart. If a coplanar coordinate system does not exist you must insert one in the 3D model.

1. Click **Tools** > **Hole Table.** The Menu Manager appears.
2. Click **Setup** to format the additional columns and settings for the hole chart. The **LIST SETUP** menu appears. Select the desired command:

- **Num Decimals**—Sets the number of decimals places for the values displayed in the hole chart. The default setting is 3.
- **Param Column**—Defines additional columns for the hole chart. When you create a hole chart the default column format includes a table heading and a name column, as well as:
    - **Holes**—X- and Y- coordinate columns and a diameter column.
    - **Datum point**s—X-, Y-, and Z- coordinate columns.
    - **Datum axes**—X- and Y- coordinate columns.

  Note: When adding a new parameter column, the parameter type must be a feature parameter. You can view a listing of all the added columns by clicking **List**.
- **Label Size**—Defines the size of the text for hole notes on the drawing view. The default size is `-1`. However, you can type a new value greater than zero.
- **Label Position**—Defines the default position of the text for hole notes with reference to the selected hole axis. You can select either **Upper Right** or **Center** to position the text.

  Note: By default, Pro/ENGINEER creates hole notes as offset notes with reference to the selected hole axis.
- **Max Rows**—Defines the maximum number of rows in a hole chart or table.
- **Hole Naming**—Defines the method of naming holes. The default is set to the ASME standard (**Alphanumeric**). However, the ISO standard (**Numeric**) is available.

- **Sort Setup**—Modifies the protocol for sorting entries in the hole chart or table. By default, hole charts are sorted by the name column.

3. To create the hole chart, click **Create**. The **LIST TYPE** menu appears.
4. Define the type of hole chart to create:
    - **Holes**—Creates a hole chart table that lists the location of a hole in X- and Y- coordinates and lists its diameter.
    - **Datum Points**—Creates a table that lists the locations of datum points in X-, Y-, and Z- coordinates.
    - **Datum Axes**—Creates a table that lists the locations of datum axes in X- and Y- coordinates.
5. Select a coordinate system reference for the drawing items to list in the chart. Make sure the coordinate system is coplanar with the holes, datum axes, or datum points you wish to display in a hole chart.
6. Select the location for the top left corner of the hole chart. The hole chart is displayed and populated with the appropriate data. You can move and relocate the hole chart as you would an ordinary table.

Note:

- Only drillable hole features are listed in the hole chart.
- Only straight holes with their placement surface coplanar to the Hole coordinate system (XY plane) show up when you use the Holes command for the hole chart list type.

To Update a Hole Chart

If you made changes to the holes, datum points, or datum axes in the model, or if you want to return any modified hole chart values to the accurate model values, you should update the hole charts.

1. Click **Tools > Hole Table**. The **HOLE TABLE** menu appears in the menu manager.
2. Click **Update**. The hole charts update with the most recent model information.

Note: You can modify the hole chart table cells as you would a normal table—double click the cell or right-click the cell and click **Properties**.

Creating Reports

Using the Report File Type

You can create a new file dedicated to report information using the file type Report (.rpt).

Report mode menus are abbreviated versions of the Drawing mode menus. In a Report file, you can display data for an associated 3D model in table form just as it is in drawing mode. The system updates the information when you edit the model.

This mode provides all of the forward-associative abilities that Drawing mode offers. The only associative difference is that you cannot modify the model from a report file as you can from a drawing file.

About Creating Reports in Drawing Files

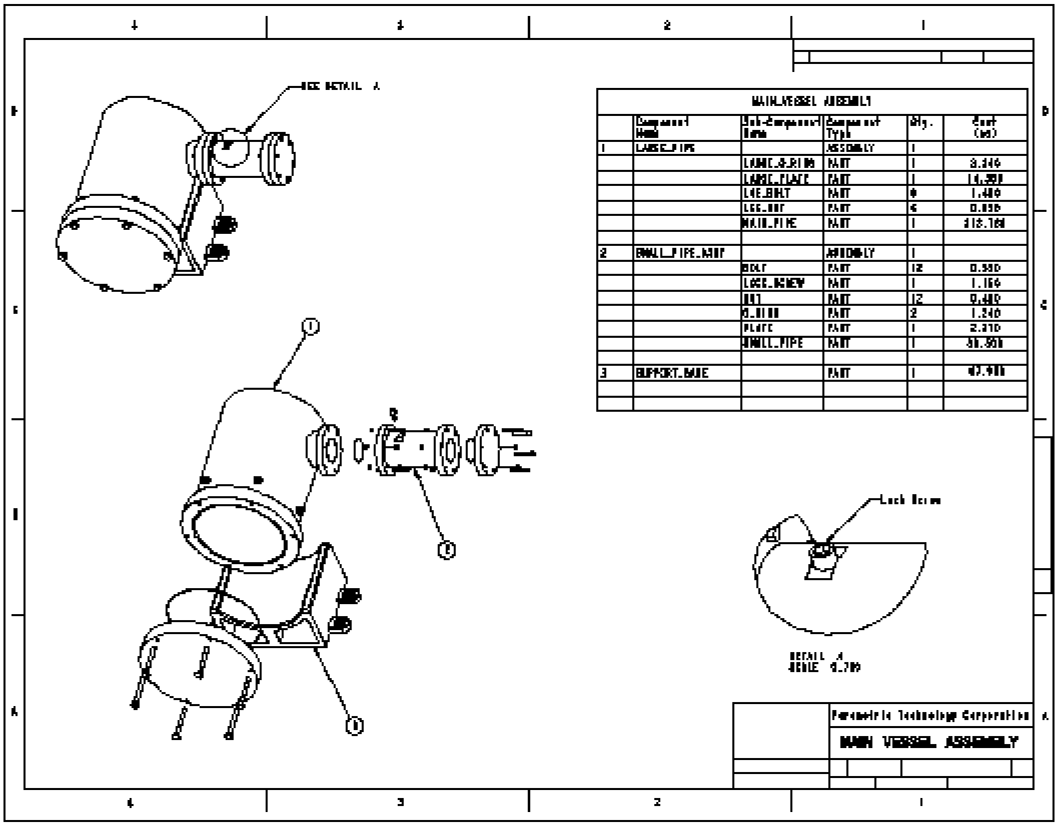
In Drawing mode, you can create dynamic, customized table reports. By defining *repeat regions* across table cells, you can add model data to a report which updates as the model changes, adding or subtracting rows or columns automatically as necessary.

You can also use the following techniques to manage the data that appears in a report:

- Add filters to eliminate specific types of data from appearing in reports, drawing tables, or layout tables.
- Search recursive or top-level assembly data for display.
- List duplicate occurrences of model data individually or as a group in a report, drawing table, or layout table.
- Directly link assembly component balloons to a customized BOM and automatically update them when you make assembly modifications.

You can generate several kinds of output using the report functions, such as family tables, associative reports, and graphical wirelists. A common example of a report is a customized Bill of Materials (BOM), such as the one shown in the next figure.

Report with Assembly Views



Working with Repeat Regions in Reports

Understanding Repeat Regions

Pro/Engineer dynamic report tables are based on the principle of "smart" table cells called repeat regions. Repeat regions are user-designated sections of a table that expand or contract to accommodate the amount of data that the associated model currently possesses.

The information they contain is determined by text-based *report symbols*, entered as text into each cell within the region. For example, if you have an assembly drawing with 20 parts, and you type `asm.mbr.name` into a cell in a repeat region, the table will expand to add one cell for each part name when you update the table.

Below is a very simple example of a table showing the parameters for a bill of materials. The top row is normal text entered into normal table cells. The bottom row contains report symbols, entered as text into a row of cells that has been designated as a repeat region.

INDEX	PART NAME	QUANTITY
rpt.index	asm.mbr.name	rpt.qty

The resulting updated report table might look like this:

INDEX	PART NAME	QUANTITY
1	COVER_FRONT	1
2	SHAFT	1
3	BUSHING	2
4	BEARING	4
5	COVER_BACK	1

You can enter report tables in any file type that supports them; such as drawings, diagrams, or report files. They function the same way in each.

To Define Repeat Regions

Use this procedure to define a simple repeat region.

1. Select a cell in the row you want to designate as a repeat region.
2. Click **Table > Select > Row**. The row is highlighted.
3. With the row still selected, click **Add Repeat Region** from the right mouse button shortcut menu. The row is designated as a repeat region.

Note: A region does not have to be one row, the selection can span two or more rows.

To Enter Text in a Table Cell

To add text to or edit text in a drawing table cell,

1. Double click the cell. The **Note Properties** dialog box opens
2. Use the dialog box to enter text or text symbols. Use the **Text Style** tab in the dialog box to set text height, slant angle etc.

   Use **Format > Wrap Text** to bring the text inside the boundaries of the cell.

Note: If the cell is part of a report repeat region, the insert parameters dialog box opens instead. If you want to enter a custom parameter in the repeat region, use the **Properties** command from the right mouse button shortcut menu to enter the text.

To Assign a Different Model to a Quantity Column

Use the **Column Model/Rep** command in the **TBL REGIONS** menu to display a different model, instance of a model, or simplified representation to control a quantity column.

1. Click **Table** > **Repeat Region**. The **TBL REGIONS** menu appears.

2. Click **Column Model/Rep**.
3. Select a column with the report parameter `rpt.qty`. The **Open** dialog box opens and displays all files in your current working directory.
4. Select a new model and click **Open**.
5. Click **Confirm** on the **TBL REGIONS** menu. The new model is assigned to the quantity column.

Note: BOM balloons are not supported for repeat regions with multiple models.

To Assign a Different Model to a Region

Use the **Model / Rep** command to display a different model or simplified representation to control a region. If you change the model associated with the repeat region, the current model appears as the default choice.

1. Click **Table > Repeat Region**. The Menu Manager opens.
2. Click **TBL REGIONS** > **Model / Rep**.
3. Select a region. A File browser opens to the working directory. Select a new file name.
4. Click **Confirm** to delete BOM Balloons and model-dependent region items.

To Remove a Repeat Region from a Table

1. Click **Table** > **Repeat Region**. The Menu Manager opens.
2. Click **TBL REGIONS** > **Remove**. You are prompted to pick the region to remove.
3. Select a region. Click **Yes** in the confirmation prompt.

About Nesting Repeat Regions

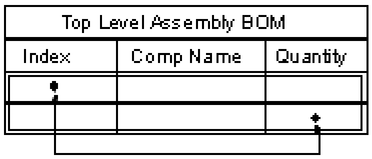
Nested repeat regions are repeat regions entirely contained by another repeat region. Repeat regions cannot overlap. When you nest them, you can define a double loop, such as one that searches through all subassemblies of an assembly and through all parts in each subassembly.

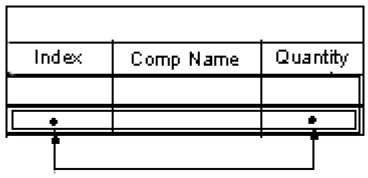
To Create Nested Repeat Regions

To create nested repeat regions, you define an initial *outer* region by selecting the cells to define the boundary, and adding another *inner* repeat region by selecting cells that are completely enclosed by the outer region.

You can remove an outer (enclosing) repeat region without removing the enclosed repeat region.

Examples: Nested Repeat Regions





The first table shows the definition points for the original repeat region. The second table shows the pick points for the nested region.

The following figure shows a table with a nested repeat region. The regions do not overlap. The attributes for both the inner and the outer repeat regions have been set using **No Duplicates** and **Flat**.

Table with a Nested Repeat Region

ITEM NAME	SUB-ITEM NAME	QTY
&asm.mbr.name		&rpt.qty
	&asm.mbr.name	&rpt.qty

When the model assembly "Main Vessel" is added to the report, the resulting table appears as shown in the next figure.

Model Assembly "Main Vessel" Added

ITEM NAME	SUB-ITEM NAME	QTY
LARGE_PIPE		1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
	MAIN_PIPE	1
SMALL_PIPE_ASSY		1
	BOLT	12

	LOCK_SCREW	1
	NUT	12
	O_RING	2
	PLATE	1
	SMALL_PLATE	1
SUPPORT_BASE		1

Entries in the outer repeat region are listed under the header "item name," while entries in the inner repeat region are listed under the header "sub-item name." Since item "Support Base" is a part and has no subitems, the row beneath its entry is blank. If you set the minimum number of repeats for the inner repeat region to 0, this prevents blank lines if there is no information at that level.

In the following figure, the attributes of the inner repeat region have been changed to **Duplicates** and **Flat**. Notice that the subitems have multiple entries where possible, and no value for quantity is listed for them.

Attributes of Inner Repeat Region Changed to Duplicates and Flat

ITEM NAME	SUB-ITEM NAME	QTY
LARGE_PIPE		1
	MAIN_PIPE	
	LARGE_O_RING	
	LARGE_PLATE	
	LGE_BOLT	
	LGE_BOLT	
	LGE_BOLT	
	LGE_BOLT	
	LGE_BOLT	
	LGE_NUT	
	LGE_NUT	
	LGE_NUT	
	LGE_NUT	
	LGE_NUT	
	LGE_NUT	
SMALL_PIPE_ASSY	SMALL_PIPE	1
	PLATE	
	O_RING	
	LOCK_SCREW	
	BOLT	
	BOLT	
	BOLT	
	BOLT	
	O_RING	
SUPPORT_BASE		1

Note: Table is broken for display purposes so as not to show all patterned bolts and nuts.

In the next figure, the attributes for the outer repeat region have been changed to **No Duplicates** and **Recursive**. The attributes for the inner repeat region have been changed to **No Dup/Level** and **Recursive**. To display the **No Dup/Level** attribute, the part "O Ring" was added to the assembly "Large Pipe" for this example only. The "O Ring" part now exists in two levels of the "Main Vessel" assembly ("Small Pipe" assembly and "Large Pipe" assembly).

Attributes of Outer Repeat Region Changed to No Dup/Level and Recursive

MAIN_VESSEL		1
	MAIN_VESSEL	1
	LARGE_PIPE	1
	MAIN_PIPE	1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
	O_RING	1
	SUPPORT_BASE	1
	SMALL_PIPE_ASSY	1
	SMALL_PIPE	1
	PLATE	1
	O_RING	2
	LOCK_SCREW	1
	BOLT	12
	NUT	12
NUT		12
	NUT	1

O Ring is listed twice in the table because it exists at two levels in the "Main Vessel" assembly, but is only listed once per level.

Table is broken so as to show only the area of No Dup/Level.

To Switch Between Symbols and Text

To switch between symbol and text in a repeat region table,

1. Click **Table > Repeat Region**. The menu manager opens.
2. Click **Switch Syms** on the menu manager. All repeat regions revert to their symbol entries. Click **Switch Syms** again to toggle the display back to text.

To Update a Repeat Region

1. Click **Table** > **Repeat Region**. The Menu Manager opens.
2. Click **Update Tables.**.

To Show Family Tables with Repeat Regions

Use the Two-D (two-directional) repeat region type to display the contents of a component's family table.

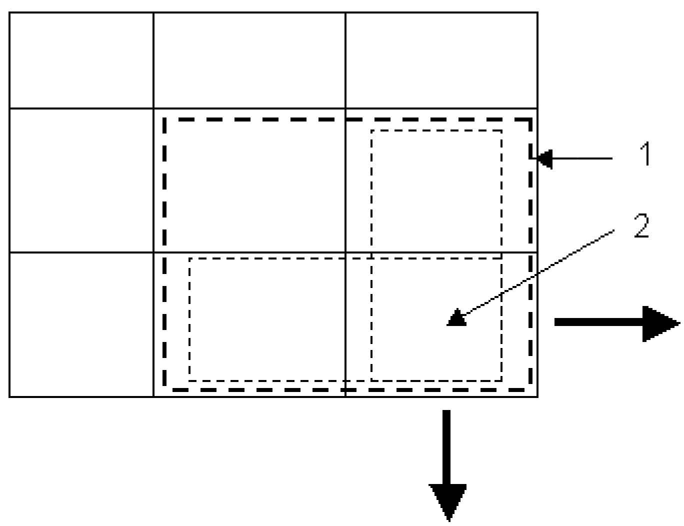
To Define a Two-Directional Repeat Region for a Family Table

Use two directional repeat regions to read out the contents of family tables. This procedure shows you how to define the repeat regions and enter the basic parameters for a family table report. It assumes you have created the table cells already.

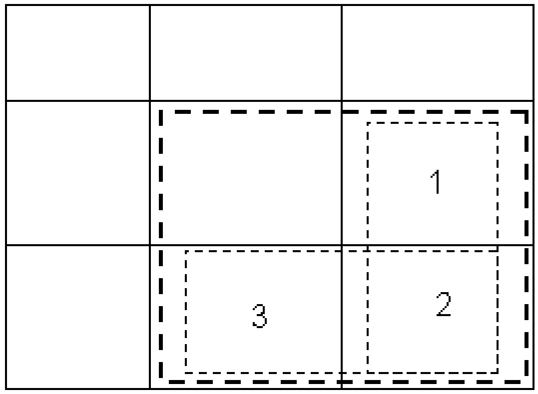
1. Click **Table** > **Repeat Region**. The Menu Manager opens
2. Click **Add**. The **Region Type** menu opens to **Simple** or **Two-D**.
3. Click **Two-D**. You are prompted *Locate corners of the outer boundary of the two-dimensional region*.
4. Select two points to define the outer repeat region. (The shape of this region is determined by how many cells in either direction you want to include in the subregions.) You are prompted *Select a cell to set the upper border of the row & column subregions.*

   This is the cell from which the table will expand outward and downward. When you click this cell, subregion cells are added in both directions *away from the cell,* to the extent of the first region, so you can use them to enter additional family table report symbols.

   In the example below 1) is the region you define first, the all-enclosing region. 2) is where you would pick to define the upper border of the subregions. The dark arrows show the direction in which the table will expand.



5. When you have defined the regions, click the middle mouse button twice to exit the routine. You will now enter the report symbols for the family table.



6. Using the diagram above as an example, double click cell 2. The **Report Symbol** dialog box opens.

7. Add the symbol `fam.inst.param.value`.

8. The report symbol appears in the cell. Now you will define the left side column for the instance name, and the top row for the parameter name..

9. Double click in cell 1, and enter the symbol `fam.inst.param.name`, Repeat the process for cell 3, and enter `fam.inst.name`.

10. When the parameters are entered, click **Tables > Repeat Region > Update Tables**. The family table is displayed in table form.

Using Assembly Simplified Representations

You can use assembly simplified representations in a drawing to control a repeat region of the model, and you can also show BOM balloons on the representation. When you use assembly simplified representations to control a repeat region, simplified representations of the same assembly behave entirely as different models.

A repeat region that reports on a simplified representation reports on that simplified representation itself, as opposed to the assembly. This includes the following:

- If you substitute a component with anything other than a simplified representation of itself, the assembly tree itself is different.
- The system does not show excluded components, even if they are in memory.
- Parameter values set by assembly relations (except mass properties) reflect the represented states of the models within the assembly.
- A repeat region is automatically associated with the current model or simplified representation.
    - If the current simplified representation changes, the repeat region is always associated with the original simplified representation when it was created or loaded.
    - If the current model is an assembly, the repeat region is associated with that assembly model.

   - If the current model is a simplified representation, it is associated with the simplified representation.

- If you substitute a part or subassembly with a simplified representation, the system still displays it in the correct location in the assembly tree.

  For example, if you substitute a part in the third level of the assembly with a simplified representation of the part, the report displays the substitution in the third level. This applies only to substitutions by simplified representations.

You can create a table or Bill of Materials with piping information in a drawing, so that the system updates the table automatically to reflect changes in the piping assembly. You can also customize drawing tables to reflect practices specific to your particular company.

Naming Conventions for Simplified Representations in a Repeat Region

The naming conventions for simplified representations in a repeat region are as follows:

- If you substitute a component with a simplified representation, the system gives it the original part name.
- If you substitute a component with a family table instance, the system gives it the instance name.
- If you substitute a component with an envelope, the system gives it the envelope component name.
- If you substitute a component with a geometry snapshot, the system gives it the original part name.
- If you substitute a component with an interchange group, the system gives it the interchange part or assembly (and part) names.

Using Parameter Values in Reports

To Enter Report Parameters into a Repeat Region

If table repeat regions have been defined, you can type report parameter strings into them manually, or assemble them from a hierarchical menu.

To enter text from the keyboard, select the target cell and click **Properties** from the right mouse button shortcut menu. Use the **Note Properties** dialog box to enter text.

Click the **Report Symbol** button in the dialog box to enter parameter elements from a hierarchical menu. The first level appears in abbreviations, **asm**, (assembly) **dgm**, (diagram) **fam**, (family) **harn**, (harness) and **rpt** (report).

1. Click an element for the first part of the parameter. The menu changes to a selection of parameters valid for that element.

2. Click an element for the next part of the parameter. Whenever an element requires another element, the element name is followed by ellipses (...). When the parameter is complete, it is entered into the table cell.

Note: If you double click a repeat region cell, the **Report Symbol** dialog box opens automatically.

To Use Report Parameters in Multi-Model Drawings

To create tables that display data for different models, you must first change the active model, and then create a new repeat region and type the parameters. Otherwise, the system takes data from the last active model.

Repeat regions always remember the models from which the system is extracting their values. A report parameter that you type in a repeat region on the drawing while a certain model is active reads values from that model only. The system extracts report parameter values only when you include the parameters within a repeat region. For example, if you type a set of report parameters in a table without a repeat region, the system does not read parameters from the active model until they are enclosed by a repeat region.

If the report parameter symbol "&asm.name" resides inside a table but outside of a repeat region, it can have a postfix number (for example, "&asm.name:3") to specify the assembly in a multi-model drawing from which it is reading its information. These postfix numbers always begin counting at "0."

Modifying User-Defined Report Parameter Values

When you modify the values of user-defined parameters within the Drawing, Layout, and Diagram modes, the changes are reflected in the model database. However, because Report mode is read only, any changes you make to user-defined parameters in Report mode do not reflect back to the model database; with the exception of the following parameters:

- `&asm.mbr.param.value`
- `&mdl.param.value`

If you modify the values of these two user-defined report parameters within a repeat region or note, the changes appear throughout the database.

To Modify a User-Defined Report Parameter

1. Click **Edit** > **Value**.
2. Select the user-defined parameter to modify.
3. Type the new value for the parameter. If the report parameter is one the three exceptions, the changes update throughout the database. Otherwise, modification of the model is not allowed from Report mode.

Note: You can not modify a user defined parameter specified by `&asm.mbr.parameter_name` in REPORT mode.

Cabling Component Parameters

About Cabling Component Parameters

You can include cabling parameters in drawing and report tables using the "asm.mbr.cbprm.User Defined" parameter symbol in a repeat region. This parameter symbol displays any user-defined parameters used in cabling components. If you use the parameter symbol "asm.mbr.cbprms.Name/Value" in a repeat region, any pre-defined cabling parameter names and values (such as wire type units, name, and so on) appear.

When you include any of these parameter symbols in a repeat region, the commands **Cable Info** and **No Cbl Info** appear in the REGION ATTR menu. You must choose **Cable Info** to display any cabling information in the selected region. The **No Cbl Info** command displays only non-cabling assembly information in the table. If you choose **Cable Info** for a repeat region, the table expands to include all spool names, tie wraps, and markers that exist on the model when "asm.mbr.name" or "asm.mbr.type" parameter symbols are used in the repeat region. You cannot use repeat region attributes **No Duplicates** or **No Dup/Level** when you use the "asm.mbr.cblprms.name" parameter symbol in that region.

Creating Pro/REPORT Tables in Flat Harnessed Drawings

About Creating Report Tables in Flat Harness Drawings

Using the top-level report symbol, "mbr," you can create Pro/REPORT tables in a flat harnessed drawing for a single assembly component/connector. When you enter an "mbr" symbol such as "mbr.name" into a repeat region, the system arbitrarily selects a component from the default drawing model and reports only on that component, producing the relevant subset of the table that would be produced if you had typed the "asm.mbr" symbol instead.

To change the component that is driving the repeat region, you must display it in the view. When you choose **Model / Rep** from the **TBL REGIONS** menu and select a region driven by the symbols beginning with "mbr," the system prompts you to select a component in a view. When you select a component, that component then drives the repeat region.

Showing Terminators in Report Tables

About Showing Terminators in Report Tables

You can show terminators in report tables for cable assemblies that have connectors with terminator parameters by using the report symbols "&asm.mbr.name" and "&asm.mbr.type." To show the terminators, set the drawing setup file option "show_cbl_term_in_region" to "yes" (the attribute **Cable Info** must be set for the repeat region). When creating new drawings, the default value is "yes." For existing drawings, the default value is "no."

Obtaining a Summation

To Get a Summation of Parameter Values of a Repeat Region

1. Click **Table** > **Repeat Region**.
2. In the Menu Manager, click **Summation**.
3. Select a region in the drawing; then choose **TBL SUM** > **Add**.
4. Select a report parameter in the repeat region.
5. Type a parameter name. You must select a table cell in which to place the parameter. The system adds a new parameter to the drawing whose value is the summation of the values of the selected report parameter.

   Note: The parameter is like any other parameter that is accessible using **Parameters** in the DWG SET UP menu, except that you cannot modify or delete it.

6. When the system prompts you to update the value, choose **TBL REGIONS** > **Update Tables**. The system updates the table and the makes the summation parameter visible.

   Notes: When you retrieve a saved table containing a summation parameter into a drawing, the system checks whether the drawing already has a parameter of that name. If it cannot find one, it adds one for you. However, if it does find a parameter by that name, you must enter a name to replace it in the table. It then uses that name for the summation parameter. If you do not enter a name, it retrieves the table, but ignores the parameter.

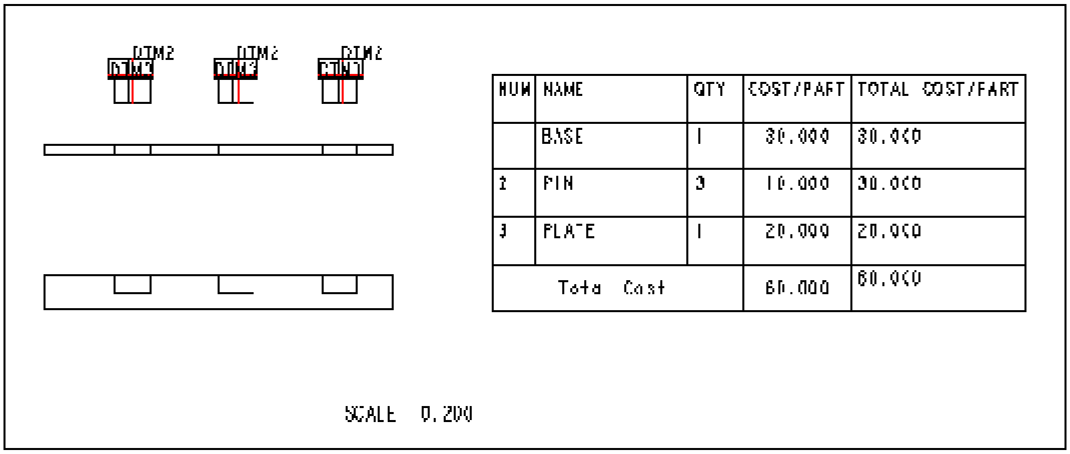
Example: Summation Parameters

To create the following example, create a table for the exploded assembly, and obtain a summation of the parameter symbol value "asm.mbr.cost" in the fourth column.

At the system prompt, type the summation parameter name as cost_sum. From the keyboard, type &cost_sum in the second cell in the last row, and type Total Cost in the first cell in the last row. After you update the table, the summation appears as 60.000. To create the "total cost/part" column, add the repeat region relation total_cost_per_part = rpt_qty * asm_mbr_cost.

Note: For quantities greater than one for any particular part, you must have a "cost/part" column and a "total cost/part" column, as shown in this example.

Example of Obtaining a Summation



Formatting Report Tables

About Paginating Report Tables

You can paginate report tables that are on the same drawing sheet, and you can break a table into arbitrarily placed segments on the same sheet. As you add more information to the table, the system flows the information into each segment and adds more sheets as necessary, copying the table format (if there is one). It chooses the segments to create the new sheet as follows:

- The system copies the segment definitions from the format if the table was copied from a format, the table on the format was paginated on two sheets, and the drawing table currently has only one sheet.
- Otherwise, it copies segment definitions from what was previously the last sheet of the table.

To Break Report Tables on the Same Sheet

1. Select a table.
2. Click **Table** > **Paginate**. The menu manager opens.
3. Click **Set Extent** and select a row where the table will break. The remainder of the table is removed.
4. Click **Add Segment** and select the location to begin the new segment on the current sheet. You can click a point or use one of the location options on the **GET POINT** menu. You are prompted to click an extent for the segment.
5. Locate the lower extent of the new segment. The new segment is placed between the extents.

To change the pagination, choose **Clear Extent** to clear all pagination information from the table. The system removes all segments other than the first, appends their contents to the first segment, erases the table, and draws it in its unpaginated state. Repeat Steps 2 through 4.

Repeat Region Formatting Options

About Formatting Repeat Regions by Attribute

You can use the **Table** > **Repeat Region** > **Attributes** menu to format repeat region behavior in the following ways.

- **Duplicates**—Lists all occurrences of a parameter in individual rows in a repeat region. This command sorts any data from the "&asm.mbr.name" display by feature number.
- **No Duplicates**—Lists all occurrences of a parameter in one row, along with a quantity number if you use the parameter "&rpt.qty" in the region.
- **No Dup/Level**—For a selected level of the region, lists duplicate occurrences of a parameter singly, along with a quantity if you use the parameter symbol "&rpt.qty" in the region. This command allows you to list duplicates only if the object resides in different levels of the assembly. There are no duplicates per level in the assembly. It sorts by parameter value any data generated from the "&asm.mbr.name" parameter symbol display.
- **Recursive**—Searches all levels of data for parameters.
- **Flat**—Searches only the top level of data for parameters.
- **Min Repeats**—Sets the minimum number of repetitions for a repeat region. The default minimum is "1." The system leaves extra rows blank. If the minimum is set to "0," it avoids blank lines caused by the lack of data.
- **Start Index**—Begins the index numbering of a repeat region (the value of "&rpt.index") where the index numbering of another repeat region ended. You cannot assign this attribute to nested repeat regions.
- **No Start Idx**—Begins the index numbering of a repeat region at 1.
- **Cable Info**—Displays the cable parameter information in tables containing appropriate parameter symbols.
- **No Cbl Info**—Displays only assembly parameter information in tables (that is, no cabling information appears).
- **Bln By Part**—When you suppress or replace a component to which a BOM balloon is attached, this command reattaches the balloon to another placement of the same part.
- **Bln By Comp**—Specifies that simple BOM balloons reattach themselves to whatever component replaced the one that originally owned the BOM balloon.

  Note: The *template* cell drives the number of digits of a quantity. In a two-directional repeat region, modifying the number of digits in the first column affects every column.

Example: Controlling Attributes

The figure below shows a repeat region which contains the system parameter symbols "&rpt.index," "&asm.mbr.name," and "&rpt.qty"

The default repeat region attributes of **Duplicates** and **Flat** are used.

Index	Component Name	Quantity
&rpt.index	&asm.mbr.name	&rpt.qty

The resulting table looks like the one in the following figure. The index numbers appear in the left column and the names of the members of the assembly appear in the middle column. No quantity appears since the attribute **Duplicates** is being used and the quantity is always "1."

Index	Component Name	Quantity
1	LARGE_PIPE	
2	SMALL_PIPE_ASSY	
3	SUPPORT_BASE	

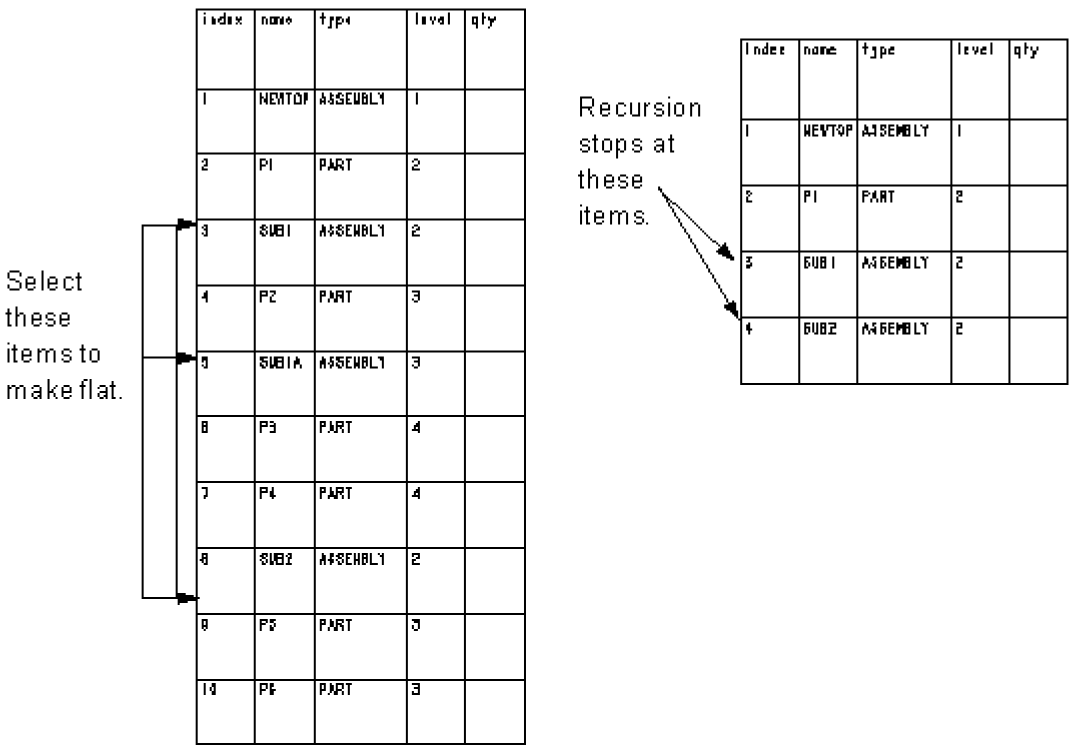
About Setting Recursive or Flat Items in a Repeat Region

You can select individual items in a repeat region to make them *recursive*—displaying items within the assembly—or *flat*—not displaying items within the assembly. When you set an item as flat or recursive, the system *marks* it until you choose **Default** or **Default All** to change it. Items that you set individually remember their setting even if you change the default attribute of the region from **Flat** to **Recursive** or vice versa.

To Set an Item as Flat

1. Click **Table** > **Repeat Region**. The **TBL REGIONS** menu opens on the menu manager.
2. Click **Flat/Rec Item**.
3. Select a repeat region.
4. Select **Flat** from the **RECUR ITEM** menu manager.
5. Select a record in the table. Use pick box selection to select more than one. Be sure that the selection box encloses the outer boundary of the region on both sides.
6. Click OK from the **Select** dialog.

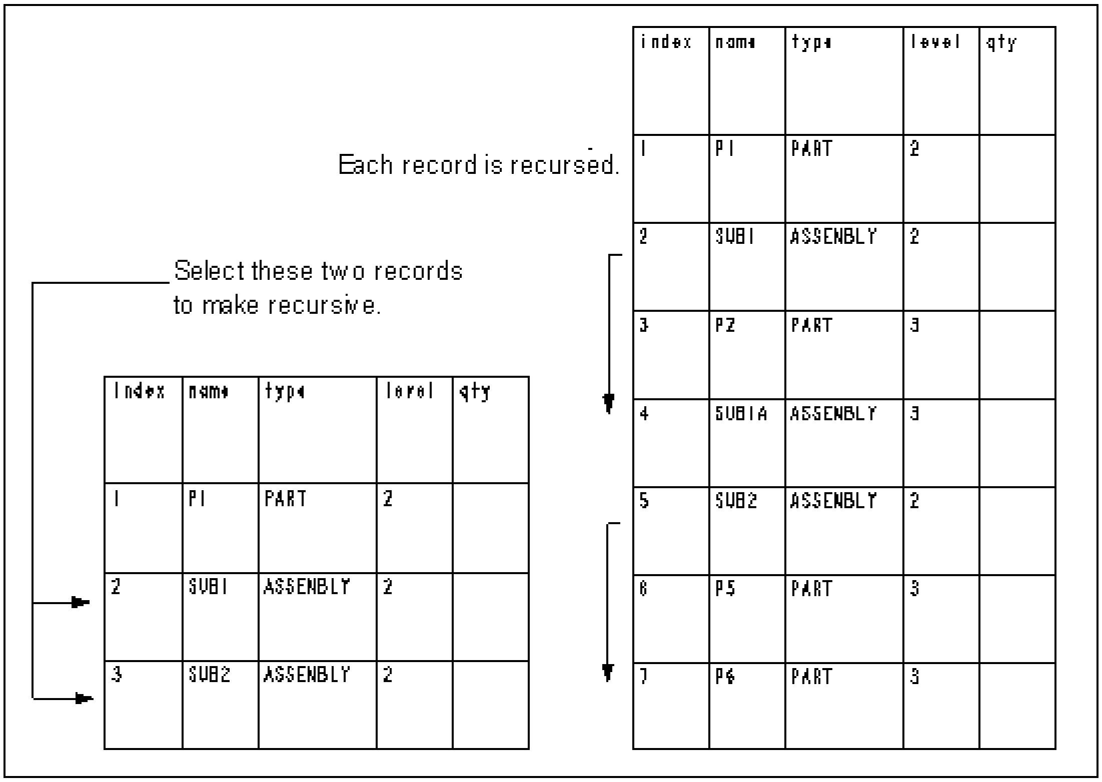
Example: Setting an Item as Flat



To Set an Item as Recursive

1. Click **Table** > **Repeat Region**. The **TBL REGIONS** menu opens on the menu manager.
2. Click **Flat/Rec Item**. Select a repeat region. The **RECUR ITEM** menu opens.
3. Select **Recursive**.
4. Select a record in the table. Use the pick box selection to select more than one. Be sure that the selection box encloses the outer boundary of the region on both sides.
5. Select **OK** from the **Select** dialog. The system updates the repeat region to display the new order. The recursion starts one level from the item that you selected.

Example: Setting an Item as Recursive



Creating Header and Footer Titles

About Creating Header and Footer Titles

When working with tables containing multiple segments, you can place a title at the top of each segment (as a header) or at the bottom of each segment (as a footer).

When creating titles, keep in mind the following rules:

- Two-directional repeat regions cannot have titles.
- For single-row headers and footers, you must select the row twice for the first and second rows.
- A region can have only one header and one footer. However, since a table can have more than one paginated region, a table can have more than one header and more than one footer.
- When selecting rows to be used in a title, you cannot choose a row that is part of a repeat region.
- Two titles cannot intersect each other. That is, when creating a new title, you cannot select a portion (one or two rows) of an existing title to be used in the new title. However, you can define the same rows to be both the header and the footer of the same region or to be titles of a number of regions.

- You cannot create a header or footer title in a row that has cells merged in a column.
- The system inserts a header at the top of a segment, and inserts a footer at the bottom of a segment, regardless of the direction of the table's growth (that is, ascending or descending).

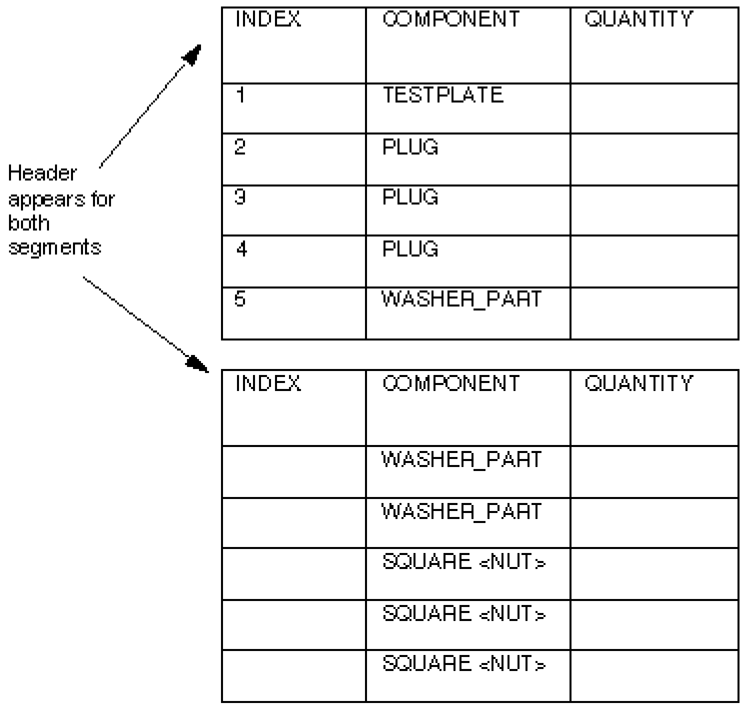
To Add a Header or Footer Title to a Table

1. Click **Table** > **Paginate**. The Menu Manager opens.
2. Select a table that contains at least one repeat region.
3. Click **Add Title.**
4. From the **REGION TITLE** menu, choose **Header** or **Footer**.
5. Select the first row (one that is *not* in a repeat region) to be used as the header or footer title of the selected region.
6. Select the second row. The segments of the table appear with the specified header or footer title.

Example: Header Title with a Descending Repeat Region

The next figure illustrates how to use a header title in a table with a descending repeat region.

Example of Table Segments with Header Title in a Table with a Descending Repeat Region



Specifying Indentation

About Specifying Indentation

Using the **Indentation** command in the TBL REGIONS menu, you can specify an indentation for report data in recursive repeat regions. The cell to which you apply the indentation must be in a repeat region that has **Recursive** specified as one of its attributes, and should contain a recursive report parameter symbol (such as "asm.mbr.name"), although this is not a requirement. If you select a cell that does not contain a recursive parameter, the system informs you, but it indents the data if you specify a value.

Pro/ENGINEER expresses indentation for a repeat region cell using a number between 0 and 100. The number indicates the value in characters (at the default text size) by which that data is indented in a recursive repeat region. Pro/ENGINEER increments the indentation by the specified amount at each new level in the table.

To Specify Indentation for a Repeat Region Cell

1. Click **Table** > **Repeat Region** The TBL REGIONS menu opens on the menu manager.
2. Click **TBL REGIONS** > **Indentation**.

3. Select the cell in a repeat region template for which you want to specify the indentation.

4. Type a value for the indentation.

5. Click **Update Tables** to redisplay the table with indentation.

Note: If you specify the indentation for a cell in an existing table, you must choose **TBL REGIONS** > **Update Tables** before you can view the result.

Example: Repeat Region Cells Using Indents

Indentation has been specified for the second column, containing values of "asm.mbr.name," and the third column, containing values of "asm.mbr.type." For each cell, the indentation was specified as 5. Note how this indentation is incremented at each recursive level.

Indented Information in a Recursive Repeat Region

LEVEL	NAME	TYPE
1	MAIN_VESSEL	ASSEMBLY
2	LARGE_PIPE	ASSEMBLY
3	MAIN_PIPE	PART
3	LARGE_O_RING	PART
3	LARGE_PLATE	PART
3	LGE_BOLT	PART
3	LGE_BOLT	PART
3	LGE_NUT	PART
2	SUPPORT_BASE	PART
2	SMALL_PIPE_ASSY	ASSEMBLY
3	SMALL_PIPE	PART
3	PLATE	PART
3	O_RING	PART
3	LOCK_SCREW	PART
3	BOLT	PART
3	BOLT	PART
3	BOLT	PART
3	O_RING	PART

Note: table broken for display purposes so as not to show all patterned bolts and nuts

Using Filters in Reports

About Adding Filters

Using the **By Rule** command on the **FILTER REG** menu, you can remove multiple items that match a specified pattern. You can use filters in the following forms to further specify the information you want displayed:

- <*symbol*>—Any parameter that is valid in a repeat region
- <*comparison operator*>—Any of the operators <, >, <=, =>, ==, and !=
- <*literal value*>—Any integer, floating point, or string value

Filters exclude from a repeat region any record that uses one of the filtered parameters and whose value does not match the constraint defined by the filter. Pro/ENGINEER omits the entire record from the table, and not just the parameter affected by the filter. For instance, the filter `&asm.mbr.type==part` in a repeat region omits all records of objects other than parts from the report.

Note: If you filter in a two-directional repeat region, it only removes the particular cell, not the entire record.

A filter such as `&asm.mbr.material!=steel` excludes all records of assembly members with a material parameter value (user-defined) of "steel" from the report.

You can also enter filters that work for multiple values, such as `&asm.mbr.name==part_a,part_b,part_j`, which would exclude from the report all records of assembly members other than "part_a.prt," "part_b.prt," and "part_j.prt." A line that defines a filter can contain up to 80 characters. When creating a filter with multiple acceptable values such as `&asm.mbr.name==part_a,part_b,part_j`, you can only use the operators "==" and "!=".

When using filters with "==" and "!=" operators in a repeat region, keep in mind the following points:

- If you add filters with the operator "==" to more than one line in a repeat region, all entries are blanked and the table appears to be empty.
- For "==" operators with more than one value, the values are linked by "or." For example, for the filter `&asm.mbr.name==part_1,part_2,part_3`, the repeat region excludes all objects other than those parts having the name "part_1," "part_2," or "part_3."
- For "!=" operators with more than one value, the values are linked by "and." For example, for the filter `&asm.mbr.name!=part_a,part_b,part_c`, the repeat region excludes those parts having the name "part_a," "part_b," and "part_c."

Use the commands of the **Filter Reg** submenu to add, edit, or clear filter statements associated with a repeat region.

You can use the **Filter** command for records that represent flexible components, family table generic components, bulk items, and included items within a repeat region.

To Add a Filter to a Repeat Region

1. Click **Table** > **Repeat Region**. The Menu Manager opens.
2. Click **TBL REGIONS** > **Filters**.
3. Select the repeat region to which you want to add a filter.
4. Click **FILTER REG** > **Add**. Type the filter expression.
5. Click **Done** to add the filter. Existing repeat regions regenerate to accommodate the filter.

Examples: Using No Dup/Level and Recursive Attributes

The attributes for the inner repeat region are specified as **No Dup/Level** and **Flat**.

Attributes for Outer Repeat Region Specified as No Dup/Level and Recursive

ITEM NAME	SUB-ITEM NAME	QTY
MAIN_VESSEL		1
	LARGE_PIPE	1
	SUPPORT_BASE	1
	SMALL_PIPE_ASSY	1
LARGE_PIPE		1
	MAIN_PIPE	1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
MAIN_PIPE		1
LARGE_O_RING		1
LARGE_PLATE		1
LGE_BOLT		6
LGE_NUT		6
SUPPORT_BASE		1
SMALL_PIPE_ASSY		1
	SMALL_PIPE	1
	PLATE	1
	O_RING	2
	LOCK_SCREW	1
	BOLT	12
	NUT	12
SMALL_PIPE		1
PLATE		1
O_RING		2
LOCK_SCREW		1
BOLT		12
NUT		12

When the filter &rpt.level>1 is added to the outer repeat region, the resulting table looks like the example in the next figure. The row containing "Main Vessel," the first level of the assembly, is removed. Only those members with a level value higher than 1 remain.

Filter "&rpt.level>1" Added to Outer Repeat Region

ITEM NAME	SUB-ITEM NAME	QTY
MAIN_VESSEL		1
	LARGE_PIPE	1
	SUPPORT_BASE	1
	SMALL_PIPE_ASSY	1
LARGE_PIPE		1
	MAIN_PIPE	1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
MAIN_PIPE		1
LARGE_O_RING		1
LARGE_PLATE		1
LGE_BOLT		6
LGE_NUT		6
SUPPORT_BASE		1
SMALL_PIPE_ASSY		1
	SMALL_PIPE	1
	PLATE	1
	O_RING	2
	LOCK_SCREW	1
	BOLT	12
	NUT	12
SMALL_PIPE		1
PLATE		1
O_RING		2
LOCK_SCREW		1
BOLT		12
NUT		12

Moving a Filter to Another Repeat Region

In the following figure, the filter asm.mbr.type==assembly is added to the inside repeat region, so no parts are listed there.

Filter "asm.mbr.type==assembly" Added to Inside Repeat Region

ITEM NAME	SUB-ITEM NAME	QTY
LARGE_PIPE		1
MAIN_PIPE		1
LARGE_O_RING		1
LARGE_PLATE		1
LGE_BOLT		6
LGE_NUT		6
SUPPORT_BASE		1
SMALL_PIPE_ASSY		1
SMALL_PIPE		1
PLATE		1
O_RING		2
LOCK_SCREW		1
BOLT		12
NUT		12

In the next figure, that filter is cleared from the inside repeat region and added to the outside repeat region. Again, all parts have been omitted from the region.

Filter Cleared from Inside Repeat Region and Added to Outside.

ITEM NAME	SUB-ITEM NAME	QTY
LARGE_PIPE		1
	MAIN_PIPE	1
	LARGE_O_RING	1
	LARGE_PLATE	1
	LGE_BOLT	6
	LGE_NUT	6
SMALL_PIPE ASSY		1
	SMALL_PIPE	1
	PLATE	1
	O_RING	2
	LOCK_SCREW	1
	BOLT	12
	NUT	12

Using Wildcard and Backslash Characters in Filters

About Using Wildcard and Backslash Characters in Filters

You can use wildcard characters (*) in report filters, but they are only allowed in filters that use the operators "==" or "!="; filters of any other type that contain "*" are in error, and you are prompted to re-type the filter. For example, you can use the wildcard in these filters:

- &asm.mbr.name==part*
- &asm.mbr.name==*my*, *your*

The first filter would match the strings "part," "part1," "part_A," and "partabcdefg." The second filter would match the strings "my," "this_is_my_assembly," "autonomy," "not_yours," and "your."

If you add a backslash (\) in the right-hand side of a filter (after the operator), the system reads the character after the backslash literally (as itself), not as a special character. You can then filter for an asterisk character. For example, &asm.mbr.name>=part* matches strings that are alphanumerically greater than or equal to the string "part*".

The system interprets backslashes in report parameters or between string quotes literally as backslash characters. It interprets the parameter "&asm.\mbr.name" as a report symbol named "asm.\mbr.name" (an invalid name), and the filter:

&asm.mbr.name=="match this\”

matches only the string "match this\”.

It treats a backslash that is not in a report parameter, not enclosed by string quotes, and not followed by another character as a null string (" "). It interprets anything between two string quotes literally; the filter:

&asm.mbr.name<":***"

matches strings that are alphanumerically smaller than ":***". For filters created prior to Release 11.0 that contain the wildcard character, the system interprets the character literally (as an asterisk).

Note: You should not use filters on the system parameter symbol "&rpt.index".

Excluding Items from a Repeat Region

About Excluding Items from a Repeat Region

You can exclude items from a repeat region by graphically selecting them in the report using the **FILTER TYPE > By Item** command.

The following restrictions apply:

- You cannot filter items if the table is frozen.
- You cannot exclude the following:
    - Items in 2-D repeat regions and subregions (you can use **By Rule** to filter out columns or rows in 2-D repeat regions).
    - Records that have comment cells and dash items.
    - Records that have no associated parametric attachment information.

To Remove Selected Items from a Repeat Region

1. Click **Tables > Repeat Regions**. The Menu Manager opens.
2. Click **TBL REGIONS** > **Filters**.
3. Select the repeat region to which you want to add a filter.

4. Click **Filter Type > By Item**.

5. Click **FILTER Item** > **Exclude**.

6. Click the line or draw a selection rectangle around the lines to exclude.

Example: Excluding Items from a Repeat Region

Select these records to exclude.

index	type	name	level	qty
1	PART	TESTPLATE	1	
2	PART	PLUG	1	
3	PART	PLUG	1	
4	PART	PLUG	1	
5	PART	WASHER_PART	1	
6	PART	WASHER_PART	1	
7	PART	WASHER_PART	1	
8	PART	SQUARE	1	
9	PART	SQUARE	1	
10	PART	SQUARE	1	

The system updates the repeat region to show the new order.

index	type	name	level	qty
1	PART	TESTPLATE	1	
2	PART	SQUARE	1	
3	PART	SQUARE	1	
4	PART	SQUARE	1	

Sorting in a Repeat Region

About Sorting in a Repeat Region

Using the **Table > Repeat Region > Sort Regions** menus, you can sort the contents of a repeat region to change the order in which the system lists entries in a table:

- When you choose **Forward**, it sorts entries forward, by ASCII character value.
- When you choose **Backward**, it sorts entries in reverse order.

You can specify more than one parameter symbol for sorting a region; the system sorts entries by the first parameter, and then by each succeeding parameter, if necessary. You can enter user-defined parameters as sorting parameters.

For the purposes of sorting, the system considers a text string in a repeat region to consist of two parts:

- A nonnumerical part, which comprises everything in the string that precedes the numerical part.
- A numerical part, which comprises all contiguous numbers at the end of the string, possibly including a decimal point.

Note: The system considers a leading "+" or "-" to be a portion of the numerical part of the string if the string contains no other nonnumerical data.

In any given string, either the numerical or the nonnumerical part can be empty. The system determines the sort order by making a comparison between each pair of strings in the field selected as the sort key. Instead of comparing each pair of strings strictly as strings, it considers the numerical and nonnumerical parts separately. If the nonnumerical parts of two strings are identical, it determines the order by considering their numerical parts as numbers. A sorting parameter is valid only for the specified repeat region. Repeat regions that are nested are not affected by sorting parameters at a higher region.

Use the commands of the **Sort Region** menu to add, edit or remove sorting rules.

To Add a Sorting Parameter to a Region

1. Click **Table > Repeat Regions**. The menu manager opens.
2. Click **TBL REGIONS** > **Sort Regions**.
3. Select within the region you need to sort.
4. Click **SORT REGION** > **Add**.
5. From the **SORT ORDER** menu, choose **Forward** or **Backward**.
6. Select the repeat region parameter by which you want to sort. Click **Done Sel** when you have finished.
7. Click **SORT REGION** > **Done**. The system reorders the entries in the repeat region.

Indexing Repeat Regions

About Sequentially Indexing Separate Repeat Regions

You can link the indexing of separate repeat regions in the same table by using the REGION ATTR menu. By doing so, you can specify that the index numbering for one repeat region should continue in sequence from where the numbering for another repeat region ended.

To Link the Indexing of Two Repeat Regions

1. Click **Table > Repeat Regions**. The menu manager opens.
2. Click **TBL REGIONS** > **Attributes**.
3. Select inside the repeat region for which you want to set the indexing.
4. Click **REGION ATTR** > **Start Index**. Select the repeat region from which you want to continue the indexing.
5. Click **Done/Return** to finish; the system updates the value of "&rpt.index" for the selected repeat region.

Note: You cannot use sequential indexing within nested repeat regions.

Example: Sequentially Indexing a Report Table

In the following report, the index number in the second region begins at 1. This might not be desirable, since the assemblies listed in the second region are still members of assembly "Main Vessel," like the parts in the first region. The **Start Index** command continues the numbering in the second region in sequence from where the numbering in the first region ends.

Index Number in the Second Region Beginning at 1

INDEX	NAME	TYPE	QTY
1	SUPPORT_BASE		1
INDEX	NAME	TYPE	QTY
1		LARGE_PIPE	1
2		SMALL_PIPE_ASSY	1

To Display the Table Shown Next

1. Click **Table** > **Repeat Region**. The menu manager opens.
2. Click **TBL REGIONS** > **Attributes** and select the second repeat region.
3. Click **Start Index** and select the first repeat region.
4. Click **Done/Return**. The system updates the index numbering of the second repeat region so that it continues the numbering begun in the first region. To return the index numbering to its original state so that it starts at 1, choose **REGION ATTR** > **No Start Idx**.

Example: Updated Index Numbering of Second Repeat Region

INDEX	NAME	TYPE	QTY
1	SUPPORT_BASE		1
INDEX	NAME	TYPE	QTY
2		LARGE_PIPE	1
3		SMALL_PIPE_ASSY	1

Fixing an Index

About Fixing an Index

Using the **Fix Index** command on the **TBL REGIONS** menu, you can *fix* the index of a repeat region record so that it remains the same even after you insert additional items into the repeat region or sort the repeat region differently. When fixing an index of a repeat region, keep the following restrictions in mind:

- If you fix the index of a record to be larger than the size of the repeat region, that record appears at the end of the repeat region.

- If you remove a record from a repeat region whose index has been fixed (for example, a component is suppressed in an assembly), the fixed index does not appear in the repeat region until you unfix it or use it for another record.
- If you change the attribute of a repeat region from **Duplicate** to **No Duplicate** (or vice versa), the fixed index no longer appears. However, if you change the attribute back to its original setting, Pro/ENGINEER replaces the fixed index.
- You cannot use an index with the following symbols:
    - All symbols of the type "asm.mbr.cparam"
    - All symbols of the type "asm.mbr.cparams"
    - All symbols of the type "asm.mbr.cblprm"
    - All symbols of the type "asm.mbr.cblprms"
    - All harness symbols
    - All family table symbols
    - All symbols showing cabling terminator names or types
- If a record cannot have comment cells and dash items, you cannot fix its index.
- If you fix a record’s index, you cannot dash its "rpt.index," and vice versa.
- You cannot use a fixed index for 2D repeat regions.
- You cannot fix the index of a process symbol, that is, all symbols of type "prs".
- A fixed index takes precedence in the following situations:
    - Start index of a repeat region. For example, if a repeat region starts at index 12 (taken from the last index of another repeat region), but one of its records is fixed at 2, that record appears first in the repeat region with index 2.
    - Sort keys of a repeat region. The system determines the position of a record by its fixed index if it has one. For example, a record is always at the beginning of a repeat region if its index is fixed to 1.

You can fix the index or set the start index for records that represent flexible components, family table generic components, bulk items, and included items along with the standard components within a repeat region.

To Fix the Index of a Repeat Region

1. Click **Table** > **Repeat Region**. The Menu Manager opens.
2. Click **TBL REGIONS** > **Fix Index**; then select a repeat region. The **FIX INDEX** menu displays the following commands:

- **Fix**—Fixes one or more indexes for one or all records in a repeat region.
- **Unfix**—Unfixes an index, all indexes in a record, or all indexes in a repeat region.

- **Record**—Fixes or unfixes one or all indexes in a record of a repeat region (use with **Fix** or **Unfix**).
- **Index**—Unfixes an index (this command is only available if you choose **Unfix**).
- **Region**—Fixes all indexes in all records in a repeat region, or unfixes all fixed indexes in a repeat region (use with **Fix** or **Unfix**).

3. Click **Fix** and **Record**. Select a record in the current repeat region (to fix the indexes in all records in a repeat region, choose **Fix** and **Region)**.
4. Type the desired index for that record (you cannot use indexes that are already fixed for other records).
5. Click **Done**. The system fixes the index for that record.

   Note: To display any changes that you make using the commands in the FIX INDEX menu, you must choose **Done**.

To Unfix an Index of a Repeat Region

- To unfix all indexes in a record, choose **Unfix** and **Record**. The system highlights all records whose indexes are fixed. You can then select the records that you want to unfix.
- To unfix one index, choose **Unfix** and **Index**; then type a fixed index.
- To unfix all fixed indexes in all records in a repeat region, choose **Unfix** and **Region**; then choose **Confirm** from the CONFIRMATION menu.

Using Comment Cells

About Adding Comment Cells

A comment cell is a cell in a repeat region that contains user-supplied text rather than data that is read from a model. Using comment cells, you can annotate data in a row of a repeat region. Your additional text remains associated with that row, even if the row's location within the region changes.

Pro/ENGINEER tracks a comment cell to a particular model (not a parameter value), so that when the model name changes, the comment is not lost. You can use comment cells in all reports except those for family tables and cable harnesses. However, you cannot use comment cells with the following parameter symbols:

- All parameter symbols of the following type:
  - "asm.mbr.cparam"
  - "asm.mbr.cparams"
  - "asm.mbr.cblprm"
  - "asm.mbr.cblprms"
  - "PRS"

- All harness parameter symbols
- All family table symbols
- All parameter symbols showing cabling terminator names or types

Before creating a comment cell, you must list the data in the table so that you can choose a particular object to comment.

You can add comment cells for records that represent flexible components, family table generic components, bulk items, and included items along with the standard components within a repeat region.

To Create a Comment Cell

1. Click **Table** > **Repeat Regions**. The Menu Manager opens.
2. Click **Comments** > **Define Cell**.
3. Select an empty cell within an existing repeat region. The system highlights the cell. All cells in that column are now comment cells.
4. To enter text, select the cell and click **Edit** > **Properties**. The **Note Properties** dialog box opens.

   Type text in the comment cell for a particular entry in the **Text** tab of the **Note Properties** dialog box. The text remains associated with that entry regardless of how the system sorts the region.

To Delete a Comment Cell

1. Click **Table** > **Repeat Region**. The Menu Manager opens.
2. Click **Comments** > **Clear Def**.
3. In the repeat region template, select a comment cell to delete. The system removes all comment text from this cell and from all of the comment cells that follow it.

Whenever you remove a row in the repeat region from the table (either by using a filter or by changing the setting for **Duplicates/No Duplicates/No Dup/Level** or **Recursive/Flat**), the system deletes the text of a comment cell in that row. It does not redisplay the text if you return the row to the table by clearing the filter or resetting the region attributes.

Using Dash Symbols with Parameters in Reports

To Use Dash Items

Using the **Dash Item** command in the **TBL REGIONS** menu, you can convert selected "rpt.qty" and "rpt.index" values in a drawing or report table to a dash "-". To remove a dash symbol from the table, select the symbol and the appropriate parameter value redisplays in the table.

Tip: Dash Symbol Associativity

Dash symbols are associated with the data they replace, just as table comments are, and can be lost if you modify the repeat region's attributes. For example, if you replace a report quantity value of 3 with a dash symbol, and then modify the region's attributes to duplicates, the system warns you that the content of comment cells or dash items can be lost and asks you if you want to continue. If you continue, it converts the dashed symbols to the appropriate parameter value display.

Writing Relations for Reports

About Writing Relations in Reports

Using the **Relations** commands in the **Table > Repeat Region > TBL REGIONS** menu, you can write relations among parameter symbols in a repeat region and output the computed information in the same repeat region. The system stores the relations' parameter symbols created in a repeat region with it and you cannot reference them outside of the region. In assignment statements, you can put only new parameter symbols on the left-hand side. You must refer to parameter symbols in repeat regions by specifying their full name and converting the period (.) to an underscore (_), as shown in the following figure. To use the following example, you *must* add "rpt_qty" as a repeat region parameter symbol.

Example of a Relation

```
IF asm_mbr_type=="BULK ITEM"
   Qty=50
ELSE
   Qty=rpt_qty
ENDIF
```

To Write a Relation Among Parameter Symbols in a Repeat Region

1. Click **Table** > **Repeat Region**. The **TBL REGIONS** menu manager opens.
2. Click **TBL REGIONS** > **Relations** and select the region. the **Relations** menu opens.
3. Click **Add**
4. Create a relation using the report parameters of the selected region, or create new parameters to show in the same region.

   Note: If a report parameter is not in a repeat region, you can use it to create a relation if you add it to the relation set first by choosing **RELATIONS** > **Add Param**.
5. Click rpt.., rel.. and User Defined in the **REPORT SYMBOL** dialog box to use this relation in the repeat region.
6. Type the name of the parameter in the repeat region relation.

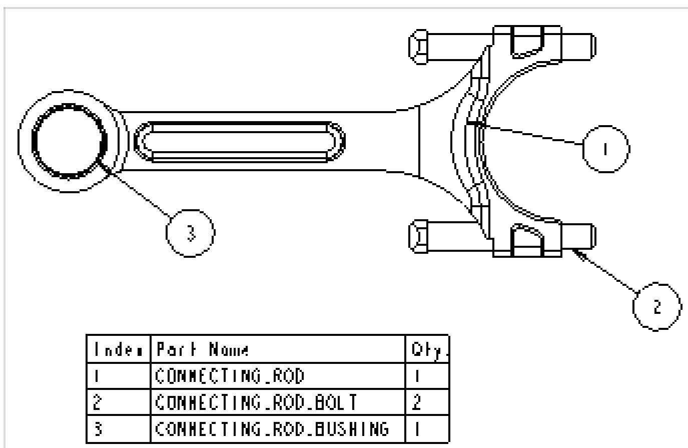
Tip: Accessing Dimension Values

To access dimension values in a repeat region, use the report parameter symbol "asm.mbr.d*". To use them in a relation, type param1 = asm_mbr_d1 * asm_mbr_d2. If the report parameter symbols "asm_mbr_d1" and "asm_mbr_d2" are not in the repeat region text, you can add them as parameters into the repeat region relation first by choosing **Add Param** from the RELATIONS menu. You can only access dimension values of the *current* model.

Using BOM Balloons

About BOM Balloons

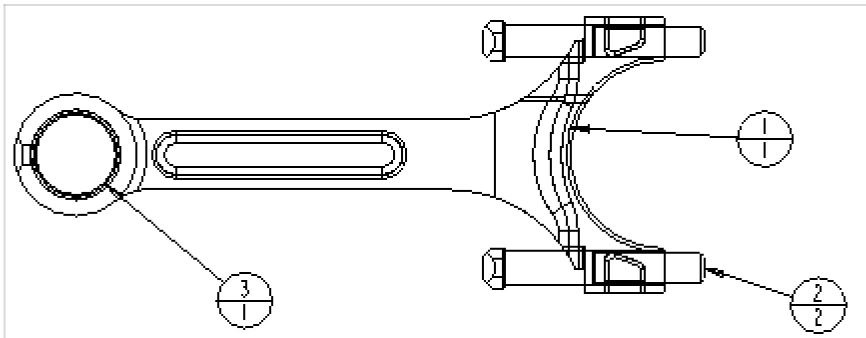
BOM balloons are circular callouts in an assembly drawing that show Bill of Materials information for each component of an assembly view. The information is derived from a report table repeat region that you also specify as a BOM balloon region. Before you can add BOM balloons, you must create the table, add the repeat region, enter the desired report symbols, and designate the BOM balloon region. When you have done this, you can show BOM balloons on a selected assembly view.



BOM balloons usually show an index number, corresponding to a part name in the table. The report symbol for the index number is rpt.index. The report symbol for the part name is asm.mbr.name.

BOM Balloon Types

Optionally, you can split the BOM balloon into an upper and lower half. The upper half will carry the index number, (or any other parameter you may want to assign) and the lower half will carry the quantity, or how many of this part exist in the view. To show the quantity, the report symbol for quantity, rpt.qty, must be included in a cell within the repeat region.



Index	Part Name	Qty
1	CONNECTING_ROD	1
2	CONNECTING_ROD_BOLT	2
3	CONNECTING_ROD_BUSHING	1

Balloons that show only one report symbol (usually the index number) are called **Simple** balloons. Balloons that are split to show the index number in the upper half and the quantity of the part in the lower half of the circle are called **With Quantity** balloons.

There is a third type of balloon called the **Custom Balloon**. This type lets you specify a drawn symbol that you have created and stored to use as the BOM balloon symbol. You can use notes to assign variable text attributes to the symbol. These notes may have the BOM balloon read out the values of any number of parameters you want to include in it.

Selecting and Highlighting BOM Balloons

When you select a BOM balloon, its parent line in the table region is highlighted. If the BOM balloon is one of a multiple quantity, the other balloons for the same component are also highlighted. You can also select lines in the table region to highlight the associated BOM balloons.

To Control BOM Balloon Size

The BOM balloon size is controlled by the text size of the balloon contents. To make a selected balloon or balloons larger or smaller,

1. Click **Text Style** from the right mouse button shortcut menu. The **Text Style** dialog box opens.
2. Use the **Height** field to enter a height for the text.
3. Click **Apply** or **OK** to finish.

If you want to apply a range limit to BOM Balloon size, use the detail setup options (**File** > **Properties** > **Drawing Options**)

`min_balloon_radius`

`max_balloon_radius`

Set a numeric value for each. the values apply to all newly created balloons. If you set a value of zero there is no min. or max. radius.

To Set a BOM Balloon Region in a Table

1. Click **Table** > **BOM Balloons...** The **BOM Balloon** menu opens on the Menu Manager.
2. Click **Set Region**. The **BOM Bal Type** menu opens on the Menu Manager.
3. Click the type of BOM balloon you want to set.
4. Click the repeat region in the table that you want to be the source for the BOM balloons. You are prompted: "Balloon attribute has been set for the selected region."
5. You may select another region to set, or click Done to finish. Use **Clear Region** to remove the BOM balloon region.

To Show BOM Balloons in an Assembly View

To show BOM balloons in an assembly view, the drawing must have a table with a repeat region containing at least the report symbol for the report index number (rpt.index) and the model name (asm.mbr.name). Additionally, the repeat region must be set as a BOM balloon region.

You can assign BOM balloons to bulk items and included items, that are nonsolid representation of components, along with standard components within an assembly.

When these conditions in place, you can add balloons as follows:

1. Select the target component on the Model Tree or in the drawing.

   Note: You can hold down the CTRL key to select multiple components at once on the Model Tree or in the drawing.
2. Use the shortcut menu to select **Create BOM Balloons.** You are prompted for the view in which to place the balloons.
3. Select the view in which to show the balloon. For quantity balloons, you may be prompted to enter a number for how many of the total quantity you want the balloon to represent.

Note: If a balloon is shown for a component, you can use the shortcut menu to add a reference balloon for other instances of the component.

Alternatively, use one of the methods in the Menu Manager-

1. Click **Table** > **BOM Balloons**. The **BOM Balloon** menu appears on the Menu Manager.
2. Click **Create Balloons**.
3. Click a BOM balloon region in a table. The **BOM View** menu appears on the Menu Manager.

4. Click one of the following commands:

   Show All - Shows all balloons associated with the table region. They may be spread across several views depending on the orientation of the views.

   Note: If you have shown a balloon for every row in a repeat region, you can add additional balloons for every instance of the component in the model by clicking **Show All** again. You are prompted to add additional balloons for each assembly component.

   By View - If a region refers to more than one view, select the view in which to show balloons.

   By Comp - Select a specific component or components for which to show balloons.

   Comp & View - If a region refers to more than one view, select the view in which to show balloons.

   By Record – Select a record in the BOM table to show balloons. You can only select records that represent either bulk items or included items.

To Clean the BOM Balloon Layout

To automatically clean the BOM balloon layout:

1. Select BOM balloons or a view having BOM balloons.
2. Click **Edit** > **Cleanup** > **BOM balloons**. The **Clean BOM Balloons** dialog box opens.
3. Use the dialog box to automatically set the position and spacing of a set of BOM balloons. You can set an offset from the view outline, guide by existing snap lines, and-or set up a stagger increment value. You can determine whether the leaders will point to edges or surfaces.
4. To preview the results, click **Apply**. To accept them, click **OK**.

These detail setup options control the default settings for the **Clean BOM Balloon**s dialog box:

def_bom_balloons_view_offset	offset distance between ballons and view boundary
def_bom_balloons_stagger	specifies whether balloons are shown staggered by default
def_bom_balloons_stagger_value	distance between consecutive offset lines in staggered is Yes
def_bom_balloons_snap_lines	determines if snap lines are created around the view
min_dist_between_bom_balloons	minimum distance between bom

	balloons
def_bom_balloons_attachment	the default attachment method

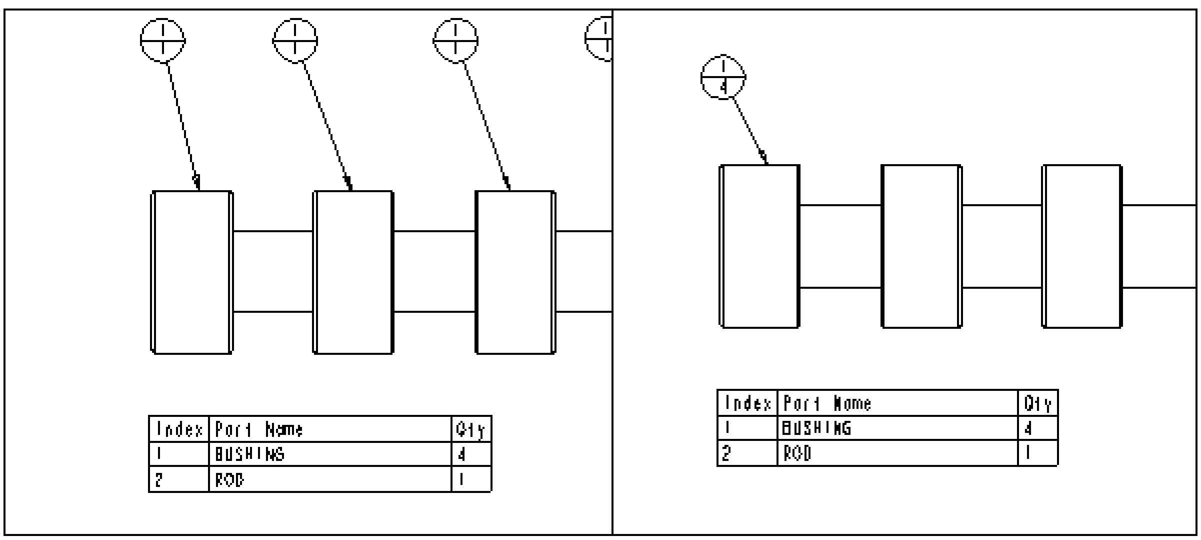
To Change the BOM Balloon Type

To change the balloon types that are already shown:

1. Click **Table** > **BOM Balloons**. The BOM Balloons menu opens on the Menu Manager.
2. Click **Change Type** and select the BOM balloon region in the source table. The **BOM Bal Type** menu opens on the Menu Manager.
3. Click the alternate type you want to use, and click **Done**. The type is changed.

To Merge and Stack BOM Balloons

You can show quantity balloons as one balloon showing the total instances of the component in the quantity portion, or as one balloon for each instance of a component. See examples below-



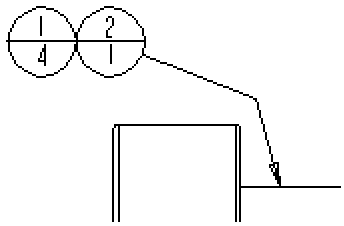
To merge quantity balloons, (for example, to go from the example on the left to the one on the right)

1. Click **Table** > **Bom Balloons**. The menu manager opens.
2. Click **Merge**. You are prompted to select a quantity balloon to be merged.
3. Select the balloon. You are prompted to select a balloon to be merged into.
4. Select the "host" balloon. The first balloon is removed and the quantity of the host balloon is increased by one.

   The merge command is still active and you can repeat the process. Click OK in the select dialog box to end the merges.

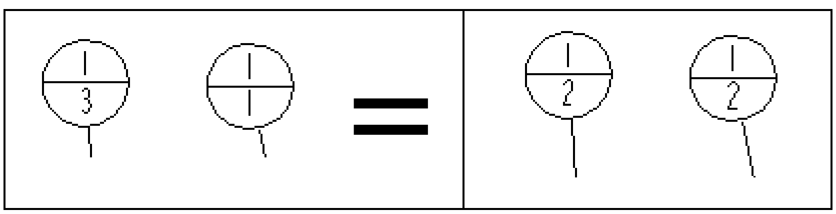
To reverse the process, select one of the components without a balloon and click **Create BOM Balloon** from the right mouse button shortcut menu. If necessary, you are prompted for the number you want in the quantity value. (You can also use the **Split** command in the **BOM Balloons** menu manager to create new balloons from merged balloons.)

To Stack BOM Balloons



Use the **Merge** command to stack quantity balloons with different index numbers. If the index numbers are different, the balloons are adjoined horizontally. Use the **Detach** command in the **BOM Balloons** menu manager to separate the stacked balloons.

To Redistribute Quantity Among BOM Balloons



1. Click **Table** > **BOM Balloons**... The Menu Manager opens to the **BOM BALLOONS** options.
2. Click **Redistribute** in the Menu Manager.
3. Select a BOM balloon with a quantity greater than 1. If there are multiple possibilities for redistributing the quantity, you are prompted to perform step 4. Otherwise, you are prompted to select the "target" balloon.
4. Type the quantity that you want to subtract from this balloon and add to another balloon for the same model.
5. Select the balloon to which you want to add the quantity. The quantities of the balloons change accordingly.

To Change the Balloon Leader Attachment Point and Style

1. Select the balloon to change.
2. Right-click and click **Edit Attachment** on the shortcut menu. The **ATTACH TYPE** menu appears.

3. Use the menu to apply an attachment location and a leader style to the balloon. For example, to change an arrowhead pointing to an edge to a clear dot attached to a surface,

    a. Highlight On Surface and Dot in the menu

    b. Click the surface to which you want to attach the dot. The leader end is changed to a dot and relocated to the surface area.

4. Click **Done** to complete

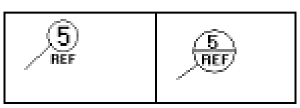
Note:

- For a **Flat** repeat region, you can move the BOM balloon for a part or subassembly on the top level, only to another part or subassembly on the top level if the region is specified as **No Duplicates**.
- For a **No Dup/Level** region, you can move the BOM balloon for a part or subassembly only to a part or subassembly having the same parent assembly.
- For a **Recursive/No Duplicate** region, you can move the BOM balloon for a part or subassembly anywhere in the model where the same part or subassembly is used.
- For a **Duplicates** region, you cannot move the BOM balloon for a part or subassembly to any other component.

To Add Reference BOM Balloons

You can create additional reference BOM balloons for each component in different views. The REF designation appears as shown below for simple and quantity or custom balloons. Reference BOM balloons are only available for components that already show BOM balloons.

Note: You cannot create reference balloons for bulk items, included items, flexible components, and family table top generics.



To add a reference balloon,

1. Set the drawing filter to "component."
2. Click the part to which to add the balloon. (You can also select from the model tree.)
3. From the right mouse button shortcut menu, click **Add Reference Balloon.** The reference balloon is added to the part.

Note: The text for the REF balloon is set by the drawing setup option `reference_bom_balloon_text`

To Swap Custom Symbols

1. Click **Table** > **BOM Balloons**. The menu manager opens.

2. Click **BOM BALLOONS** > **Alt Symbol**. (If multiple regions are present, you are prompted to select a balloon region.)
3. Select quantity or custom balloons of the selected repeat region, then click OK.
4. From the **GET SYMBOL** menu, select a user-defined symbol. Any selected balloons change to match the selected symbol, and the system redisplays the new balloons using the selected symbol for any selected components.
5. For regions that use a user-defined symbol to display balloons, choose **GET SYMBOL** > **Reg Default**. The selected balloons revert to the default symbol of the region, and the system shows new balloons for the selected components using that symbol.

   Note: You cannot change individual simple-customized balloons to quantity-customized balloons, and vice versa. All balloons for any given region must be either quantity or simple balloons.

Creating Customized BOM Balloons

Using the **Custom** command in the BOM BAL TYPE menu, you can create customized balloons with user-defined symbol instances. When you use a customized symbol to show balloons, the system substitutes variable text with corresponding report parameters. If the default value of the variable text matches a report symbol specification, it displays the value of that report symbol in the symbol.

For example, if the variable text default value is `asm.mbr.name`, the system displays the member name in the balloon. It updates (in the symbol instance) only the report parameters that it displays in the balloon region; other variable texts simply display their default values. In addition to report parameters displayed in the table, the system uses two special keywords as default values for variable text: it replaces "qty" with the quantity, as in default quantity balloons, and it treats "index" as the index field in default balloons. By default, for variable text whose default value is "index," it fills them in with the value of "rpt.index."

You can use the **Set Param** command in the **BOM BALLOON** menu to specify a different parameter for the index field. The system considers a customized balloon to be a quantity custom balloon if it contains variable text whose default value is "qty."

Tip: Setting the Default Arrow Style for BOM Balloons

To set the default arrow style for BOM balloons, use the drawing setup file options

`def_bom_balloons_edge_att_sym`

and

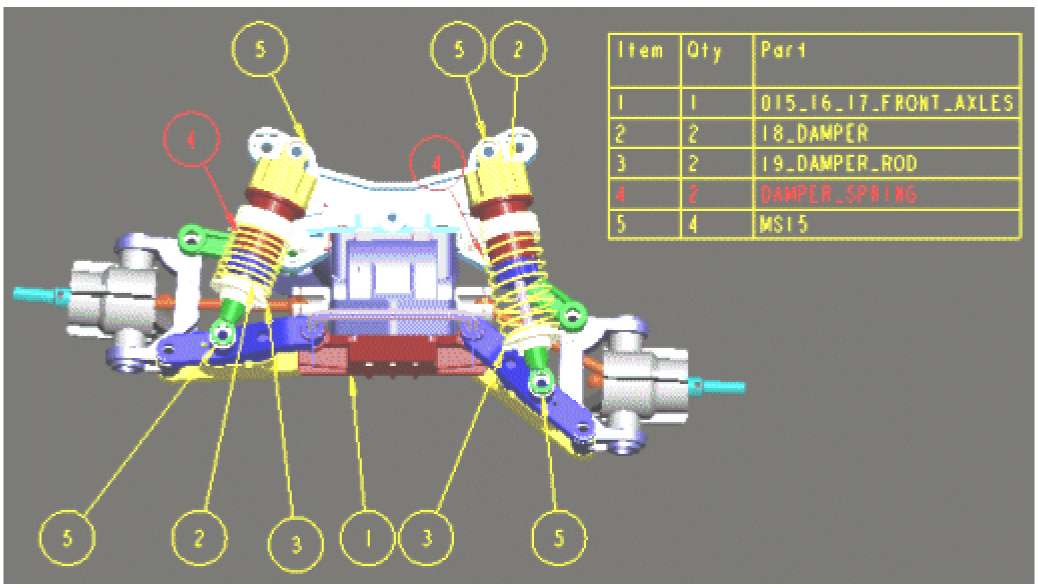
`def_bom_balloons_surf_att_sym`

The respective choices are listed in the options pull-down list in the **File** > **Properties** > **Drawing Options** dialog box.

Creating BOM Balloons for Flexible Components

You can create BOM balloons for components defined as flexible within an assembly. The BOM balloons for flexible components are created regardless of the attribute you set for the repeat region. That is, you can set the attribute for the repeat region as Duplicate or No Duplicate.

For example, the following assembly consists of two springs (`DAMPER_SPRING`) that are defined as flexible components, each with a different flexible value. The attribute of the repeat region is set to No Duplicate. These two springs are represented in a single row with an identical index number. However, the quantity appears as two. When you create BOM balloons for the first time, a single balloon is created that represents all instances of flexible components with the same name. However, when you create BOM balloons the second time, each instance of the flexible components with the same name is assigned a balloon with an identical index number.

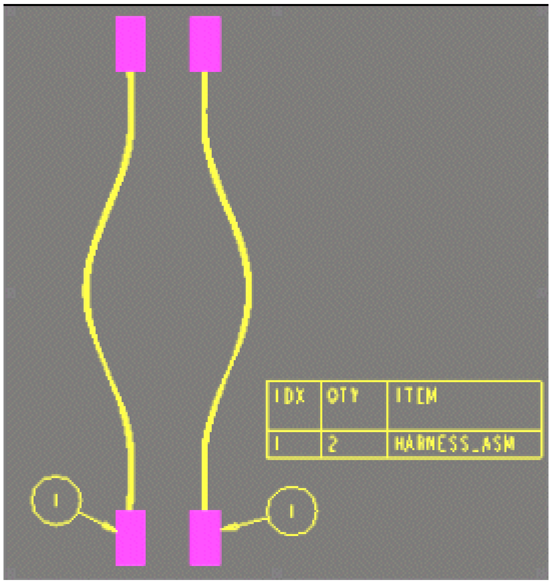


Creating BOM Balloons for Family Table Instances

You can create BOM balloons associated with the components that represent family table top generic and family table generic instances. The `asm.mbr.top_generic.name` and `asm.mbr.generic.name` report parameters represent these instances in a report table.

For example, the following assembly contains two instances of the same harness, one with a left-handed route and the other with right-handed route. These harness are represented as 2 different instances of `harness.prt` in a family table. Here the BOM reports the top level generic name of the harness assemblies (`HARNESS_ASM`) used in a repeat region with the No Duplicate attribute.

You can show BOM balloons associated with the BOM table records that are attached to the family table instances. When you create BOM balloons for the first time, a single balloon is created that represents all instances of the family table generic and top generic components. However, when you create BOM balloons the second time, each instance of the family table generic and top generic components is assigned a balloon with an identical index number.



Controlling Drawing Details with Layers

About Controlling Drawing Details with Layers

You can organize and control your drawing detailing by assigning drawing items to layers. You can hide and show layers as necessary, which allows you control the display of multiple drawing items simultaneously. For example, you can assign notes to a specific layer and then hide them before plotting the drawing.

Using layers, you can organize items and streamline selection because all the layer items are treated as a group. Layers can include other items in a Pro/ENGINEER database such as features, dimensions, notes, geometric tolerances, and other layers.

You can place drawing items on layers in the following ways:

- **Manually—**Place the items on a layer after they are added to the drawing. You can individually select the items to add or you can use region selection.
- **Default Layers**—Define specific items to automatically place on a layer as they are created. Detail items, such as driven dimensions, draft entities, and snap lines can be added using this method. You can either specify detail items using the individual default layer settings or you can add all detail items to a default layer. The same default layers that exist in 3D mode, for example, layers for curves, mechanical features, or rounds, exist in Drawing mode.

Although drawing views do not have individual layers for each view, you can control the display status of drawing layers for individual views by overriding the model layer status. Controlling the layer display for individual views enables you to:

- Include items from parts and assemblies directly in a drawing layer without including them on a layer in a model.

- Control the display of drawing layers to include items from the model without marking the model as changed.
- Vary the display of a single model in different drawing views.

Layers and their contents can be accessed from the Model Tree. To show the Layer Tree, click **Show** > **Layer Tree** on the Model Tree window. The display status (hidden, unhidden, or isolated) of a layer controls whether or not items in the layer are displayed.

Note: When blanking or displaying layers containing datum curves, the hidden lines of three dimensional objects are recalculated to ensure that the drawing is updated and accurate. You can reduce the regeneration time by excluding datum curves from these hidden line calculations using the `hlr_for_datum_curves` drawing setup file option.

About Utilizing Default Layers in Drawings

You can automatically place specific drawing item types on a layer by defining a default layer. You can use the `def_layer` configuration option to define which drawing item type to automatically place on a layer. When you create or show a drawing item type, it is simultaneously placed on the layer.

For example, if you use the `layer_dim_<layer name>` configuration option and create or show a dimension, it is automatically placed on the designated layer. You can either define a default layer that contains only one drawing item type, such as dimensions, or you can place all drawing item types on a particular layer.

You can also activate a layer to control the placement of drawing items on that layer. Though multiple layers can exist in a drawing, only one layer is active at a time.

You must manually activate and deactivate an existing or a created layer.

A default layer is activated automatically if you create it using the `def_layer` configuration option. Otherwise, you must manually activate it. If you manually create the layer (**Layer** > **New Layer**), you must manually activate it.

If a drawing has one model or multiple models, drawing item types are placed on the layer for the reference model. For example, if you create a geometric tolerance in a drawing that uses a part as its model reference, the geometric tolerance is stored in the active layer of the part.

Note:

- Items shown for the first time are added to the active layer. However, if a shown item was previously erased or removed from the layer, it is not added to the layer.
- You cannot activate ruled layers.
- You can use default layer models to generate ruled layers in parts the same way the `def_layer` configuration option works. This allows more complex ruled layers to be automatically created in models when needed.

About Changing the Display Status of Individual Drawing View Layers

You can change the display status of layers in a drawing only after you have set the drawing to ignore the layer status of its model by setting the `ignore_model_layer_status` drawing setup file option to `yes`.

The status of layers of drawing views is always independent of the status of the main drawing layer. Changes that you make to the view layer are not reflected in the layer of the main drawing.

To make the status of a drawing view layer dependent on the status of the drawing layer, select the required view from the Active Layer Object Selection list above the Layer Tree and click **Layer** > **Drawing Dependent**.

Making a view layer status dependent on the drawing layer is a temporary operation and the change is only seen in the graphics window. For example, if you have a note in a drawing view on the view layer and if you hide this layer, the layer is hidden at the view level and the note disappears from the graphics window. The note reappears when you make the view layer dependent on the main drawing layer using the **Drawing Dependent** command.

When you use the **Drawing Dependent** command on a view layer, the selected object in the Active Layer Object Selection list automatically moves back from the selected view layer to the main drawing or model. You can switch back to the independent view by selecting it from the Active Layer Object Selection list above the Layer Tree.

You can make the view layer permanently dependent on the drawing layer by clicking **Layer** above the Layer Tree, and clicking **Copy Status From** > **Drawing**. Alternatively, select a layer on the Layer Tree, right-click, and click **Copy Status From** > **Drawing**. However, once you do this you cannot reset the status of the view layer.

Invisible Drawing Model Items

Items in the drawing model are visible by default unless one of these conditions is met:

- The item is included directly on a drawing layer.
- The item is on a part or assembly layer with the same name as a drawing layer and the drawing setup file option `draw_layer_overrides_model` is set to `yes`.
- The item is on a part or assembly layer that has been included in a drawing layer.

For these three cases, the drawing layer status information determines whether to show the item.

To Place Drawing Items on Layers

Placing drawing items on layers enables you to control the display of multiple items simultaneously. Layers are a good way to organize items and can streamline selection since all the layer items are treated as a group.

You can place drawing items on layers manually or automatically.

To Manually Place Drawing Items on Layers

1. On the Model Tree, click **Show** > **Layer Tree**. Any existing layers are listed.
2. Define the layer to place the drawing items on:
    - To create a new layer, click **Layer** > **New Layer**. The **Layer Properties** dialog box opens.
    - To place the items on an existing layer, right-click the layer and click **Layer Properties**. The **Layer Properties** dialog box opens.
3. Click **Include** and select the desired items from the graphics window or Model Tree. Use the drawing filter to limit the items available for selection. A + appears next to the included items.
4. Click **OK**.

To Automatically Place Drawing Items on Layers

Create a default layer by setting the `def_layer` configuration option. You can more easily control on which layer the items are placed by setting the layer as active.

You can place the following drawing items on an active layer:

- Draft entities including draft datums, groups, and axes
- Notes and parametric notes
- Geometric tolerances and their datums
- Dimensions (model, drawing, reference)
- Datum targets
- Symbols and Surface finish symbols
- Tables
- Snap lines

You can also add the following three-dimensional detail items:

- Notes
- Driven dimensions
- Reference dimensions
- Geometric tolerances
- Symbols and Surface finish symbols

To Control Individual View Display Using Layers

You can change the display status of model layers within a drawing without changing the part or assembly in which that item was created.

You can force a drawing to completely ignore the layer status in its model when determining if it should display an item on a layer. Pro/ENGINEER displays all items on model layers in the drawing, and you can manipulate the items separately at the drawing level. You can blank or display layers at the drawing level without making changes to the part or assembly in which the item was created, and the model does not change.

To set a drawing so that it ignores the layer status of its model, do one of the following:

- On the Model Tree, click **Settings** > **Drawing Layer Status**. The **Layer Status Control** dialog box opens. Select the **Ignore display status of layers in the model** box.
- Set the drawing setup file option `ignore_model_layer_status` to `yes` (the default value). If you set this option to `no`, the drawing layer status follows that of the model.

After you separate the drawing layer from the model layer you can specify independent view display, which takes precedence over the main drawing for that view. Then:

- When you create an independent view, the layer display defaults to that of the main drawing.
- When you create a detailed view, the layer display defaults to that of the parent view.
- You can then modify the display for any view independently, or reset it to follow the drawing display.
- You can copy the display status for the drawing or a view to any other view or the drawing, and then modify the display individually for each.

  Note: You can control the display of datum curves and surfaces in the shaded view of a drawing only through the model or assembly layer tree.

Tip: Modifying Layer Display

Modifying the display of a layer in a drawing does not mark any parts or assemblies as modified. Therefore, you do not need to save parts and assemblies and resubmit them to Pro/PDM when you toggle layers and save them.

Also, the drawing looks as it should when you retrieve it, regardless of the changes that you may have made to the model layers. Because one of the benefits of the `ignore_model_layer_status` setting is that changes are not be made to the model or model layers, you cannot add items or remove them from model layers. However, you can manipulate items directly on drawing layers.

2-D Drafting

About Drafting in Drawing Mode

You can add drafted entities to a drawing at any time. Using commands in the Sketch menu, you can create various geometry types: lines, circles, arcs, splines, ellipses, points, and chamfers.

You can:

- Use the same referencing and geometric constraints as you can in 3D Sketcher.
- Group separate entities with each other so they move as one.
- Group drafted geometry with a view, so that it scales up or down with the view scale.

Pro/ENGINEER also considers entities read in to a drawing from an IGES, DXF, or SET file to be drafted entities.

You can set preferences for some of snapping and sketching behaviors in the drawing environment from within the Sketch Preferences dialog box (**Sketch** > **Sketcher Preferences**). This enables you to set up common grid, angle and snapping properties for the drawing tools. You can also set sketching preferences, like activating chain sketching or parametric sketching.

Dimensions and Sketch Objects

A dimension placed between two draft entities, (created with the **Sketch** commands) or between a view and a draft entity is called a *draft dimension*. The dimension value is based on the *draft scale* and the actual drawing sheet units and size.

Associating a dimension with a draft entity links the two together so that the dimension reflects changes to the entity. When you have made a draft dimension associative (using the drawing setup file option `associative_dimensioning`), the draft dimension's value changes if you change the draft scale, move the entity, and trim and intersect the entity. The system reflects the changes to the draft scale in the value of the draft dimension when it regenerates the drawing and creates new draft entities at the scale value.

Note: Regular nonassociative draft dimensions do not change with draft scale.

Obtaining Draft Geometry Information

To locate draft geometry on a drawing, use **Edit** > **Highlight by Attributes**. The **Highlight by Attributes** dialog box lets you isolate and highlight draft entities on the sheet.

To obtain specific geometry information about one selected draft entity, such as precise endpoint positions, length, angle, and item ID number, use the **Info** > **Draft Entities...** command.

For information on multiple draft entities, such as the distance between parallel lines, the coordinates of an intersection point, or tangency point, click **Analysis** >

Measure Draft Entities. The Measure Draft Entities dialog box lets you specify the type of measurement you want to make, and to select the multiple entities you want to measure.

Drafting with Absolute Coordinates

As an alternative to freehand drafting, you can snap a line end to a specific coordinate while the line is in progress. You can also use this technique to specify precise angles and relative coordinates. or coordinates relative to the current line point location.

The `draw_points_in_model_units` configuration file option defines the current draft view's coordinate values as model units rather than drawing units. The **GET POINT** menu uses the scale of the draft view and the draft view's model units for relative and absolute coordinate entry and display in the message area.

1. With a draft line or shape command active, click **Absolute Coords** from the right mouse button shortcut menu. The **Absolute Coordinates** input box opens.
2. Enter the X and Y coordinates you want the current point to snap to. Press Enter. The point snaps to the coordinates, leaving the free end attached to the cursor. You may specify more coordinates, or left click to complete.

Understanding the Draft Scale

Draft scale is the relation between what an entity size appears to be versus its actual dimension. For example, assume that an edge of a part is 4 inches long:

- If the draft scale is 1.0, the entity appears to be 4 inches in the drawing.
- If the draft scale is 4.0, the entity appears to be 16 inches in the drawing (it appears to be four times larger than it actually is).
- If the draft scale is 0.5, the entity appears to be 2 inches in the drawing (it appears to be one-half the size it actually is).

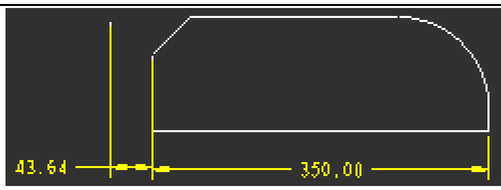
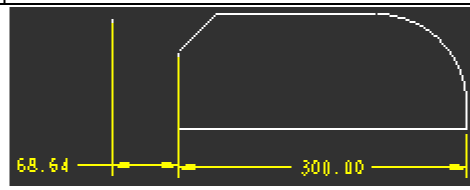
(Use the **File** > **Properties** > **Drawing Options** dialog box to change `draft_scale` to determine the default draft scale.)

Sketching Draft Geometry

Parametric Sketching in Drawings

While sketching in your drawings, you can parametrically associate draft entities with the model geometry or other draft entities. If you parametrically associate the draft entities, changes to its reference result in changes to the draft geometry.

For example, if you sketch a line and parametrically associate it to an edge and the edge moves, the dimensions of the drafted line are dynamically updated with the new edge position:

Original Draft Placement	Automatically Adjusted Draft Line
	

The draft entity constraints are applied when you sketch the entity. You cannot apply parametric constraints to previously sketched entities.

You can create dimensions for draft entities. However, they cannot drive the model geometry. You must change the model constraints from within the model.

To Create Parametric Draft Entities

1. Before sketching the draft entity, click . Parametric sketching is enabled.
2. Sketch the draft entities as you would normally sketch within your drawings.

Note:

- After sketching a straight edge parametric draft entity, you can manually extend or trim the draft entity with the drag handles that appear when the entity is selected. The length modification only removes the parametric associativity for the length value; all other constraints remain intact.
- Drag handles are not available for non-straight draft entities.
- If the model geometry is modified and causes parametric draft entities to fail in the drawing, the draft entities are highlighted and a warning prompts you to either break the parametric associativity, delete the failing draft entities, or temporarily ignore the missing model references.

To Chain Entities During Sketching

When chain geometry is in effect, the ending point of one entity automatically serves as the starting point for the next.

Chaining geometry affects only the creation of the entities. Once you have created them, you can select and move each one separately. To make drafting entities stay in relative position to each other, use the parametric drafting commands, or use **Edit** > **Group** to create a group of separate objects.

You can chain circles and ellipses together when they use the same centers. Once you have established the center of the first ellipse or circle of the chain, the system uses it for every circle or ellipse that follows, until you end the chain.

To Initiate Chain Sketching

1. Click **Sketch** > **Sketch Preferences**. The Sketch Preferences dialog box opens.

2. Under **Sketching** tools, select **Chain Sketching**.

When chaining is activated, a small square indicates the point from which the chain continues. If you select points with the left mouse button, the chain continues; if you select them with the middle mouse button, the system creates an endpoint and the chain pauses, or stops at that point.

To Create a Line

1. Click **Sketch** > **Line** to open the **Line** menu. The **Snapping References** dialog box opens. Use the arrow icon in the dialog box to add any references you want to set up before you sketch.
2. You can sketch the following types of lines continuously as a chain or as individual entities:
    - Horizontally or vertically between any two points
    - At an angle measured from the x-axis (0)
    - Tangent to a curve at its second endpoint
    - Parallel or perpendicular to a specified line
    - Normal to a curve
    - Tangent to an arc or a spline
    - Tangent to two circles or splines
    - Tangent to a curve and parallel or normal to a reference line

To Create a Circle

Click **Sketch** > **Circle** or click the circle icon on the Sketcher toolbar. The **Snapping References** dialog box opens. Use the arrow icon in the dialog box to add any references you want to set up before you sketch.

Choices for circles include:

- Locating its center and defining a radius point
- Making it tangent to other entities
- Locating its center and defining a point as the tangency with another entity
- Making it tangent to three other entities
- Making it tangent to two other entities, with a specified radius
- Using any three points that lie on the circle

To Create an Arc

Click **Sketch** > **Arc**. Using the options in the Arc menu, you can create an arc by doing the following:

- Selecting an endpoint that is tangent to another open entity (that is, not a circle or an ellipse)
- Selecting any three points that lie on the arc
- Using the arc center and two endpoints
- Using the arc center, a start point, and an angle that strokes the arc from the start point in a counterclockwise direction
- Making it tangent to three other entities
- Making it tangent to two other entities, with a specified radius

To Create a Chamfer

Chamfers are beveled edges or corners. Within drawings, you can sketch chamfers at the intersection of two draft entities.

The intersecting draft entities do not have to actually touch. The chamfer calculates the appropriate intersection. When you sketch a corner or edge chamfer, the existing intersection is deleted.

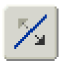
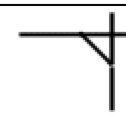
1. Click **Sketch** > **Chamfer**. The **Select** dialog box opens.
2. Select the first chamfer reference, such as a point on a model edge or draft entity. Ensure that you select a point close to the desired start point of the chamfer, as your selection point determines the default angle.
3. Press `CTRL` and select a model edge or draft entity that is the second chamfer reference. If the references are valid, the chamfer previews on the drawing.

   Note: You can select multiple sets of references to sketch several chamfers simultaneously.

4. Middle-click or click **OK** in the **Select** dialog box to confirm the chamfer references. The **Chamfer Properties** dialog box opens.
5. Define how you want to dimension the chamfer:

- **45 x d**—Create a chamfer 45 degrees from the first selected line and of dimension `d` along both selected lines (where `d` is determined by point selected on the draft entity)
- **d x d**—Use the same dimension value along both of the selected draft entities
- **d1 x d2**—Use a different chamfer dimension for each draft entity
- **Ang x d**—Use a chamfer dimension at an angle to the draft entity

6. By default, the most recently used chamfer dimension is selected. Select a chamfer dimension from the list or type a new value. If applicable, click  to interchange the angle and dimension for the chamfer.
7. Select a trim style to define how to display the intersection being chamfered:

Full Trim—Deletes the intersection, displaying only the chamfered intersection.	
Thin Line Trim—Displays the intersection with thin lines, with the chamfer in normal pen width.	
Solid Line Trim—Displays both the intersection and chamfer in the normal pen width, and any overlapping of intersecting chamfer references is trimmed .	
Half Trim—Displays half the intersection, while deleting one of the extending chamfer reference lines. The chamfer is displayed in normal pen width. **Note**: Click  to trim the remaining portion of the intersection.	
No Trim—Displays the entire intersection and chamfer in normal pen width; any overlapping remains.	

Note: If you selected multiple sets of chamfer references, and the first and last reference physically intersect, you can select **Complete loop** to add the same chamfer to all the intersections within the selected references.

8. Click **OK** in the **Chamfer Properties** dialog box. The chamfer is created.

Note: You can modify individual chamfers. You can access the chamfer properties in any of the following ways: double-click the fillet, select the chamfer and click **Edit** > **Properties**, or right-click the chamfer and click **Properties** on the shortcut menu.

To Create a Fillet (2 Tangent Edges)

A fillet rounds the corner of a selected intersection.

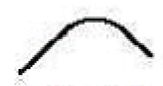
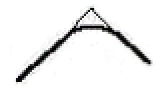
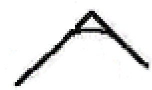
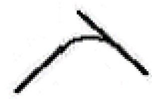
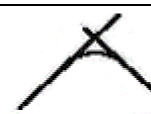
Note: If you select **Parametric sketching** in the **Sketch Preferences** dialog box before fillet creation, and select model edges as fillet references, then the selected model edges are automatically converted to draft entities on fillet creation. If you set your parametric sketching preferences to **Erase model edges behind draft entities**, the relevant model edges are erased when the fillet is complete. The **Erase model edges behind draft entities** option is selected by default when you select **Parametric sketching** in the **Sketch Preferences** dialog box.

1. Click  or **Sketch** > **Fillet** > **2 Tangent**. The **Select** dialog box opens.
2. Select either a point on a model edge or a draft entity as the first fillet reference. Ensure that you select a point close to the desired start point of the fillet, because your selection point determines the default radius.
3. Press `CTRL` and select a model edge or draft entity as the second fillet reference. You can select multiple sets of references to sketch several fillets simultaneously.

4. By default, Pro/ENGINEER provides a radius value based on your selection point locations. You can type a new radius value or accept the default. Click **OK** in the **Select** dialog box. The **Fillet Properties** dialog box opens.

5. In the **Radius Value** box, the calculated or the most recently used fillet radius value, if available, is selected. You can also type a new value.

6. Under **Trim Style**, select from the following trim styles to define how to display the intersection of the tangent edges:

Full Trim—Deletes the intersection, displaying only the fillet intersection.	
Thin Line Trim—Displays the intersection with thin lines, with the fillet in normal pen width.	
Solid Line Trim—Displays both the intersection and fillet in the normal pen width, and any overlapping of intersecting fillet references is trimmed.	
Half Trim—Displays half the intersection, while deleting one of the tangent lines. The fillet is displayed in normal pen width.  is available for selection when **Half Trim** is selected.	
No Trim—Displays the entire intersection and fillet in normal pen width; any overlapping remains.	

Note: The **Complete loop** check box is available for selection if you have selected multiple fillet references and the first and last references intersect, either theoretically or physically. If available for selection, the **Complete loop** check box is selected by default. When selected, a fillet is automatically added to complete the loop between the first and last selected references.

7. Click **OK** in the **Fillet Properties** dialog box to complete the fillet creation. The fillet appears in the drawing.

8. Click **OK** in the **Select** dialog box or middle-click to quit fillet creation.

To Modify Fillets

Fillets created with parametric sketching maintain relationships with the fillet references. If you modify the fillet radius, the reference draft lines are automatically trimmed or extended to maintain tangency.

You can modify individual fillets using the **Fillet Properties** dialog box. To open the **Fillet Properties** dialog box, do one of the following:

- Select the fillet that you want to modify and double-click.
- Select the fillet and click **Edit** > **Properties**.

- Select the fillet, right-click, and click **Properties** on the shortcut menu.

Note: The **Trim Style** list in the **Fillet Properties** dialog box is not available if you create a non-parametric fillet on one or more model edges, that is, if you select references after clearing the **Parametric Sketching** check box. The **Trim Style** list is available for non-parametric fillets on draft entities only.

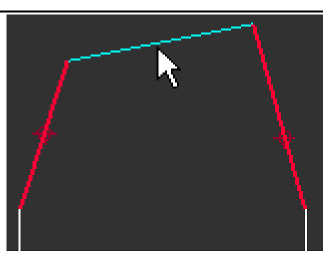
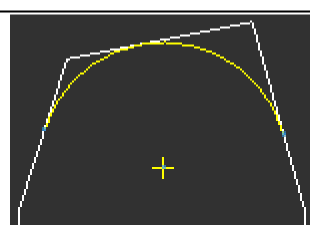
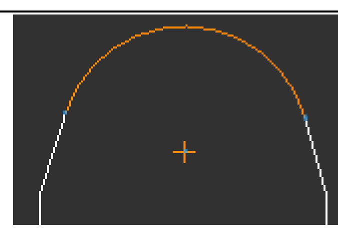
To Create a Fillet (3 Tangent Edges)

You can select multiple sets of fillet references to sketch multiple fillets simultaneously. You must select the references for 3 tangent fillets in the specified order: start point, end point, midpoint.

If you select **Parametric sketching** in the **Sketch Preferences** dialog box before fillet creation, and select model edges as fillet references, the selected model edges are automatically converted to draft entities on fillet creation. If you set your parametric sketching preferences to **Erase model edges behind draft entities**, the relevant model edges are erased when the fillet is complete. The **Erase model edges behind draft entities** option is selected by default when you select **Parametric sketching** in the **Sketch Preferences** dialog box.

1. Click  or **Sketch** > **Fillet** > **3 Tangent**. The **Select** dialog box opens.
2. Select a point on a model edge or draft entity as the first fillet reference. The first fillet reference is the start point for the fillet.
3. Select a model edge or draft entity as the second fillet reference. The second fillet reference is the end point for the fillet. The start and end points of the fillet are highlighted.
4. Select a model edge or draft entity as the final fillet reference. The third fillet reference is the midpoint of the fillet.

   Note: If the references are valid, the fillet appears in the drawing and remains selected.
5. Click **OK** in the **Select** dialog box or middle-click to confirm the fillet creation. The fillet is created tangent to the three edges.

Fillet References	Nonparametric Sketching	Parametric Sketching
		

Note: You cannot modify fillets that reference three tangent edges. If you want to make changes to a fillet, you must delete the existing fillet and create a new fillet.

To Create an Ellipse

Using commands in the **Sketch** > **ELLIPSE** menu, you can create ellipses by doing the following:

- Selecting the center, a point to locate the major axis, and an angle that rotates the geometry about the major axis.
- Defining the length and orientation of the major axis and specifying the rotation angle about the major axis.
- Locating the center, a point locating the major axis, and a point locating the minor axis.
- Selecting two points to define the major axis, and a third point anywhere else on the ellipse.
- Using a rotation angle about the major axis. The angle represents what you see of a circle after you have rotated it; the more you rotate it, the less you see of it. For example, if you type 0 degrees, the system creates a circle; if you type 90 degrees, it creates a line.

To Create a Spline

1. Click **Sketch** > **Spline**. The snapping references dialog box opens. Use the arrow icon in the dialog box to add any references you want to set up before you sketch.
2. Click to define a start point and then any number of interpolation points to define the spline.
3. To end the spline at the last selected point, press the middle mouse button.

To Create a Construction Line

Construction geometry entities are lines and circles that you can use to locate and create 2-D draft geometry. They appear on the screen in phantom font, but you can transfer them through IGES files and plot them.

1. Click **Sketch** > **Construction Line**.

- Click **Single** to create a single line.
- Click **Crossed Pair** to create two construction lines perpendicular to each other, intersecting at the pick point.

  The **Snapping References** dialog box opens. If you want to specify an entity to use for an offset or angle base, click the arrow in the dialog box, and then select the entity. The entity is highlighted and added to the list or references.

2. Click a start point for the line. The line will span the display and pass through the pick point.
3. Place the second point of the line freehand, or use the right mouse button shortcut menu **Offset**, **Angle**, **Absolute Coordinates** or **Relative Coordinates**

commands to enter precise centerpoint or angle values, using your selected references.

To Create a Construction Circle

1. Click **Sketch** > **Construction Circle**.
2. Use commands in the **CIRCLE** menu to create a circle.

Converting Views to Draft Items

The **Edit** > **Convert to Draft Entities** command converts drawing views into a collection of draft items that are no longer associated with their corresponding model. During the conversion, the following changes occur:

- All visible geometry, axes, datums, and other entities in the view become draft entities.
- All draft entities that were previously associated with the view become free.
- All attached drawing items such as notes, geometric tolerances, symbols, and draft dimensions are converted to the respective draft entities or notes.
- All visible model dimensions become draft dimensions.
- View dimensions become unassociative draft dimensions.

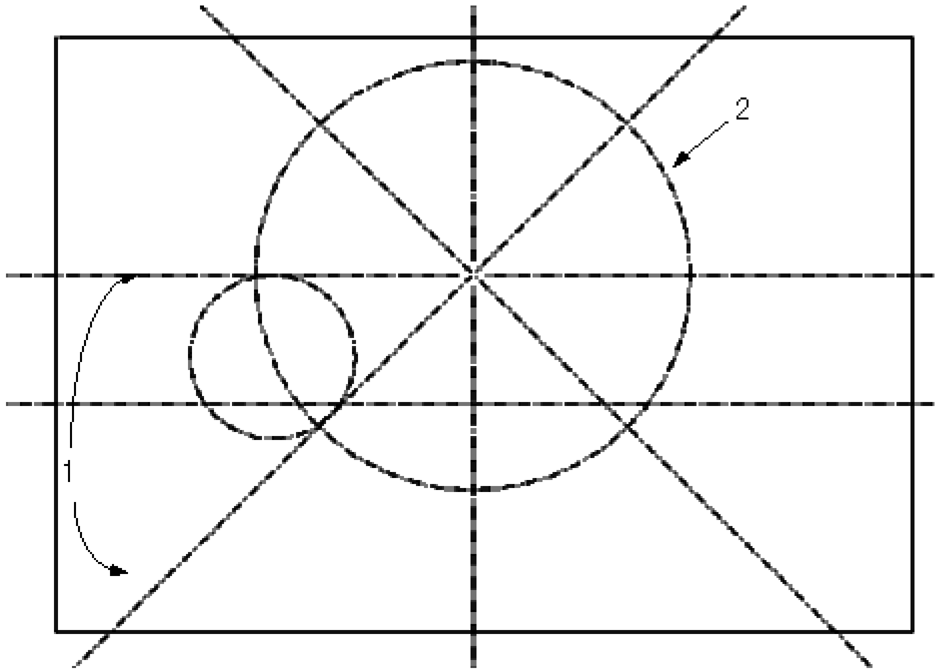
Note: Model entities in a view are converted to the corresponding draft entities as shown in the following table:

Model Entity in a View	Converted Draft Entity
Line	Line
Arc	Arc
Ellipse	Ellipse
None of the above	Spline

Consequences of Converting

After converting a view to a draft entity, Pro/ENGINEER deletes the original view, as well as child views, children on other sheets, and erased children. However, Pro/ENGINEER does not automatically delete the model that is used to generate the view. If required, delete the model manually from the drawing.

Example: Construction Circles



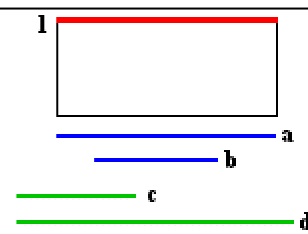
1. This circle is filleted to two lines.
2. This circle is centered in the drawing and passes through the center of the other circle.

Using a Model Edge to Create Draft Entities

About Using a Model Edge to Draft

You can use displayed model edges as the basis for draft entities in your drawings. Using the model edges enables you to modify the display of model geometry while maintaining some associativity when the model changes.

As you sketch the draft entity you can erase some of the model edges that reside behind the newly created draft entities. You can automate the erasure of model edges if you are using parametric sketching. The model edges are erased in the following manner:

(1 is the selected model edge converted to a draft entity) • Edges that are the same size and aligned with the new draft entity are erased (a) • Edges that are shorter than and lie within the endpoints of the new draft entity are erased (b) • Edges that extend beyond an endpoint of the draft	

entity are not erased (c) • Edges that are collinear with the model edge but do have an endpoint behind the draft entity are not erased (d)	

Draft entities that use model edges are automatically related to the view. When you erase the view, all attached detail items are erased with it. However, you can use **Relate View** in **Edit > Group** to remove the draft items and make them independent of the view.

To Create a Draft Entity Using a Model Edge

You can convert existing model edges to draft entities, which enables you to modify the geometry display within the drawing. Any changes you make do not affect the three dimensional geometry in the model.

1. Click **Sketch > Edge > Use**. The **Select** dialog box opens.
2. Select the model edge to use. To select multiple edges, hold down the `CTRL` key and select the edges.
3. Click **OK** in the **Select** dialog box to confirm the edges you selected. The model edges are converted to draft entities. You are prompted to erase the model edges that exist behind the newly created draft entities.
4. Define whether or not to erase the model edges:

- To erase the selected edge, click **Yes**. This replaces the selected edge with the drawn line.

  Note: Edges are not erased when the drawing display is set to wireframe or shading.

- To leave the selected edge, click **No**. The draft entity is copied separately.

5. Drag handles appear on the newly converted draft entities. Modify the draft entities as necessary. If you chose to erase the model edges, applicable model edges are erased when you modify the new draft entity.

Note: When using parametric sketching, you can automatically erase the applicable model edges that exist behind the newly created draft entities. Set the **Erase model edges behind draft entities** option in your sketch preferences (**Sketch** > **Sketcher Preferences**).

Creating Offset Draft Entities

To Create Offset Draft Geometry

Using the **Offset** command, you can create offset draft geometry from other draft entities or model geometry. If the parent object is scalable, for example scalable view geometry, or if parametric sketching is enabled when you create the offset line, the new geometry updates with any changes to the parent geometry.

1. Click the offset icon  on the Sketcher toolbar. The **OFFSET OPER** menu manager opens.
2. Click **Single Ent**. to create offset lines from a single entity. Click **Ent Chain** to select multiple edges from which to offset.
3. Select the entity or entities. (Hold down the CtRL key to select more than one entity. Click **OK** in the confirmation window when finished.) You are prompted for the offset distance in the direction the arrow indicates.
4. Enter the offset distance. Use a negative distance (-) to go opposite the arrow direction. The entity is copied at the offset.

Modifying Draft Entities

To Break a Draft Entity

1. **Insert** > **Break**.
2. Select the entity.
3. Indicate breaks by selecting the start point and endpoint of each break.
4. After the system breaks a draft entity into two separate draft entities, select the first break point on either of the two entities to start another break.

To Rotate Draft Entities

1. Click **Edit** > **Transform** > **Rotate**.
2. Select the items to rotate. Click **OK** in the **Select** dialog box. The **GET POINT** menu opens on the Menu Manager.
3. Use the **GET POINT** menu to select a rotation center. You are prompted to enter the rotation angle in the counterclockwise direction.
4. Enter the angle and press Enter. The Item is rotated.

To Stretch a Draft Entity

If you have created a drawn object using several unassociated lines, you can use the **Stretch** command to temporarily group selected objects and stretch them all in a given direction. Lines that adjoin the selected lines, but are not selected, will stretch to follow the moved lines.

1. Click **Edit** > **Transform** > **Stretch**.
2. Drag a selection box to enclose the entities you want to move. The Menu Manager opens to the **Get Point** menu.
3. Define the translation vector by selecting the first location—the from location; then select the final destination point. The specified entities are stretched.

Note: Filled draft entities cannot be stretched.

To Trim Draft Geometry

You can trim the length of draft entities to better meet a desired drawing display.

Note: Trimming or extended a draft entity so that they extend to another entity should not completely remove any parametric associativity to a model edge. Only the length associativity between the draft entity and the model geometry is removed. Any subsequent update of the length of the model geometry edge does not update the draft entity length.

1. Click **Edit** > **Trim**. A flyout menu appears.
2. Define how you want to trim the draft entity:

- **Divide at Intersection**—Create two entities from one by dividing at an intersection.

  Splitting two draft entities at their intersection does not remove any parametric associativity to a model edge. Each segment is parametrically associated to the underlying edge.

- **Divide by Equal Segments**—Divide a selected line into a number of equal sections. You are prompted for the number of segments. If you select multiple entities, Pro/ENGINEER divides them all the same way into equal segments. After the entity divides, the break points are highlighted.

  Splitting a draft entity in multiple segments of equal length does not remove any parametric associativity with a model edge. Each segment is parametrically associated to the underlying edge. For example, if a draft line was split into N segments, and the underlying model edge is modified and increased by X, each draft segment is increased in the drawing by (X/N).

- **Corner**—Trim two entities to their intersection. Some draft entities may be shortened while others may be extended.
- **Bound**—Trim to a specified entity or point. You first select the boundary, then the items that trim to it.
- **Length**—Trim to a specified length. You enter a length, then select items to trim or expand to that length.

  Note: The draft entity remains collinear with the model geometry it references.

- **Increment**—Trim or extend by a specified increment.

  Note: Type a negative value to trim the draft entity and a positive value to extend the draft entity.

3. Select the draft entity to trim, or extend. The drawing display is updated with the draft entity modifications. Model edges are erased if an overlapping draft entity is trimmed.

Note: You can re-display erased model edges by clicking **Info** > **Show Modified Edges**. Any erased model edges are highlighted . Click **View** > **Drawing Display** > **Edge Display** to re-display the desired edges.

To Change the Line Style of Draft Entities

To change the line style of selected entities as well as redefine some or all line style elements such as line font, color, and line width (thickness):

1. In the menu bar, click **Format** > **Line Style**.
2. Click one of the following options in the **LINE STYLES** menu:
    - **Modify Lines**—Allows you to select draft entities whose line styles you want to modify, using the **Modify Line Style** dialog box.
    - **Edit Styles**—Opens the **Line Style Gallery** dialog box, from which you can create a new line style, edit an existing style, or delete a style.
    - **Edit Fonts**—Opens the **Line Font Gallery** dialog box, from which you can create a new font, edit an existing font, or delete a font.
    - **Clear Style**—Undoes the last line style change you made.

To Modify the Diameter of an Arc or Circle

1. Select the circle or arc and right-click to display the shortcut menu.
2. Click **Edit Diameter Value** from the shortcut menu.
3. Type a new diameter for the draft entity. The entity reappears at its modified value.
4. To update any associated draft dimensions, choose **Edit** > **Regenerate** > **Draft**.

When you modify the diameter of an arc or circle, the system does not update unassociated draft dimensions to reflect changes in the entity size.

To Mirror an Entity

Using the **Mirror** command you can create copies of draft entities, unattached symbols, and unattached notes by mirroring them about a draft line straight entity.

1. Click **Edit** > **Transform** > **Mirror**. You are prompted to select the items to mirror.
2. Select detail items to mirror; then click the middle mouse button. You are prompted to select the line to mirror about. You can select the following entities for mirroring:
    - Drawn line
    - Construction line
    - Draft axes

- Draft datum planes
- Snap lines
- Model datum planes that are perpendicular to the drawing sheet
- Model datum axes
- Model Edges

When you select the line an entity, the mirror copy is created opposite the original, equidistant from the center line.

Note: For mirroring you can select any model edge that appears straight. This includes circular or non-straight edges in a model that have their line of sight perfectly straight in the drawing view.

The following are not a part of straight entities, and hence are not supported as mirroring planes:

- Straight-line entities within dimensions
- Straight-line entities within symbols
- Leader lines on symbols, dimensions, and notes
- GTOL control frames
- Half view lines from Half Views
- Entities in formats

Copying Draft Entities

To Copy and Paste Detail Items

1. Select the detail items you want to copy, and then click **Edit** > **Copy**.
2. Click **Paste**. The Drawing Clipboard dialog box opens and contains the copied detail items.
3. Define the translation origin point by picking a point in the **Drawing Clipboard** dialog box. The translation origin point appears as a yellow square.
4. Select the location on the drawing where you want to place the copied items. This action pastes the copied items, with the transition point at your pick point.

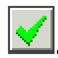
To Make and Place Multiple Copies

1. Click **Edit** > **Transform** > **Translate and Copy**.
2. Select the items to copy and click **OK** in the **Select** dialog box. The **GET VECTOR** menu appears.
3. Select one of the following options to define the translation vector:

- **Horiz**—Enter an offset along the X-axis.

- **Vert**—Enter an offset along the Y-axis.
- **And/Length**—Enter the translation vector in polar coordinates.
- **From - To**—Pick the end points of the translation vector.

  Note: If you select the translation vector **From - To** in the **GET VECTOR** menu, the **GET POINT** menu appears. Click the required commands on the **GET POINT** menu and select the final destination point to translate and copy the specified entities.

4. Enter the number of copies and click . The specified entities are translated and copied.

To Rotate and Copy Draft Entities

1. Click **Edit** > **Transform** > **Rotate and Copy** .
2. Select all of the entities to copy and translate. Click **OK**.
3. Type a rotation angle for the entities and press ENTER. The system measures the angle from the horizontal in the counterclockwise direction.
4. Select the center point for the copy rotation using **GET POINT** menu commands.
5. Type the number of copies to create and press ENTER. The system creates the copies and rotates them.

The GET VECTOR Menu

You can use the **GET VECTOR** menu for various operations, such as translating and copying draft entities. In this case, you use the **GET VECTOR** menu to specify the direction for translating new draft entities.

- **Horiz**—Translates the entities along the horizontal direction. Type a value, in drawing units, to translate the entities. Positive direction is toward the right of the sheet.
- **Vert**—Translates the entities along the vertical direction. Type a value, in drawing units, to translate the entities. Positive direction is toward the top of the drawing sheet.
- **Ang/Length**—Translates the entities at an angle, and a specified distance in that direction, measuring the angle relative to the horizontal in the counterclockwise direction. An arrow appears showing the positive direction of translation.
- **From-To**—Translates the entities along a vector defined by using a start point and endpoint.

Modifying a Spline

About Modifying Splines

You can modify splines in the following ways:

- Move a single point or a range of points on the spline
- Add points to the interior of the spline, or extend the spline by adding points to the exterior
- Delete points from the spline
- Use a deviation value to reduce the number of spline points
- Smooth the spline
- Use the spline control polygon to modify the spline

To Move a Single Spline Point

1. Select the spline, and then right-click to display the shortcut menu.
2. Click **Edit Spline** from the shortcut menu. The **MOD SPLINE** and **MOVE PNTS** menus appear in the Menu Manager.
3. To move a single point on the spline, click **Single Pnt**, and then select the point you want to move. The point and the part of the spline that is affected moves as you move the mouse pointer to another location on the screen.
4. Click in the new location to place the point.
5. Click **Done Modify** when you are finished. To cancel the move operation, click **Quit Modify**.

To Move a Range of Points on a Spline

1. Select the spline, and then right-click to display the shortcut menu.
2. Click **Edit Spline** from the shortcut menu. The **MOD SPLINE** and **MOVE PNTS** menus appear in the Menu Manager.
3. Select two spline points limiting the adjustment range.
4. Select the point (within the adjustment range) you want to move.
5. To place the point in a different location, click another location on the drawing window. The selected point moves to its new location, and the part of the spline enclosed within the range adjusts accordingly.

To Add Points to a Spline

You can add points to the interior of a spline, or to the exterior, thus extending the spline.

1. Select the spline, and then right-click to display the shortcut menu.
2. Click **Edit Spline** from the shortcut menu. The **MOD SPLINE** and **MOVE PNTS** menus open in the Menu Manager.
3. In the **MOD SPLINE** menu, click **Add Points**. **Interior** is selected by default in the **NEW POINTS** menu.
4. Do one of the following:
    - Click **Interior** and add points to the interior of the spline by selecting any location on the spline between any two existing points.
    - Click **Exterior** and extend the spline by adding points beyond its current endpoints. Select the spline endpoint to extend; then select additional points.

To Modify a Spline's Controlling Polygon

You can modify a spline by using its control polygon (the polygon that the system creates to surround the spline when you first create it). The control polygon displays in white when you choose the **Control Poly** command.

1. Select the spline, and then right-click to display the shortcut menu.
2. Click **Edit Spline** from the shortcut menu. The **MOD SPLINE** and **MOVE PNTS** menus open in the Menu Manager.
3. Click **MOD SPLINE** > **Control Poly**.

   The spline’s control polygon appears in white. Spline control points appear with line segments between them on the spline. The line segments begin and end at the startpoint and endpoint of the spline, and intermediate segments remain tangent to the spline. These line segments are visual aids for modifying the shape of the spline.

   Note: You cannot use this command on a spline that has a tangency condition defined for one end only.
4. Adjust the shape of the spline by selecting a point on the control polygon and dragging it to a new location. You cannot select endpoints. The spline rubber-bands to its new shape as you move the point.
5. Place the control point by pressing the left mouse button.

To Delete Points from a Spline

1. Select the spline, and then right-click to display the shortcut menu.
2. Click **Edit Spline** from the shortcut menu. The **MOD SPLINE** and **MOVE PNTS** menus open in the Menu Manager.
3. Click **MOD SPLINE** > **Delete Points**.
4. Select a spline point to delete. Click **OK**, then **Done Modify**, when you are finished removing points.

The system redraws the spline according to the new shape created when you removed the points. Repaint the drawing to see the new spline shape.

To Decrease Spline Points Automatically

1. Select the spline, and then right-click to display the shortcut menu.
2. Click **Edit Spline** from the shortcut menu. The **MOD SPLINE** and **MOVE PNTS** menus open in the Menu Manager.
3. Click **MOD SPLINE** > **Sparse**.
4. Type a deviation value to redraw the spline with fewer points. This value must be a positive number. The system highlights the spline and tells you how many points it is going to remove.
5. To accept the changed spline, choose **Accept**; to reject it, choose **Reject** and type a different deviation value. (For each spline, you might need to try several different deviation values before achieving the desired result.)

To Smooth the Spline

1. Select the spline, and then right-click to display the shortcut menu.
2. Click **Edit Spline** from the shortcut menu. The **MOD SPLINE** and **MOVE PNTS** menus open in the Menu Manager.
3. Click **MOD SPLINE** > **Smooth**.
4. Type the number of points to average in order to smooth the spline:
   - Type [1] to keep the spline as is.
   - Type [3] to average the centermost point out of the group and one point on each side of it.
   - Type [5] to average the centermost point in the group and two points on each side of it.
5. The resulting spline appears in green. To accept it, choose **Accept**; to reject it, choose **Reject** and try again with a different number of points.

Working with Drafted Cross Sections

About Working with Draft Cross-Sections

You can use **Edit** > **Hatch/Fill** and then use the **Hatch** or **Fill** commands in the **MOD XHATCH** menu to fill or hatch a closed draft shape that you are using to represent a part or an assembly 2D cross-section. Pro/ENGINEER defines a draft cross-section boundary by a closed loop of draft geometry.

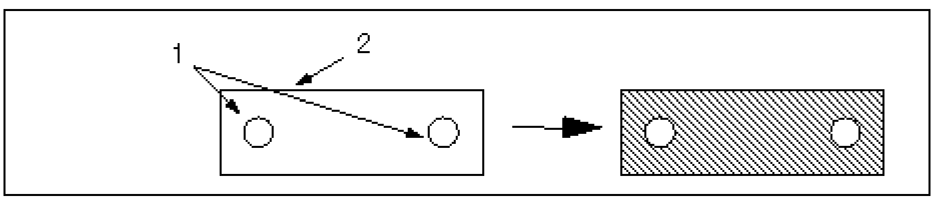
When working with draft cross-sections, remember the following points:

- You can define unfilled or uncrosshatched areas within the closed loop.

- Pro/ENGINEER does not consider a draft cross section to be a single entity. Therefore, to place it on a layer, you must add the line and crosshatch pattern individually.
- You can modify only one draft cross-section at a time.
- You can delete either the entire cross-section (with the crosshatching or filling and boundary) or only the crosshatching or filling.
- You can hide or unhide crosshatch patterns within cross-sections using the **View** menu.
- Draft entities that define crosshatched or filled areas should not intersect each other or the outer boundary of the section.

To Create a Draft Cross-Section or Filled Area

1. Select a closed draft entity to which you want to add cross hatching or solid fill. A drafted cross-section may have "islands" within the hatched area that will not be hatched.



1. Islands

2. Outer boundaries

Note: If draft entities that create a cross-section boundary overlap with each other, the resulting hatch or fill may be incorrect.

2. Click **Edit** > **Hatch/Fill** or right-click and click **Hatch/Fill** on the shortcut menu.
3. Pro/ENGINEER displays a default cross-section name in the format XSEC<nnnn>, where <nnnn> is a number series unique to a drawing. Change this name if required.
4. Press ENTER. The draft cross-section is created. The **MOD XHATCH** menu appears.
5. Click **Hatch** to fill the draft cross-section with crosshatches or click **Fill** to fill the draft cross-section with solid color.
6. Use the menu manager to set the spacing, color, and line style for the crosshatch.

Use **Add line** to add another hatch line pattern at a different angle to the original. Use **Delete line**, **Prev line**, and **Next line** to select and delete hatch line patterns.

Use **Spacing**, **Angle**, **Color**, **Offset**, and **Line Style** to edit properties of selected hatch patterns.

7. Click **Done** on the **MOD XHATCH** menu when you are finished.

Note: By default, all new crosshatches are displayed in Letter color, that is the system default color. Use the **Line Style** command in the **MOD XHATCH** menu to change the color of the crosshatches. Use the **Color** command in the **MOD XHATCH** menu to change the color of the filled section.

To Modify a Draft Cross-Section

1. Select the cross-hatching.
2. Click **Edit** > **Properties**. The **MOD XHATCH** menu opens in the Menu Manager.
3. Click commands from the **MOD XHATCH** menu. You can retrieve a previously saved crosshatch pattern to use in the draft cross section.

To Modify the Color of Filled Crosshatches

1. Select a filled crosshatch.
2. Click **Edit** > **Properties** or right-click and select **Properties** from the shortcut menu. The **MOD XHATCH** menu opens in the Menu Manager.
3. Click **Line Style**. The **Modify Line Style** dialog box opens.
4. Click **Color**.
5. Define a new color using the options available in the **Color** dialog box.
6. Click **Apply**. Pro/ENGINEER displays the selected crosshatch with the specified color.

To Delete a Crosshatched or Filled Area

1. Select the crosshatched area you want to delete, and then click the right mouse button to display the shortcut menu.
2. Click **Delete** from the shortcut menu. The system deletes the cross-hatching or filling, leaving the bounding entities.

Grouping Draft Entities

About Grouping Detail Objects

Use the **Edit** > **Group** commands to create groups of detail objects that move together as one object.

It is important to note that drafting lines use a different grouping routine than do other detail objects like notes or GTOLs.

- To group drafting lines with other drafting lines, use the **Edit** > **Group** > **Draft Group** commands.

- To group other detail objects with each other, for example, to group a note with a GTOL, use the **Edit** > **Group** > **Relate to Object** commands.
- To group any object with a view, including drafting lines, use the **Edit** > **Group** > **Relate to View** commands.

Drafting object groups work like groups in any other graphics program, once the items are grouped, selecting one object selects the group, and they all move at the same time.

By contrast, when you create groups using the **Relate to Object** or **Relate to View** commands, you create one "parent" to which the other objects are related. When the parent moves, the related objects move, however, you can move the related objects separately.

Relating Draft Lines and Views

You can automatically relate draft entities to views, which ensures the draft entities move and rescale with the view, while maintaining their location relative to that view. You can associate notes (without leaders), symbols, and geometric tolerances with a draft view in addition to draft geometry.

By default, Pro/ENGINEER does not associate all draft lines with views. When you can set a drawing view to be the current draft view, Pro/ENGINEER associates all new draft entities with that view. You can regard the related objects as a group.

However, some draft entities are automatically related to the view that the model geometry belongs to; including fillets, chamfers, drafts that use edges, and trimming done at intersections and by equal segments. The relation of these draft entities to their respective views overrides any other default draft view setting.

Dimensions for draft entities related to views are provided in model units and adhere to the drawing scale. If you relate an existing draft entity, it is updated to display in the model units. Dimensions not related to a view display according to the drawing units and adhere to the drawing scale.

To Relate Draft Entities to Views

- Sketch draft entities; select desired draft entities to relate and click **Edit** > **Group** > **Relate to View**.
- Select a view and click **Edit** > **Group** > **Set Default Relate View**. Sketch the desired draft entities. The sketched draft entities will automatically be related to the selected view. You should unset the group when you are finished sketching.
- Parametric sketching automatically relates draft entities to a view.

To Unrelate Draft Entities from Views

- Select desired draft entities and click **Edit** > **Group** > **Un-relate**.
- Click **Edit** > **Group** > **Unset Default Relate View** when are finished sketching.

Note:

- A draft entity related to a view may not be switched to another sheet. You cannot switch sheets if items belonging to a view other than the background view are snapped to a draft entity snap line.
- Trimming (**Edit** > **Trim**) a draft entity by corner, bound, length, or increment, does not affect its relation to a view. The related view remains the same.

To Create a Draft Group

1. Click **Edit** > **Group** > **Draft Group**. The **Draft Group** menu appears.
2. Click **Create** and select the draft items to add to the group.
3. Middle-click to complete the selection. You are prompted for a group name.
4. Type a name for the group and press ENTER. At this point you can either select draft items for another group, or middle-click to complete the creation of the draft group.

To Ungroup Items from a View

1. Select the items to remove from the group
2. Click **Edit** > **Group** > **Unrelate**. The selected items are un associated from the view.

To Set a Drawing View as the Current Draft View

1. Click **Edit** > **Group** > **Set Default Relate View**.
2. Select a view to be current. The system sets the selected view as the current view.

To Relate Draft Items to a View

1. Select the draft items to relate to a view
2. Click **Edit** > **Group** > **Relate to View**.
3. Click the view to relate to. The items are related to the view.

   To remove an item from the related group, use **Edit** > **Group** > **Unrelate**

To Suppress a Draft Group

1. Click **Edit** > **Group** > **Draft Group**. The **Draft Group** menu opens on the Menu Manager.
2. Click **Suppress**. The **Group Access** menu opens on the Menu Manager.
3. Using the **Group Access** menu, do one of the following:
   - Click **Select** and select a group.

- Click **By Name** and click the group name.

The selected group is suppressed. Use the **Resume** command to unsuppress it.

To Resume a Suppressed Group

1. Click **Edit** > **Group** > **Draft Group**. The Menu Manager opens.
2. Click **Resume**.
3. Select the group to resume from the **GROUP NAMES** menu.

To Ungroup a Draft Group

1. Click **Edit** > **Group** > **Draft Group**. The **Draft Group** menu opens on the Menu Manager.
2. Click **Explode**. The **Group Access** menu opens on the Menu Manager.
3. Using the **Group Access** menu, do one of the following:
   - Click **Select** and select a group.
   - Click **By Name** and click the group name.

   The selected group is ungrouped.

To Modify a Draft Group

1. Click **Edit** > **Group** > **Draft Group**. The **Draft Group** menu appears.
2. Click **Edit.** The **GROUP ACCESS** menu appears. Select one of the following:
   - Click **Select** to graphically select a draft group with draft items.
   - Click **By Name** and select the group name.
3. Click **Add** or **Remove** on the **EDIT GROUP** menu to add or remove draft items from the selected group. Middle-click to finish.

To Group Objects with Dimension Text

You can group a note, draft geometric tolerance, or symbol with dimension text so that it moves with the dimension when the dimension changes location. When you erase, redisplay, or delete the dimension, the system also does the same to all items related to it.

1. Select the item to relate to a dimension.
2. Click **Edit** > **Group** > **Relate to Object.**
3. Select the dimension. The item is related to the dimension.

Use **Unrelate** to remove the relationship.

To Return Items to a Draft Group

1. Select the draft items that have been removed from a draft group in the current session.
2. Right-click and click **Return to Draft Group** on the shortcut menu. The draft items that were removed from their draft groups in the current session are returned.

   Note: The **Return to Draft Group** command is available on the shortcut menu only when you remove draft items from their draft groups in the current session.

Managing the Draft Environment

Setting the Draft Scale

About Setting Draft Scale

By setting a draft scale using the drawing setup file option `draft_scale`, you can create draft entities with a scaled size. If you set it at 0.5, any entities that you create subsequently are one-half the size that you specified. For example, creating a horizontal line of 4 inches (by selecting one endpoint and placing the second endpoint using **Rel Coords** with x=4 and y=0), the line measures as 2 inches on paper, but dimensions as 4 inches. If you change the draft scale after creating and associatively dimensioning draft entities, the dimensions change.

The draft scale, like other parameter values in the drawing setup file, is completely independent of a drawing scale. When the drawing size changes, the value of the draft scale does not change. The system scales the draft geometry by the proportion necessary to maintain its relative location on the sheet, and it appears to be the same size that it was before. The geometry is, in fact, a different size (because the sheet is a different size), and this is reflected by any new dimensions that you create for it; the ratio of the new dimension to the previous one is equal to the ratio of the size of the new sheet to the old one. If you set the drawing setup file option `associative_dimensioning` to `yes` (the default), when the size of dimensioned draft entities changes, the system updates the dimensions to the latest value).

Understanding the Draft Scale

Draft scale is the relation between what an entity size appears to be versus its actual dimension. For example, assume that an edge of a part is 4 inches long:

- If the draft scale is 1.0, the entity appears to be 4 inches in the drawing.
- If the draft scale is 4.0, the entity appears to be 16 inches in the drawing (it appears to be four times larger than it actually is).
- If the draft scale is 0.5, the entity appears to be 2 inches in the drawing (it appears to be one-half the size it actually is).

(Use the **File** > **Properties** > **Drawing Options** dialog box to change `draft_scale` to determine the default draft scale.)

To Scale Draft Geometry

1. Click **Edit** > **Transform** > **Rescale**.
2. Select draft geometry and dimensions to scale, and then click **OK**.
3. Select the point about which to scale the entities.
4. Type a scale value and press Enter. A number greater than 1 makes the entities larger; a number less than 1 makes them smaller. For example, a value of .25 scales the drawing to one-quarter (1/4) of the actual size of the model.
5. Regenerate, if necessary, to recalculate associative dimensions.

Defining Line Fonts

About Line Fonts

You can create and customize the fonts used for your two-dimensional draft lines. A line font is a pattern of dashes and spaces that are proportionate within the specified length. For example, 5 dashes and 5 spaces with a pattern length of 1 inch results in two dashes and spaces for a 2-inch line in the drawing. The same pattern definition with a 1/4-inch pattern length results in 8 dashes and spaces for a 2-inch line in the drawing. You can have a maximum of 16 dash-space combinations in a line font file.

User-defined line font files, `<fontname>.lsl`, must reside in your local directory for you to be able to use them. When you retrieve a drawing, and the system cannot find a line font file, it displays any line that has been set with that style as solid and notifies you that it has not loaded a font file.

If you have specified auxiliary font files in the drawing setup file (this occurs automatically when you create the font when that drawing is active), it identifies the font files that it could not find.

Note:

- You can set a default length for the line font using the `line_style_length` drawing setup file option.
- You can import and export IGES line font defintions using the `iges_in_dwg_line_font` (Import) and `iges_out_dwg_line_font` (Export) configuration options.

To Create a New Line Font

You can create your own line fonts by specifying the font length and pattern.

1. Do one of the following:
   - Click **Format** > **Line Style**. The **LINE STYLES** menu appears in the Menu Manager. Then, choose **Edit Fonts** from the **LINE STYLES** menu.
   - Click **Format** > **Line Font Gallery**.

   In either case, the **Line Font Gallery** dialog box opens.

2. Click **New**. The New Line Font dialog box opens.
3. Type a name in the **New Name** box, select an existing font to copy from the **Font** list, or click **Select Line**.
4. Specify the font length by typing the length of one unit in the **Unit Length** box.
5. Specify the font pattern by typing a series of dashes (-) and spaces ( ) in the **Font Pattern** box.
6. Click **OK** to accept the font as currently defined.

To Modify a Line Font

1. Do one of the following:
   - On the menu bar, click **Format** > **Line Style**. The **LINE STYLES** menu appears in the Menu Manager. Then, choose **Edit Fonts** from the **LINE STYLES** menu.
   - On the menu bar, click **Format** > **Line Font Gallery**.

   In either case, the **Line Font Gallery** dialog box opens.
2. In the **Line Font Gallery** dialog box, select an existing font from the scrollable **Fonts** container.
3. Click **Modify**.
4. In the **Modify Line Font** dialog box, make the necessary modifications to the selected font:
   - Change the length by typing a new value in the **Unit Length** box.
   - Change the font pattern by typing a series of dashes (-) and spaces ( ) in the **Font Pattern** box.
   - The default font length can be set with `line_style_length` drawing setup file option. Any changes you make to the line length will override that option.
5. Click **OK** to accept the modifications.

Deleting Line Fonts

You can not delete additional line font from a `DTL` file because fonts might also be used within drawing notes. Instead, you should replace the old font name with a different font. The integer before the font name serves as a cross-reference between the geometry and the line font, enabling you to make blanket changes to fonted geometry. For example:

Within the `aux_line_font` drawing setup file option, replace the value of from `100 dash-1` to `100 solidfont` changes all geometry that was originally `dash-1` to `solidfont`.

Defining Line Styles

About Creating and Modifying Line Styles

You can define and save a line style by using the following elements:

- A line font (a pattern of solid, dashes, spaces, or dots, or a combination of all of them)
- A width for the font
- A color

You can apply line styles to table grids, symbols, axes, draft entities, cosmetic features, and model edges. Model edges can be regular, silhouette, or non-analytical silhouette edges. In the Part mode, user-defined line fonts appear in solid font.

Note:

- You cannot modify the line style of quilt edges.
- Pro/ENGINEER retains modified edge attributes and maintains the line style that you previously assigned only when you modify geometry in such a way that the edge ID remains the same. If you change the edge ID, the line style changes are not propagated to the new edge.
- For silhouette edges that overlap with each other, ensure that you select all the edges before modifying their line style.
- In case of multiple edges that are stacked together, select all the stacked edges using **Pick from List** on the shortcut menu before modifying their line style.

To Assign a Line Style

1. Select the line you want to modify.
2. Click **Format** > **Line Style**. The **Modify Line Style** dialog box opens.

- To assign a line style to entities and model edges, select a line style from the **Style** list and click **Apply**.
- To set the line font, select a line font from the **Line Font** list and click **Apply**.
- To set the width, type a value in the **Width** box and click **Apply.** This text box is only available in the Drawing mode. It does not apply when modifying the line style of part entities in any other mode.
- To set the color, click **Color** and select a color from the **Color** dialog box. You can define a new color if required.

3. For model edges, select one of the **Model Edge Options** to assign the line style.
4. Click **Close** to finish.

To Create a New Line Style

1. On the menu bar, click **Format** > **Line Style Gallery**. The **Line Style Gallery** dialog box opens.
2. Click **New**. The **New Line Style** dialog box opens.
3. Type a name in the **New Name** box, select an existing style to copy from the **Style** list, or click **Select Line**.
4. Set the line font, width, and color.
5. To accept the new line style as currently defined, click **OK**.

To Modify a Line Style

You can modify line thickness and color for individual or group selections of drafted objects and model edges.

1. Click **Format** > **Line Style** and select one or more drafted objects or model edges to modify. Alternatively, select one or more drafted objects or model edges, right-click, and click **Line Style** on the shortcut menu. The **Modify Line Style** dialog box opens.
2. Use the **Copy From** options to set an existing style for the selection, or to copy a style from a line you select in the drawing. Use the **Attributes** options to set a combination of line font, width, and color.
3. For model edges, select one of the following options to apply the line style changes:
    - **Selected edges only**—Line style changes are applied only to selected model edges.
    - **All occurrences in sheet**—Line style changes are applied to selected model edges in all the visible views on the sheet.

      Note:

      You cannot use **All occurrences in sheet** for model edges in erased views.

      If you modify the display style of an edge such that the edge is erased or hidden and then apply a line style to the edge, line style changes will not be visible as long as the edge is erased or hidden.
4. Click **Apply** to modify the line style.

To Specify the Default Line Style Setting

1. On the menu bar, click **Format** > **Default Line Style**. The **SEL STYLE** menu appears in the Menu Manager.
2. On the **SEL STYLE** menu, choose one of these commands:
    - **Hidden**—Displays as hidden line (gray) geometry on the screen, and plots as dashed lines.

- **Geometry**—Displays as regular visible geometry (white) on the screen, and plots as solid lines.
- **Leader**—Displays as dimensions (yellow) on the screen, and plots as yellow lines.
- **Cut Plane**—Displays as white phantom lines on the screen, and plots as phantom lines with Pen 1.
- **Phantom**—Displays as gray phantom lines on the screen, and plots as phantom lines with Pen 3.
- **Centerline**—Displays as yellow centerlines on the screen, and plots as centerlines.

To Create a Custom Line Color

1. Select the item to modify
2. Click **Properties** from the right mouse button shortcut menu.
3. On the menu bar, click **Format** > **Line Style**. The **LINE STYLES** menu appears in the Menu Manager. The **Modify Line Style** dialog box opens.
4. In the **Modify Line Style** dialog box, click **Color**.
5. In the **Color** dialog box, click **New**.
6. In the Pro/ENGINEER Color Editor tool, using the left mouse button, define the new color. Move the RGB **controls** from right to left incrementally; then click on the color bar (or click **OK**) at the desired setting. The new color appears in the **User-defined Colors** box.
7. Click **Apply**. The system displays the selected item with the specified color and stores the color with the model.

To Delete a User-Defined Line Style

1. Click **Format** > **Line Style**. The **LINE STYLES** menu appears in the Menu Manager.
2. Click **Edit Styles**. The **Line Style Gallery** dialog box opens.
3. Select an existing line style from the **Styles** list.
4. Click **Delete**.
5. Click **Yes** when prompted for confirmation. Pro/ENGINEER removes the line style.

Markups and Overlays

Markup Mode

About Markups

A markup is an object that, like a set of transparent sheets on top of a drawing sheet, enables you to superimpose text and sketched entities in a variety of colors to indicate where changes may be required.

A markup is an informal sketch that you can create within Pro/ENGINEER with text superimposed over any object.

The basis for the markup is the object that you select to mark up. To create a markup, you use an object in the mode in which it was created:

- Part (includes all objects with a .prt extension).
- Assembly (includes all objects with an .asm extension).
- Drawing
- Manufacturing
- Layout
- Report
- Diagram

The object accompanies the markup file, which uses the file extension .mrk. **Note**: The **Orientation**, **Model Display**, and **Advanced** commands are not available in the Pro/ENGINEER **View** menu after you create a new markup object. You must orient your three-dimensional object before you create it. The system saves the orientation with the markup.

Each markup is like a separate transparent sheet laid over the object and contains sketched entities and/or notes in a single color.

During the markup process, the system does not change the object in any way, and saves all sketched entities and notes created in Markup mode with the markup file, not with the object. Since the driving object does not change, the system does not store it when it stores the markup.

You can also create a markup for a multisheet drawing, continuing over all of the sheets.

Note the following rules of operation for Markup mode:

- With assemblies, the following applies:
  - When you create an exploded view for a markup item, it does not affect the exploded status of the views in other items.
  - You can simultaneously see only those markup items that have the same explode status, explode dimensions, and view orientation.

- You can modify explode dimensions by using **Mod Explode** in the ENTER MARKUP menu.

- The system displays three-dimensional objects with their colors as well as drawing colors.
- A markup file must reside in the directory of the object that the markup references.
- Layout dimensions appear as symbolic.

When doing a comparison with three-dimensional markup items using the **Markup**, **Setup**, and **Show** commands, you can toggle between **Seen** and **Unseen** only with those items that have exactly the same view as the current markup item.

To Create a Markup

1. Click **File** > **New**.
2. In the **New** dialog box, click **Markup** and **OK**.
3. From the **Markup** dialog box, select or retrieve an object.
4. Type a name for the object you are creating. The default name is your user ID.
5. Click any of the following commands from the **MARKUP** menu to mark and annotate the object:
   - **Setup**—Changes the color of the markup, creates a plot of it, toggles its display, or changes to another sheet of the markup and drawing. The **MRKP SETUP** menu displays the following commands:

     Color—Changes the color of the markup. The default markup color is red.

     Show—Lists the markups in the current working area along with their color. A check mark appears next to the markup's name and color if it is currently displayed. Choosing the markup name again removes the check mark and turns off its display.

     Switch Sht—lets you view and mark other sheets of the drawing.

     Text Height—Sets the text height before creating notes.

     Line Width—Sets the width of sketched entities before creating them.
   - **Note**—Places a note without a leader.
   - **Arrow**—Sketches an arrow. Use the left mouse button to select the point to which the arrow points; then select the point for the other end of the arrow. Press the middle button to abort arrow creation.
   - **Curve**—Selects the points of a curve to which a spline is to be fitted. Use the left mouse button to select the point, the middle button to stop the curve, and the right button to abort curve creation.

- **Sketch**—Sketches a curve without selecting points for it. Hold the left mouse button down to create a free curve. Release the left button to end it.
- **Line**—Creates a line using the left mouse button.
- **Move**—Moves a note, curve, or line.
- **Modify**—Changes the note text or line width of a markup item.
- **Delete**—Deletes a markup item (note, arrow, curve, sketch, or line).

Note: The Once you have chosen **Note**, **Arrow**, **Curve**, or **Sketch**, you can continue creating that type of entity until you choose another command.

Drawing Overlays

About Using Drawing Overlays

Using overlays, (**Tools** > **Overlay**) you can superimpose selected views or an entire sheet of one drawing over the current drawing sheet. This functionality is available for drawings, layouts, reports, and diagrams; each can reference objects of these four types, as well as formats.

When working with overlays, keep in mind the following:

- Overlays are *read-only* in the current drawing.
- The system updates overlays to changes in the source drawing.
- Overlaid views appear with all detail items.
- You cannot select overlays for any drawing procedure.
- If the size of the current drawing is different from the size of the source drawing, the overlays brought into the current drawing maintain the same screen size (that is, they occupy the same portion of the graphics window) as they had in the source drawing.

To Create an Overlay

1. Click **Tools** > **Overlay**. The menu manager opens.
2. Click **Add Overlay** > **Place Views**.
3. Position the overlay by doing one of the following:
    - If the source drawing has only *one* visible view on each of the sheets, position the origin by using the **GET POINT** menu to select a new location. Press the middle mouse button or choose **Done** to finish.
    - If the source drawing has *more than one* view on any sheet, choose **Place** > **Views** > **Select Views** and select the desired views. (If the view that you want to use as an overlay is on another sheet, choose **Change Sheet** and select the required sheet). Locate the origin of the main view by selecting a point on the screen. To accept the current position of the view

origin, press the middle mouse button or choose **GET POINT** > **Done**. The system locates all of the dependent views correspondingly.

To Overlay a Drawing onto the Current Drawing

1. Click **Tools** > **Overlay**. The Menu Manager opens.
2. Click **Add Overlay** > **Place Sheet**.
3. If the source drawing has only one sheet, the system confirms it and overlays the source drawing onto the current sheet.

   Otherwise, if it has more than one sheet, type the appropriate sheet number. If you do not remember the required sheet number, type [?] to display the source drawing in the subwindow. Find the sheet using commands in the PLACE SHEET menu; then choose **Done**. The specified sheet overlays the current drawing sheet.

To Delete an Overlay

1. Click **OVERLAY DWG** > **Del Overlay**.
2. Do one of the following:
   - Click **Sel Overlay** to specify an overlay to delete by selecting the source drawing object from the SEL OVERLAY menu.
   - Click **All Overlays** to delete all overlays.

To Move an Overlay

1. Click **OVERLAY DWG** > **Move Overlay**.
2. From the **SEL OVERLAY** menu, select an overlay name or select it on the screen.
3. From the **SEL OVLY ITM** menu, select a view. Specify the translation vector by using commands in the **GET VECTOR** menu.

   Note: You can move an overlay by choosing **OVERLAY DWG** > **Move Overlay**; however, you cannot move *overlaid sheets*.

Draft and Model Grids

About Using Model and Draft Grids

When you are working with a drawing, you can use a three-dimensional (model) grid that was defined in Part or Assembly mode or a two-dimensional grid. To use model grids, click **View** > **Drawing Display** > **Model Grid**, and then use the **Model Grids** dialog box. To use draft grids, click **View** > **Draft Grid**, and then use the **GRID MODIFY** menu in the Menu Manager.

To Create or Modify a Model Grid

You can create a model grid in Part or Assembly mode by choosing **Grid** from the **SET UP** menu and then selecting a coordinate system to define the origin of the grid from the **Model Grids** dialog box. In an assembly, this coordinate system must belong to the top-level assembly.

To create a model grid in Drawing mode:

1. On the menu bar, click **View** > **Drawing Display** > **Model Grid**. The **Model Grids** dialog box opens.
2. Use the various options on the **Grids** and **Settings** tabs to create the model grid.

To modify the model grid, follow the same procedure.

To Display a Model Grid in a Drawing

When in a drawing, you can use commands in the **DWG MDL GRID** menu to display and erase the model grid. To display it, you must orient the views so that one axis of the coordinate system is normal to the screen and one axis is parallel to a drawing sheet border.

1. Click **View** > **Model Grid** > **Show Grid**. The **Model Grids** dialog box opens.
2. Using the **Grids** and **Settings** tabs, specify the display.

   Note: If you add a view to a sheet that has a sheet grid, the view's model has a defined grid origin, *and* the view is in planar orientation to this grid, the system moves the view origin to the grid origin and snaps that point to the grid point of the sheet.

To Show Model Grid Balloons

You can show model grid balloons by using the **Model Grids** dialog box or by using the configuration options `model_grid_balloon_size`, `model_grid_neg_prefix`, `model_grid_num_dig_display`, and `model_grid_offset`.

You access the **Model Grids** dialog box by clicking **View** > **Model Grid**.

- To specify the default grid balloon size, use **Setup Options** in the **Model Grids** dialog box, or set `model_grid_balloon_size`.
- To control the prefix of negative values in grid balloons, use the **Negative Prefix** option in the **Model Grids** dialog box, or set `model_grid_neg_prefix`. The default value is "-."
- To set the number of digits displayed in grid balloons, use the **Decimal Places** option in the **Model Grids** dialog box, or set `model_grid_num_dig_display`.

To specify the offset of new model grid balloons from the drawing view, set `model_grid_offset`. If you set it to `default` (the default value), the system offsets new model grid balloons from the drawing view by twice the current model grid spacing. If you specify a number as the value, the system offsets balloons by that number of inches (not drawing units) from the view.

To Erase Balloons

When you erase a balloon, you blank it from the display. The balloon still exists in the background. You can redisplay the balloon using the **Show/Erase** dialog box, accessed by clicking **View** > **Show and Erase** in the menu bar.

1. Select the balloon(s) you want to erase, and then right-click anywhere in the drawing window to display the shortcut menu.
2. Click **Erase**. The selected balloons display in grey, indicating that they have been erased from the display.

Note: If you have erased a balloon in error, you can undo the erase operation immediately if you have not yet clicked another place in the drawing window. While the grayed-out version of the balloon still exists on the screen, right-click and select **Unerase** from the shortcut menu. The balloon becomes highlighted in magenta; if you click anywhere else in the drawing window, the balloon then displays in the normal system color. The **Unerase** option is not available if you have repainted or regenerated the display, or if you have clicked another place in the drawing window after erasing the balloon.

To Erase a Model Grid from a Drawing

1. On the menu bar, click **View** > **Model Grid**. The **Model Grids** dialog box opens
2. Do one of the following:
    - To erase the grid by view, click **View** and then select the view. Click **Erase**. The system erases the grid but does not permanently remove it from the model.
    - If you want to erase a grid line, click **Line** and then select individual lines of the grid. Click **Erase**.
    - If you want to erase by sheet, click **Sheet**. The system will ask you if you want to erase the grid on the current page. Accept the default response `yes`.
3. To redisplay individually erased lines, click **Show** > **View**.

To Modify the Grid Size

1. On the menu bar, click **View** > **Model Grid**. The **Model Grids** dialog box opens.
2. If the system displays the grid on individual views, select the view that you want to change.
3. Under **Spacing**, Type a new spacing for the grid. The system updates the grid automatically in all views in which it appears.

Modifying Model Grid Size

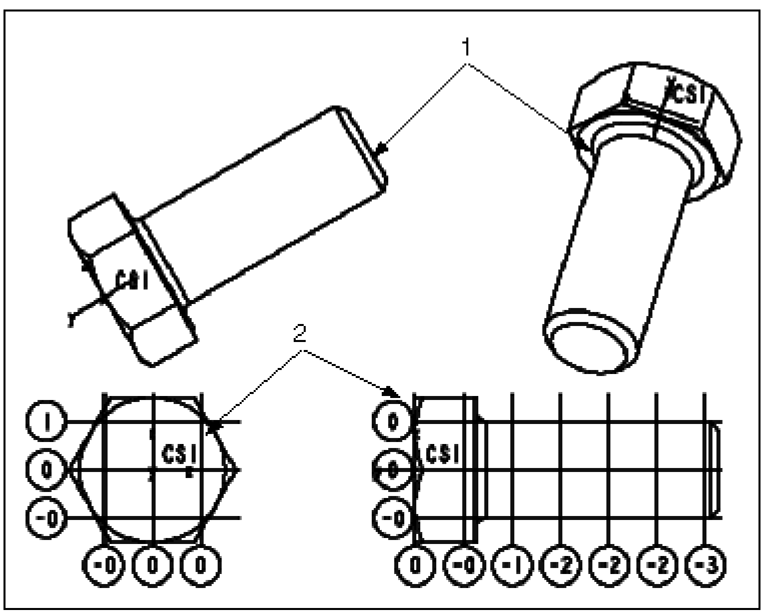
You can set the default model grid spacing by specifying the value for the configuration file option `model_grid_spacing` in drawing units. You can also use the **Spacing** options in the **Model Grids** dialog box (**View** > **Model Grid**).

Considerations When Using the 3-D Model Grid

When using the 3-D model grid, keep in mind the following:

- If you reorient a view with a grid, the system may erase the grid from the display. Therefore, before completing the procedure, you must confirm that you want to reorient the view.
- When you change the view scale, the model grid adjusts to cover the entire view.
- The system extends the grid lines beyond a drawing view outline at a distance that equals two default grid spaces. It uses the default grid when the model grid first appears. You can modify it later, but the extension distance does not change.

Example: Model Grid



1. Invalid views for model grid.
2. Model grid shown by view.

The Model Grids Dialog Box

Create and modify model grids by using the **Model Grids** dialog box. To access the dialog box in Drawing mode, click **View** > **Model Grid** on the menu bar. In Part or Assembly mode, click **Set Up** > **Grids** in the Menu Manager.

The **Grids** tabbed page contains the following options:

- **Origin**—Allows you to select the coordinate system for the grid origin.
- **Display**—Allows control over the grid display.
    - **Show**—Show the grid and grid labels by view, line, or sheet.
    - **Erase**—Erase the grid and grid labels by view, line, or sheet.
    - **Show/Erase by**:

      View—Show or erase the grid in selected views.

      Line—Show or erase individual grid lines.

      Sheet—Show or erase the grid over the entire sheet. All relevant views snap to the grid.
- **Spacing**—Allows you to assign the same spacing to the entire model grid or individually by axis.
    - **All**—Spaces all the grid lines at the distance you define in the text box.
    - **By Axis**—Spaces the grid lines on any axis individually, at the distances you define in the text box.
    - **View**—Applies modifications to the model grid after you display the model grid.
- **Delete Grid**—Deletes the grid in the current view.

The **Settings** tabbed page contains the following options:

- **Setup Options**—Allows you to assign various values to the following:
    - **Balloon Radius**— Type in the balloon radius.
    - **Offset Distance**—Type in the offset distance of the grid lines.
    - **Decimal Places**—Type in the number of decimal places.
    - **Negative Prefix**—Type in the symbol for the negative prefix to be displayed in balloons. The default sign is "-".
    - **Text Orientation**—Assign the text to be horizontal or parallel to the grid line.
    - **Text Position**—Assign the text to be centered, above the grid line or below the grid line.
    - **Display Balloon**—Allows you to display a balloon.
- **Label Text**
    - **Show Additional Text**—Select this check box if you want to type additional text as a prefix or suffix on any axis.

Draft Grids

About Creating a Draft Grid

Using commands in the GRID MODIFY menu (accessed by clicking **View** > **Draft Grid** on the menu bar), you can create two types of draft grids in a drawing: Cartesian and Polar. The system creates a *Cartesian* grid by locating points on a plane, measuring their distance from either of two intersecting straight-line axes along a line parallel to the other axis. It creates a *Polar* grid by locating points in a plane, measuring their distance from a fixed point on a line and the angle this line makes with a fixed line. To change from one type to another, choose **Type** from the **GRID MODIFY** menu.

When using a two-dimensional draft grid in a drawing, keep in mind the following.

- The grid snap falls on the grid lines when grid snap is on (set in the **Environment** dialog box).
- The grid origin and angle affect the coordinate values specified for geometry points. The x-axis is always along one direction of the grid, while the y-axis is along the other.
- The grid origin affects the coordinate values appearing in the message area when sketching.
- If you use the **GET SELECT** menu to select an entity, the entity does not snap to grid.

To Change the Grid Display

1. Click **View** > **Draft Grid**. The **GRID MODIFY** menu appears in the Menu Manager.
2. Use the **Show Grid** and **Hide Grid** commands in the **GRID MODIFY** menu.

Note: Turning on the display of the grid does not affect the snapping of sketched entities to grid intersections. To change to grid snap, select **Snap to Grid** from the **Environment** dialog box.

To Move the Grid Origin

1. On the menu bar, click **View** > **Draft Grid**. The **GRID MODIFY** menu opens in the Menu Manager.
2. Click **GRID MODIFY** > **Origin**.
3. From the GRID ORIGIN menu, choose the type of entity to set the origin:
    - **Get Point**—Selects a point to be the origin.
    - **Edge/Entity**—Selects a point on an edge or entity.
    - **Default**—Uses the default grid origin.

4. If you choose **Get Point** or **Edge/Entity**, select the entity to locate the origin of the grid.

You can locate the grid origin at one of the following places. Use **View** > **Draft Grid** on the menu bar, then click **Origin** on the **GRID MODIFY** menu.

- Sketched entity endpoint or center
- Sketched point
- Datum point
- Edge or curve vertex

To Modify Grid Spacing

Using the following procedure, you can modify the spacing and angle of the grid lines. The commands that are available depend upon the type of grid. The system saves the values for the grid spacing and retrieves them with the drawing.

1. On the menu bar, click **View** > **Draft Grid**. The **GRID MODIFY** menu appears in the Menu Manager.
2. Click **Grid Params** from the **GRID MODIFY** menu. For a Cartesian grid, the **CART PARAMS** menu displays. For a Polar grid, the **POLAR PARAMS** menu displays.
3. Modify the grid spacing according to the menu that displays as mentioned in the preceding step.

The CART PARAMS Menu and the POLAR PARAMS Menu

Depending on whether the draft grid in a drawing is Cartesian or Polar, the **CART PARAMS** or **POLAR PARAMS** menu is displayed when you click **Grid Params** on the **GRID MODIFY** menu (**View** > **Draft Grid**). Cartesian grid (and thus the **CART PARAMS** menu) is the default draft grid.

For a Cartesian grid, the **CART PARAMS** menu displays these commands:

- **X&Y Spacing**—Sets the spacing of the grid lines in both the x-direction and y-direction to the same value.
- **X Spacing**—Sets the x-spacing of the grid lines only.
- **Y Spacing**—Sets the y-spacing of the grid lines only.
- **Angle**—Modifies the angle between the screen horizontal and the x-direction grid.

For a Polar grid, the **POLAR PARAMS** menu displays these commands:

- **Ang Spacing**—Sets the angular spacing between radial lines. The value you type must divide evenly into 360.
- **Num Lines**—Sets the number of radial lines to use. The angular spacing is 360 per number of lines.

- **Rad Spacing**—Modifies the spacing of the circular grid.
- **Angle**—Modifies the angle between the horizontal and the 0-degree radial line.

Setting Drawing Parameters

About Working with Drawing Parameters

Using the **Tools** > **Parameters** menu you can access drawing parameter functionality for drawings and drawing formats. Drawing parameters work in the same way as do model parameters. A *drawing parameter* is nongraphical information you can add to a drawing. It is useful for keeping some additional information with a drawing that you may not want to include in a note. You can show the parameter value in a note by including [`&parameter`] in the note string.

When working with drawing parameters, keep in mind the following:

- The system associates parameters only with the object that is current at the time you add the parameter.
- You cannot use drawing parameters in relations.
- You can designate parameters for Pro/PDM.
- Drawing attributes defined in versions of Pro/ENGINEER prior to Release 15.0 convert to drawing parameters automatically. The system converts attribute names that begin with a number, but you cannot use them in notes.

To Create a Drawing Parameter

1. Click **Tools** > **Parameters.** The **Parameters** dialog box opens.
2. Use the dialog box to add a parameter to the drawing. Click the plus mark at the bottom of the dialog to add a line for the parameter, and use the type, value and designate columns to describe the parameter properties. The types are as follows:

   Interger—Adds a parameter in the form of an integer (whole number).

   Real Number—Adds a double parameter. (Value can accommodate decimal places)

   String—Adds a parameter in the form of a text string as a value.

   Yes No—Adds a parameter with a value of `yes` or `no`.

   Check **Designate** to make the parameter visible to database management.

To Modify or Delete an Existing Drawing Parameter

1. From the **MODEL PARAMS** menu, choose **Modify** or **Delete**.
2. From the **PARAMETER** menu, select a drawing parameter.
   - To modify the parameter, type the new value.

- To delete it, choose **Done**.

3. To display an Information window showing all of the existing parameters and their values, choose **MODEL PARAMS** > **Info**.

Note: You also have access to Part and Assembly mode parameters while in Drawing mode. Once you delete a parameter, you cannot undo the deletion.

To Get Information About Drawing Parameters

1. Click **Tools** > **Parameters**. The **Parameters** dialog box opens.
2. Click **File** > **Export (Info)** in the dialog box

An information window opens and lists information about all parameters in the drawing.

To Save Drawing Parameter Information as a File

1. Click **Tools** > **Parameters**. The **Parameters** dialog box opens.
2. Click **File** > **Export Relations** in the dialog box
3. Click **File** > **Save As** in the Information window.
4. In the **Save As** dialog box, navigate to the directory where you want to save the parameter information file.
5. In the **New Name** box, use the default file name or type a new name. Parameter information files always contain the extension `.inf`.
6. Click **OK** to save the information file.

System Parameters for Drawings

The following table lists all system parameters available for use in drawings, classified according to functionality.

PARAMETER NAME	DEFINITION
&d#	Displays a dimension in a drawing note, where # is the dimension ID.
&ad#	Displays an associative dimension in a drawing note, where # is the dimension ID.
&rd#	Displays a reference dimension in a drawing note, where # is the dimension ID.
&p#	Displays an instance number of a pattern in a drawing note,

PARAMETER NAME	DEFINITION
	where # is the pattern ID.
&g#	Displays a gtol in a drawing note, where # is the gtol ID.
&<param_name>	Displays a user-defined parameter value in a drawing note.
&<param_name>:att_cmp	An object parameter that indicates the parameters of the component to which a note is attached.
&<param_name>:att_edge	An object parameter that indicates the parameters of the edge to which a note is attached.
&<param_name>:att_feat	An object parameter that indicates the parameters of the feature to which a note is attached.
&<param_name>:att_mdl	An object parameter that indicates the parameters of the model to which a note is attached.
&<param_name>:att_pipe_bend	An object parameter that indicates the parameters of the pipe bend to which a note is attached.
&<param_name>:att_spool	An object parameter that indicates the parameters of the spool to which a note is attached.
&<param_name>:EID_<edge_name>	An object parameter that references edges.
&<param_name>:FID_<feat_ID>	An object parameter that includes a feature parameter in a note by ID.
&<param_name>:FID_<FEAT_NAME>	An object parameter that includes a feature parameter in a note by name.

PARAMETER NAME	**DEFINITION**
&<param_name>:SID_<surface_name>	An object parameter that references surfaces.
&angular_tol_0_0	Specifies the format of angular tolerance values in a note from one to six decimal places.
&current_sheet	Displays a drawing label indicating the current sheet number.
&det_scale	Displays a drawing label indicating the scale of a detailed view. You *cannot* use this parameter in a drawing note. Pro/ENGINEER creates this parameter with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.
&dtm_name	Displays datum names in a drawing note, where name is the name of a datum plane. The datum name in the note is read-only, so you cannot modify it; unlike dimensions, a datum name does not disappear from the model view if included in a note. The system encloses its name in a rectangle, as if it were a set datum.
&dwg_name	Displays a drawing label indicating the name of the drawing.
&format	Displays a drawing label indicating the format size (for example, A1, A0, A, B, and so forth).
&linear_tol_0_0	Specifies the format of dimensional tolerance values in a note from one to six decimal places.

PARAMETER NAME	DEFINITION
&model_name	Displays a drawing label indicating the name of the model used for the drawing.
&parameter:d	Adds drawing parameters to a drawing note, where *parameter* is the parameter name and *:d* refers to the drawing. .
&pdmdb	Displays the database of origin of the model.
&pdmrev	Displays the model revision.
&pdmrev:d	Displays the revision number of the model (where *:d* refers to the drawing).
&pdmrl	Displays the release level of the model.
&scale	Displays a drawing label indicating the scale of the drawing.
&scale_of_view_detailed_bar	
&sym(<symbolname>)	Includes a drawing symbol in a note, where symbolname is the name of the symbol.
&todays_date	Displays a drawing label indicating the date on which the note was created in the form dd-mm-yy (for example, 2-Jan-92). You can edit it as any other nonparametric note, using **Text Line** or **Full Note**. If you include this symbol in a format table, the system evaluates it when it copies the format into the drawing. To specify the initial display of the date in a drawing, use the configuration file option

PARAMETER NAME	DEFINITION
	"todays_date_note_format."
&total_sheets	Displays a drawing label indicating the total number of sheets in the drawing.
&type	Displays a drawing label indicating the drawing model type (for example, part, assembly, etc.).
&view_name	Displays a drawing label indicating the name of the view. You *cannot* use this parameter in a drawing note. Pro/ENGINEER creates it with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.
&view_scale	Displays a drawing label indicating the name of a general scaled view. You *cannot* use this parameter in a drawing note. Pro/ENGINEER creates it with a view and places it in notes automatically. You can modify its value, but you cannot call it out in another note.
Pro/REPORT System Parameters	
&asm.mbr.comp....	Retrieves information about the component from the model data and displays it in the report table.
&asm.mbr.cparam....	Retrieves a given component parameter.
&asm.mbr.cparams....	Lists information pertaining to all component parameters for the current model.
&asm.mbr.name	Displays the name of an assembly member. To show tie wraps and markers, the region

PARAMETER NAME	DEFINITION
	attributes must be set to **Cable Info**.
&asm.mbr.param....	Displays information about parameters in an assembly member.
&asm.mbr.type	Displays the type (part or assembly) of an assembly member.
&asm.mbr.User Defined	Lists the specified user-defined parameter for the respective assembly components. Note that "&asm.mbr." can be used as a prefix before any user-defined parameter in an assembly member.
&dgm....	
&fam....	Retrieves family table information about the model.
&harn....	Shows cable harness parameters for 3-D harness parts and flat harness assemblies.
&lay....	Retrieves layout information about the model.
&mbr....	Retrieves parameters about a single component.
&mdl....	Retrieves information about a single model.
&prs....	Retrieves process-specific report parameters used to create reports on the entire process sequence.
&rpt....	Displays information about each record in a repeat region.
&weldasm....	Retrieves welding information about the model.

PARAMETER NAME	DEFINITION
&asm.mbr.cblprm....	Lists values for a given cabling parameters.
&asm.mbr.cblprms....	Lists values for cabling and wire parameters.
&asm.mbr.connprm....	Lists parameters for connector pins in flat harness assemblies.
&asm.mbr.location...	Lists the location callouts in a specified view or all views of the drawing in session.
&asm.mbr.pipe....	Shows pipeline, pipe segment, and Pro/REPORT bend information parameters.
&asm.mbr.generic.name….	Lists the generic name information for a family table instance in a table.
&asm.mbr.topgeneric.name….	Lists the top generic name information for a family table instance in a table when working with a nested family table.

Pro/REPORT Parameters for Manufacturing Process Drawings

Drawings of manufacturing processes are the same as drawings of any other Pro/ENGINEER object. However, special parameters are available using Pro/REPORT to create a custom table detailing the component process. An entire list of available system parameters appears in the following table. You can generate customized reports on your manufacturing processes using the Pro/REPORT functionality. Pro/REPORT allows you to access manufacturing parameters for documentation and customize the report format to suit your specific needs.

When creating a report, it is important to consider the structuring of the symbols in the report. Pro/REPORT parameters for MFG are based upon either the current step or operation in the drawing or all steps operations or both in the drawing.

Parameter Name	Definition
`&mfg.actoper.actstep.comp.name`	Lists the names of all manufacturing processes in the currently active step associated with the currently active

Parameter Name	Definition
	operation.
&mfg.actoper.actstep.comp.type	Lists the types of all manufacturing processes in the currently active step associated with the currently active operation.
&mfg.actoper.actstep.comp.param.name	Lists the names of all parameters for each process component in the currently active step associated with the currently active operation.
&mfg.actoper.actstep.comp.param.value	Lists the values of all parameters for each process component in the currently active step associated with the currently active operation.
&mfg.actoper.actstep.name	Lists the name of the currently active step associated with the currently active operation.
&mfg.actoper.actstep.param.name	Lists the names of all parameters associated with the current process step associated with the currently active operation.
&mfg.actoper.actstep.param.value	Lists the values of all parameters associated with the current process step associated with the currently active operation.
&mfg.actoper.actstep.tool	Lists the tool of the currently active step associated with the currently active

Parameter Name	Definition
	operation.
&mfg.actoper.actstep.type	Lists the type of the currently active step associated with the currently active operation.
&mfg.actoper.csys	Lists the coordinate system name referred by active operation.
&mfg.actoper.name	Lists the name of the active operation.
&mfg.actoper.step.comp.name	Lists the name for each component associated with the currently active operation.
&mfg.actoper.step.comp.type	Lists the types of all manufacturing processes for all steps associated with the currently active operation.
&mfg.actoper.step.comp.param.name	Lists the names of all parameters for each process component associated with the currently active operation.
&mfg.actoper.step.comp.param.value	Lists the values of all parameters for each process component associated with the currently active operation.
&mfg.actoper.step.name	Lists the name for each step associated with the currently active operation.
&mfg.actoper.step.param.name	Lists the names of all parameters for each step associated with the currently active

Parameter Name	Definition
	operation.
&mfg.actoper.step.param.value	Lists the values of all parameters for each step associated with the currently active operation.
&mfg.actoper.step.axis	Lists the number of axes used while machining.
&mfg.actoper.step.csys	Lists the coordinate system name referred by this step.
&mfg.actoper.step.mfg_criteria	Lists the Manufacturing Criteria name. Manufacturing Criteria is an optional property of manufacturing annotation element.
&mfg.actoper.step.same_behavior_set_name	Lists the name of a same behavior group.
&mfg.actoper.step.status	Lists the current status of the step.
&mfg.actoper.step.template_name	Lists the name of the manufacturing template.
&mfg.actoper.step.tool	Lists the tool for each step associated with the currently active operation.
&mfg.actoper.step.type	Lists the type for each step associated with the currently active operation.
&mfg.oper.User Defined	Lists the specified operation parameters.
&mfg.oper.name	Lists the operation names.
&mfg.oper.ncseq.name	Lists the NC sequence names.

Parameter Name	**Definition**
`&mfg.oper.ncseq.User Defined`	Lists the specified NC sequence parameters.
`&mfg.oper.ncseq.cutmtn.name`	Lists the cut motion names.
`&mfg.oper.ncseq.cutmtn.User Defined`	Lists the specified cut motion parameters.
`&mfg.oper.step.comp.name`	Lists the names of all components for all operations.
`&mfg.oper.step.comp.param.name`	Lists the names of all parameters for each process component for each step associated with all operations.
`&mfg.oper.step.comp.param.value`	Lists the values of all parameters for each process component for each step associated with all operations.
`&mfg.oper.step.comp.type`	Lists the type of component being processed for each step associated with all operations.
`&mfg.oper.step.name`	Displays the name of each step associated with all operations.
`&mfg.oper.step.param.name`	Lists the names of all parameters associated with the steps associated with all operations.
`&mfg.oper.step.param.value`	Lists the values of all parameters associated with the steps associated with all operations.
`&mfg.oper.step.tool`	Displays the tool for every step associated with all operations.

Parameter Name	Definition
&mfg.oper.step.type	Displays the type for every step associated with all operations.
&mfg.oper.workcell.name	Lists the workcell names.
&mfg.oper.workcell.User Defined	Lists the specified workcell parameters.
&mfg.oper.workcell.head.head_number	Displays the workcell head number.
&mfg.oper.workcell.head.tooltbl.tool_id	Lists the tools in the turret.
&mfg.oper.workcell.head.tooltbl.tool_pocket.tool_comment	Lists the tool comments for the turret.
&mfg.oper.workcell.head.tooltbl.tool_pocket.tool_position	Lists the tool pocket locations in the turret.
&mfg.oper.workcell.head.tooltbl.tool_pocket.tool_register	Lists the tool registers in the turret.
&mfg.oper.workcell.head.tooltbl.User Defined	Lists the specified tool parameters.
&mfg.oper.workcell.turret.holder_size	Lists the holder sizes for the turrets.
&mfg.oper.workcell.turret.index	Lists the turret indices.
&mfg.oper.workcell.turret.indexable	Lists the turret indexability.
&mfg.oper.workcell.turret.offset_reg	Lists the offset registers for the turrets.
&mfg.oper.workcell.turret.orient	Lists the turret orientation.
&mfg.oper.workcell.turret.standard	Lists the turret standard.
&mfg.oper.workcell.turret.tool_name	Lists the tool names for the turret.

You can access any manufacturing parameter for an operation, NC sequence, or cut motion by selecting **User Defined** in the report table at the appropriate level and entering the parameter name.

Material Parameters for Drawings

You can set the following material parameters in the report table in a drawing. When you set the material parameters and update the report table, values displayed in the report table are the same as in the drawing.

PTC_YOUNG_MODULUS

PTC_POISSON_RATIO

PTC_MASS_DENSITY

PTC_THERMAL_EXPANSION_COEF

PTC_TENSILE_ULTIMATE_STRESS

PTC_COMPR_ULTIMATE_STRESS

PTC_SHEAR_ULTIMATE_STRESS

PTC_THERMAL_CONDUCTIVITY

PTC_SPECIFIC_HEAT

PTC_HARDNESS

PTC_CONDITION

PTC_INITIAL_BEND_Y_FACTOR

PTC_BEND_TABLE

PTC_MATERIAL_DESCRIPTION

PTC_MATERIAL_NAME

PTC_HARDNESS_TYPE

To Access Part or Assembly Material Parameters in Drawing

1. With the drawing open, click **Tools** > **Parameters**. The **Parameters** dialog box opens.
2. In the **Look In** field, set Current Context to **Part** or **Assembly**.
3. Select **Materials** in the Object Type list in the **Look In** field. The **Select Material** dialog box opens and displays all the materials assigned to the corresponding model.
4. In the **Select Material** dialog box, select the required material for which you want to list all the parameters and click **OK**. All the parameters of the selected material are listed in the **Parameters** dialog box. You can add new parameters or modify the properties of the existing parameters.

Getting Drawing Information

About Getting Drawing Information

By using the **Info**, **Edit**, and **Analysis** menus, you can access information about a drawing. You can:

- Highlight currently displayed items using filters. By using **Edit** > **Highlight by Attributes**, you can filter the display of specified items according to item type (such as dimension, note, or geom tol), layer, and view.
- Perform measurement analyses on draft entities that exist in the drawing, by using **Analysis** > **Measure Draft Entities**. You can get information about distance, angle, intersection point, and tangent point.
- Display geometric and cosmetic information about an entity, and save the information to a file, by using **Info** > **Draft Entity**.
- Save a note as a text file, by using **Info** > **Save Note**.
- Display the grid angle and spacing values, by using **Info** > **Draft Grid**.
- Obtain information about out-of-date displays in the drawing, by using **Info** > **Check Display Status**. You can save the information to a file and select a recommended action to update the drawing (such as repainting and regenerating).
- Obtain information about why a template failed when you attempted to open it, by using **Info** > **Template Errors**.

To Get Information About Draft Entities

Use the following procedure to obtain cosmetic and geometric information about a selected draft entity, and to save the information on the screen or to a file.

1. Click **Info** > **Draft Entity**.
2. Select a draft entity. An Information window appears containing information about the selected draft entity.
3. To save the information on screen or to a file, click **File** > **Save As**.

To Display Drawing Grid Information

Use this procedure to display the grid angle and spacing values for a grid in a drawing.

- Click **Info** > **Draft Grid**.

The angle and spacing values for the grid are displayed in the message area.

To Get Information About Drawing Template Failures

Use this procedure to obtain information about why a drawing template failed when you attempted to open it.

- Click **Info** > **Template Errors**.

An Information window opens displaying the errors encountered when you attempted to open the template.

To Highlight Items by Type and Attributes

Use this procedure to highlight displayed items on the current sheet, using the type and attributes filters provided in the **Highlight by Attributes** dialog box.

1. Click **Edit** > **Highlight by Attributes**. The **Highlight by Attributes** dialog box opens.
2. Under **Item Type**, select the types of items you want to highlight.
3. Under **Owner Model**:
    - To highlight items that were created in the drawing (but not in the associated model or models), select **The drawing**.
    - To highlight items that were created in the current model, and shown in the drawing, select **A 3D model**. In a multimodel drawing, the system highlights only items that were created for the current model.

      Both options are selected by default.
4. Under **On Layer**, select one of the following options:
    - **Any or none**—Highlights the specified items, whether or not they exist on a layer.
    - **At least one**—Highlights only specified items that exist on at least one layer. Items that are not on any layer are not highlighted.

   Selected and **Select Layers**—Highlights only specified items that are on the layers you specify.
5. Under **Dimension Type**, select the attributes of the type of dimensions you want to highlight.
    - **Owned by a model**—Highlights dimensions that were created and reside in the associated model or models (such as assembly and feature dimensions). In multimodel drawings, all dimensions that reside in all associated models are highlighted.
    - **Created and associative**—Highlights dimensions that were created in the drawing and are associative; that is, they update when the referenced entity is regenerated (resized).

      Note: If you have set the drawing setup file option `associative_dimensioning` to `no`, this option is disabled.

- **Created but non-associative**—Highlights dimensions that were created in the drawing, but are non-associative; that is, they do not update when the referenced entity is regenerated.

6. Under **Displayed in View**, select **Any** to display all items of the specified type in any and all views. Select **Selected** or click **Select View** to select a view in which to display the items, using the **GET SELECT** menu.
7. Click **Highlight** to highlight the specified items. The items are highlighted in magenta. Click **Close** to close the dialog box.

To Get Information About Out-of-Date Displays

1. Click **Info** > **Check Display Status**. An Information window appears displaying the sheet number, view ID, view name, any information about missing display status, and the recommended action to update the drawing.
2. To save the information to a file, click **File** > **Save As** in the Information window.

To Perform Measurement Analyses on Draft Entities

1. Click **Analysis** > **Measure Draft Entities**. The **Draft Measure** dialog box opens, and the **GET SELECT** menu appears.
2. In the **Type** list, select the type of draft to measure (**Distance, Angle, Intersection Point,** or **Tangent Point**).
3. Under **From** and **To**, select the first and second entities to measure.

   Note: To redisplay the GET SELECT menu, click the **Select** button .
4. If you selected **Angle**, you can select **Use horizontal** to calculate the angle, instead of selecting a second entity.
5. Click **Compute**. The computations appear under **Results**. The results include Angle, Distance, Intersection Point, and Tangent point.
6. Click **Compute** to perform another analysis, and **Info** to display the results in an Information window.
7. Click **Close** to remove the **Draft Measure** dialog box.

To Save a Drawing Note as a File

Use the following procedure to save drawing notes as .txt files.

1. Select the note you want to save.
2. Click **Info > Save Note** from the right mouse button shortcut menu.
3. Enter a name for the file on the Pro/ENGINEER prompt line. The note is saved in your working directory by default. Alternately you can enter an absolute path to the file.

Importing and Exporting Data

About Importing Draft Data from External Applications

You can add draft entities to a Pro/ENGINEER drawing through Detailed Drawings or Pro/ENGINEER Interface. You can also import the following file types. However, you cannot modify the files without Detailed Drawings.

- Drawing (*.drw)
- Report (*.rep)
- IGES (*.igs, *.iges)
- SET (*.set)
- DXF (*.dxf)
- STEP (*stp, *.step)
- CGM (*.cgm)
- DWG (*.dwg)
- Parasolid (x_t, xmt_txt, x_b, xmt_bin, x_n, xmt_neu, xmt)
- Stheno (*.tsh)
- Medusa

You can create a group of entities which maintain their group association through an IGES export.

Sheets are exported one at a time. You must create a file for each sheet. The drawing filename is appended with the sheet number used in the new file name.

Imported objects are placed as nonassociated draft items.

To Export to External Formats

You can export a drawing to a data file in the following formats:

IGES (`.igs`) CADRA (`.igs`) SET (`.set`) CGM (`.cgm`)	DXF, (`.dxf`) DWG, (`.dwg`) STEP (`.stp`) Medusa (`.she`)

Sheets are exported one at a time. You must create a file for each sheet. The drawing filename is appended with the sheet number as the new file name.

Note: If you export a drawing with shaded views to the DXF or DWG format, an image file with the .jpg extension is created along with the respective .dxf or .dwg file. To view the shaded image when you retrieve this drawing in AutoCAD, ensure that the .jpg image file is in the same folder as the .dxf or .dwg file.

1. Click **File** > **Save A Copy**. The **Save a Copy** dialog box opens.
2. Under **New Name**, enter the desired new file name.

3. From the **Type** list, select the desired file type.
4. Click **OK**.

To Import External Formats

Imported objects are placed as non-associated draft items.

1. Click **Insert** > **Shared Data** > **From File**.
2. In the **Open** dialog box, use **Look In** to find the file you want to import, and then select the file.
3. Click **OK**. The file is placed on the current sheet.

To Create IGES Groups

To create a group of entities which maintain their group association through an IGES export:

1. Click **Tools** > **IGES Group**. The Menu Manager opens.
2. Click **Create**.
3. Enter the IGES group name, and then select the items that you want to belong to the group. You cannot select items from different views. When you are done selecting items, click **Done**.

The system creates the group.

To Export a Drawing as an Image File

You can capture the current screen using the **Save a Copy** command, or you can "plot" any number of the drawing sheets to an image file using the Print command.

The following types of image files are supported:

JPG—Compressed file for Web viewing (Plot Only)

TIFF—Detail-rich image file suitable for printing (Screen capture or Plot)

PIC—A proprietary PTC format. used for file previews, picture-to-file comparisons and the Model View program. You can verify the differences between drawings using a previously saved .pic file of the sheet.

To save a TIFF file of the current screen:

1. Click **File** > **Save a Copy**. The Save a Copy dialog box opens.
2. Under **New Name**, enter the desired new file name.
3. From the **Type** list, select TIFF.
4. Click **OK** .You prompted "Pop the current window?" Click **Yes**. The file is created.

To plot a whole sheet (or sheets) as a TIF or JPG file:

1. Click **File** > **Print**. The **Print** dialog box opens.
2. Click **Destination** > **Add Printer Type** > **TIFF** or **JPEG**.
3. Check **To File**.
4. Click **OK**. The Print to File dialog box opens.
5. Use the dialog box to name the new file. Click **OK**. The new image file is created.

To save a sheet as a PIC file:

1. Click **File** > **Save a Copy**. The Save a Copy dialog box opens.
2. Under **New Name**, enter the desired new file name.
3. From the **Type** list, select **Picture** (*.pic).
4. Click **OK** .The file is created in the current working directory with the .pic extension. The default file name is the name of the drawing.

Inserting OLE Objects

About OLE Objects

An object linking and embedding (OLE) object is an external file, such as a document, graphics file, or video file that was created using an external application and which can be inserted into another application, such as Pro/ENGINEER.

You can create and insert supported OLE objects into a two-dimensional Pro/ENGINEER file, such as a drawing, report, format file, layout, or diagram. You can insert objects into a Pro/ENGINEER drawing from the **Insert Object** dialog box that opens when you click **Insert** > **Object**.

You can insert OLE objects in Pro/ENGINEER by linking or embedding. After you insert an object, you can edit it within Pro/ENGINEER or in its separate application window, outside Pro/ENGINEER.

In Pro/ENGINEER, you can insert OLE objects into the following 2D file types:

- Drawings
- Reports
- Format files
- Layouts
- Diagrams

OLE Object Display

OLE object creation and editing capabilities apply to Windows only (Windows 2000 and XP). On UNIX, a simple graphics representation is displayed. The object itself is not displayed. You cannot create or edit OLE objects in UNIX; you can only move or resize an object.

Note: The number and type of supported OLE objects may vary, depending on the other applications installed on your computer.

When you retrieve embedded objects on UNIX from another platform, they are retrieved in the form of images and cannot be modified. For example, if you view an embedded Microsoft Excel file on UNIX, the file is shown as an image. However, if you transfer it back to Windows, the file retains all its properties and can be modified.

Linking Objects

A linked object is a file created outside of, and then linked into Pro/ENGINEER. A picture, linked to the file, appears in your drawing. Any changes to the external file are reflected in your drawing and changes that you make to the object from within Pro/ENGINEER are saved to the original object.

Embedding Objects

An embedded object is saved completely within a Pro/ENGINEER drawing file and contains no links to an external file. When you insert an object as embedded, Pro/ENGINEER copies the file and places it into your document. You can still activate the object from within Pro/ENGINEER using the program that created it. Any changes that are made to the original external file are not reflected in the embedded copy. You can also create a new object (one that has no external file reference) as an embedded object. New objects you create within Pro/ENGINEER are always embedded.

Plotting Options for OLE Objects

You can plot OLE objects in the Drawing mode. The following drivers support printing of OLE objects:

- HPGL2

  Note: For the HPGL2 driver to print OLE objects, the printer must support the HP RTL extension.

- PostScript
- Color PostScript
- MS Printer Manager
- Plot to Screen
- PDF
- DXF/DWG

Note: Formats not mentioned in the supported drivers list do not support printing of OLE objects.

You can print OLE objects in the graphics mode with OpenGL or GDI, on both UNIX and Windows. To retrieve and print a drawing with an OLE object from UNIX, set the value of the `drawing_ole_image_dpi` configuration option to more than zero in the graphics mode, and save the drawing.

To Insert an OLE Object

You can insert an OLE object as a linked object or as an embedded object. Use one of the procedures below to insert an OLE object.

To Create a New Embedded Object within Pro/ENGINEER

1. Click **Insert** > **Object**. The **Insert Object** dialog box opens.
2. Select **Create New** if it is not already selected.
3. Under **Object Type**, select the type of object you want to embed into your drawing.

   Note: Unsupported object types may be listed. They contain a standard Microsoft icon.
4. Click **OK** to close the dialog box and create your object. The object window appears on the drawing in Edit mode. The toolbars that belong to the application that created the drawing appear in the Graphics window.
5. Edit the object as desired, then click anywhere outside the object window to exit Edit mode and return to Pro/ENGINEER.

To Create an Embedded Object from an External File

1. Click **Insert** > **Object**. The **Insert Object** dialog box opens.
2. Select **Create from File**.
3. Do one of the following:
   - Under **File**, type in the path and file name of the file you want to embed.
   - Click **Browse** to open the **Browse** dialog box, locate and select the file, and click **OK**. The file name and path is added to the **Insert Object** dialog box.
4. Clear the **Link** check box.
5. Click **OK** to close the dialog box and create the object. The object window appears on the drawing in Edit mode. The toolbars that belong to the application that created the drawing appear in the Graphics window.
6. Edit the object as desired, then click anywhere outside the object window to exit Edit mode and return to Pro/ENGINEER.

To Link an Object

1. Click **Insert** > **Object**. The **Insert Object** dialog box opens.
2. Select **Create New**.
3. Under **Object Type**, select the type of object you want to embed into your drawing.

   Note: Unsupported object types may be listed. They contain a standard Microsoft icon.
4. Select the **Link** check box.

   Note: **Link** allows you to create a linked OLE object rather than an embedded object. Pro/ENGINEER inserts a picture of the file contents into your drawing. The picture is linked to the file so that all future changes to the file are reflected in your drawing. **Link** is available only when you create an object from an existing file
5. Click **OK** to close the dialog box and create the object.
6. Edit the object as necessary, then click outside of the object window to exit Edit mode and return to Pro/ENGINEER.

To Modify an Inserted OLE Object

1. Right-click the object you want to modify. The OLE object shortcut menu appears.
2. On the shortcut menu, click **Open** if you want to edit the object in its own application window, or click **Edit** to modify the object within the Pro/ENGINEER environment. If you click **Edit**, the object opens within the Pro/ENGINEER window, and the application toolbars open below the message area.
3. Edit the object as desired.
4. If you are editing the object outside of Pro/ENGINEER, you must update Pro/ENGINEER with the changes to the object. Within the object application window, click **File > Update Pro/ENGINEER**.
5. To exit Edit mode and return to Pro/ENGINEER, do one of the following:
   - If you are editing the object outside of Pro/ENGINEER, open the application **File** menu and click **Close & Return to Pro/ENGINEER**, or **Exit & Return to Pro/ENGINEER**.
   - If you are editing the object from within Pro/ENGINEER, click anywhere outside the object window. The application toolbars close and the object displays as edited.

To Move or Resize an OLE Object

Although the graphics of an OLE object do not display on UNIX systems, you can still move or resize it just as you can on Windows systems.

To Move an OLE Object

1. Click and drag the OLE object to the desired location on the drawing. As you drag, a ghost outline of the object window moves with the pointer.
2. Release the mouse button. The object moves to the new location.

To Resize an OLE Object

Select the object, and then click and drag a bounding box. A ghost outline of the new object size displays as you move the mouse pointer. When you release the mouse button, the object resizes.

Comparing and Merging Drawings

Comparing Drawings

To Compare Two Drawings

1. Open the first of the two drawings whose features you want to compare.
2. Click **Analysis** > **Compare Drawing**. The **Open** dialog box opens.
3. Select the second drawing and click **Open**. Pro/ENGINEER opens the second drawing and performs the comparison analysis.

   When the comparison is finished, a report is generated in the Pro/ENGINEER browser. This report lists the differences between the current (base) drawing and the second (comparison) drawing. The report displays the entities in the comparison drawing that have been modified or newly. It also lists the modified or newly added entities in the base drawing that are missing in the comparison drawing.

Note: Pro/ENGINEER does not automatically close the second drawing when you quit the comparison operation. You must close the second drawing manually using **File** > **Close Window**.

To Create a Drawing Difference Report

1. Open the first of the two drawings that you want to compare.
2. Click **Analysis** > **Compare Drawing**. The **Open** dialog box opens.
3. Select the second drawing and click **Open**. Pro/ENGINEER performs the comparison analysis and the difference report is displayed in the Pro/ENGINEER browser.

Note:

- You can choose to display only specific information in the report by selecting only the required entities under **Filter Change Types** and clicking **Apply**.

- You can sort the difference report in ascending or descending order based on the contents of a column by clicking its column header. Clicking the column header again reverses the sort order.
- You can export the difference report by clicking **Export Report** in the HTML report page. By default, the exported file is saved in the current working directory in Comma Separated Value (CSV) format as <model name>_CMPRO.csv and <model name>_CMPRO.csv.1. Set the export_report_format configuration option to rich_text to save the report in rich text format as <model name>_CMPRO.txt and <model name>_CMPRO.txt.1. The default setting for the export_report_format configuration option is comma_delimited. The <model name>_CMPRO.csv.1 and <model name>_CMPRO.txt.1 are version files. Whenever you export the same report again, new version files are created with incremented file names and the <model name>_CMPRO.csv and <model name>_CMPRO.txt files are replaced by the latest exported file. You can open the exported files in Microsoft Excel.

Comparing Drawings Versions

About Comparing Drawing Versions

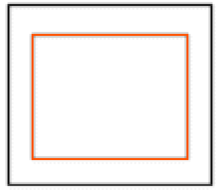
By delaying the regeneration of drawing views, you can compare the affected geometry before and after the modification. Active drawing represents the current geometry (i.e. geometry after modification) and Comparison drawing represents the previous geometry (i.e. geometry before modification). The delayed regeneration enables you to:

- Show before and after detailing of design changes
- Document design alterations to better explain why a change was made
- Retain changes for future design considerations
- Inform the downstream processes of design changes

You can initiate a drawing comparison automatically when you retrieve a changed drawing, or you can manually compare the drawings at any point in the drawing creation process.

The comparison between drawings is visible by highlighting changed geometry on drawing views. You can define a color scheme to help you recognize the similarities and differences.

In the following example, current geometry is the solid square cross-section, as compared to the previous geometry, which is the hollow square cross-section. During drawing comparison, the inner rectangle corresponding to the hollow section will be highlighted in “Comparison Drawing” color.

Current Geometry	Comparison Drawing
	

If the comparison reveals geometry you need on the active drawing, you can copy items from the comparison drawing to the active drawing. You can browse to and copy items from multiple drawings. Once the item is copied, you can work with it as you would similar items in the existing drawing. However, any geometry you copy is not parametric and will not update if the model changes.

To Compare Drawing Versions

By delaying the regeneration of drawing views, you can compare iterations of your drawings and determine when and what updates should be displayed in the drawing.

1. Retrieve an existing drawing (**File** > **Open**). The drawing opens.
2. Click **File** > **Compare**. The **Compare Drawings** dialog box opens.
3. Under **Comparison Drawing**, browse to or type the name of the drawing you wish to compare the active drawing to. The active drawing name is listed under **Active drawing**.
4. By default, the drawing comparison examines the same sheet number on the active and comparison drawings. You can vary the sheets being compared by clearing **Same as active** and changing the respective sheet numbers for the active and comparison drawings. If the number of sheets in each drawing is different, you must type the sheet number in the appropriate box.
5. To see the variations between the active and comparison drawings, you can define what type of comparison you are seeking and a color scheme to make the comparison visible:

- **Define the type of comparison**:

  To highlight the differences between the comparison drawing and the active drawing, click **Comparison drawing only**. Any variations highlight on the comparison drawing, while the active drawing displays in its original color scheme.

  To highlight common elements between the active drawing and comparison drawing, click **Similarities**. The similarities and differences between the drawings are indicated in the following ways:

  a. Items on the active drawing that differ from the comparison drawing highlight in the color designated by Active drawing.
  b. Items on the comparison drawing that no longer exist on the active drawing highlight in the color designated by **Comparison drawing**.

c. Items that are in the same position on both the active drawing and comparison drawing highlight in a color formed by combining the colors designated by **Active drawing** and **Comparison drawing**.

- **Define a color scheme**:

  The highlight color scheme is determined by the settings for **Active drawing** and **Comparison drawing**. To modify the colors, click the appropriate color pad to access additional colors for each drawing type.

  Note: When highlighting **Comparison drawing only**, only is available for the active drawing color scheme. You can only modify the active drawing color for **Similarities**. Any color changes will be remembered if you toggle between the **Comparison drawing only** and **Similarities** options.

6. If necessary, you can copy items from the comparison drawing to the active drawing. With the Compare Drawings dialog box open, click the **Items to copy** arrow and select the items to copy individually or use region selection. The selected items are highlighted in red.

   You can relate these items to a particular view by selecting a view from the **Relate to View** list. By default, copied items are not related to any view. Items can be unrelated from a view after the Compare Drawings dialog box is closed.

   After selecting the items to copy, click **OK** on the Select dialog box. The items are copied to the active drawing; and are also added to any default layers that include the copied draft entities. The copied items remain selected on the active drawing. You can browse to and copy items from multiple drawings.

7. When you are done with the drawing comparison, click **OK**. The **Compare Drawings** dialog box closes and you can continue dimensioning and detailing the active drawing.

Note: Color differences are not considered in the drawing comparison. For example, two drawings that have the same content and positioning but use different line styles are considered identical drawings; therefore no highlighting will occur.

Comparing Drawings to a Saved Image

About Verifying Differences Between Drawings

Using the **Picture** command in the Pro/ENGINEER **Utilities** menu, you can verify the differences between two versions of a drawing. After you store a picture of the drawing, you can retrieve it as a layer over the drawing. Anything that is the same cancels out, leaving on the screen only the differences between the two.

To generate a report that lists the differences between two objects, choose **File** > **Integrate**.

After you have retrieved an integration project file and the difference report, source object, and target object are displayed, the INTEGRATE menu appears, listing commands for performing the integration:

- **Next**—Move to the next item (difference) in the report.
- **Previous**—Move to the previous item (difference) in the report.
- **Show**—In the active window, highlight the entity (feature, component, and so forth) corresponding to the current item in the difference report. (This command is accessible only if the current item belongs to the object in the current Pro/ENGINEER Window.)
- **Info**—Open the Information window, which contains information about the current item in the difference report (model name, component number, internal ID number, part name, and parents and or children).
- **Action**—Open the ACTION menu. When you specify an action for one occurrence of an item (feature, component, and so forth) in the report, the specified action is set automatically for that item every time it appears in the report.

The ACTION menu contains the following commands:

- **To Merge**—Mark this source item to be merged automatically into the target.
- **To Delete**—Mark this item for automatic deletion from the target.
- **To Ignore**—Ignore this item (that is, take no merge or delete action on it even though a difference is noted).
- **Clear**—Clear all pending actions or instructions to ignore an item.
- **Do Manual**—Integrate the item manually. This involves copying the item manually from the source object to the target object.
- **Merged**—Mark an item as "Merged" after it has been merged manually.
- **Deleted**—Mark an item as "Deleted" after it has been deleted manually.
- **All Changes**—Accept and integrate all changes to the source and target drawings.
- **Source Changes**—Accept and integrate the changes to the source drawing.
- **Target Changes**—Accept and integrate the changes to the target drawing.
- **Unspecified**—Allows the indicated action to be applied to all items that do not already have an action assigned to them.
- **Overwrite**—Allows you to assign an action to all items, even if actions have already been assigned to the items. The new action overwrites the existing action.
- **Current Item**—Allows you to apply an action to the current item.
- **All Items**—Allows you to apply an action to all items that can be acted on.
- **By Type**—Allows you to apply an action to items by type: DIM COSMETICS or PARAMETERS.

- **Execute**—Perform automatically the actions specified in the ACTION column of the difference report to resolve differences between the source and target objects. Each item is marked as Merged, Deleted, or Ignored according to the action that was specified for it.
- **Save Report**—Save the difference report in your local working area as an ASCII text file.
- **Merge View**—Display the difference report with the items sorted in the order in which they have to be integrated to correctly resolve all the external references (for example, parents must be merged before children and children must be deleted before parents).
- **Diff View**—(Default) Display the difference report sorted by object and, within object, by item type. This display gives a quick report of where and what the differences are.

To Compare a Drawing to a Saved Image File

You can compare a drawing sheet to a previously saved .pic file of the sheet. You can compare the whole sheet or use any portion of it in magnification.

1. Click **Analysis** > **Compare Sheet to Picture**. The **File Open** dialog box opens.
2. Select the .pic file to compare. The pic file is "layered" over the drawing page. Common items are suppressed, and any different items, either added or deleted, are shown.
3. To return to the current drawing click Repaint.

Note: To save a drawing sheet in PIC format, use the **File** > **Save As** command.

Merging Drawings

To Merge Drawings

Merging two drawings allows increased performance and management of large drawings. Individual Pro/ENGINEER users can work in parallel and then merge their separate drawings into a single drawing file. A source file is appended to the target file as additional sheets; for example, when you merge a two-sheet source drawing into a four-sheet drawing, the target drawing then contains six sheets.

1. Click **Insert** > **Shared Data** > **From File**.
2. Use the **Open** dialog box to select the source file to merge into the current drawing. When you select a file, the source file is merged into the current (target) file.
3. Select additional source files to be merged, one file at a time, to add additional sheets to the current drawing.

Note:

- Merging removes the source file from memory. If you previously saved the source file to disk, it is removed from the session but still exists on disk.
- When you merge two drawings that have different symbols, and if the symbols have the same name, then these symbols are saved as different symbol instances with the same name in the symbol gallery. For example, if you merge two drawings that have different symbols, both named `abc`, then these symbol instances are saved as `abc` and `abc(2)` in the symbol gallery.

Rules for Merging Drawings

The following rules and restrictions apply to merging drawings or reports:

- The target drawing file must be in session.
- A drawing or report cannot be merged into itself.
- The source drawing is removed from memory after it is merged.
- The main drawing file can contain one or more formats. Each sheet can use a different format. When a source drawing is merged into a target drawing that has a different format, the source format is not automatically replaced. You can replace the source format after the merge.
- When the source drawing to be merged contains dimensions that are the same as those of the target drawing, Pro/ENGINEER automatically removes these duplicate dimensions from the source drawing.
- When files to be merged reference a different version of the same model, the latest version in memory is used for the drawing.
- The source drawing and the target drawing must use the same drawing units.
- When the source drawing and the target drawing contain a model with the same name, even if these are different models, the model currently in session is used in the final drawing.
- After a file is merged, the drawing setup file options from the target drawing override the setup options used by the source drawing.
- If the source drawing contains a drawing program, it is deleted during the merge. To preserve a drawing program, use the drawing that contains the drawing program as the target drawing.
- Drawing parameters in the source drawing are deleted during the merge.

Improving Performance with Representations

About Drawing Representations

Using drawing representations, you can improve performance, particularly when working with multimodel large assembly drawings and drawings of complex models. Drawing representation functionality is available only with Advanced Assembly.

A drawing representation is a series of commands that specifies a display configuration for the current drawing. You can use drawing representations to control which models and which views of a drawing the system retrieves and displays. For example, you can temporarily remove all models and views that are not necessary for current work.

With drawing representation functionality, you can perform the following operations:

- Improve drawing retrieval time and improve drawing performance by minimizing display calculations such as view regenerations and repaints.
- Configure the display of a drawing so that you can work with just the drawing information that you need for current work and keep unnecessary data out of memory.
- Turn off the display of views that you are not working on so that the system does not calculate unnecessary display information and, in particular, does not retrieve models into session when you retrieve a drawing.
- Load a drawing to a particular drawing sheet, zoom location, or view center—for example, retrieve only specified sheets of a drawing without loading all the sheets and displaying every view on each sheet.

To Create a Drawing Rep for the Current Drawing

1. Click **Tools** > **Drawing Representation.** The Menu Manager opens.
2. Click **Create**. The **Drawing Rep Tool** dialog box opens with the **View Display** page visible. The dialog box automatically fills in the appropriate commands and updates the selections to the current state of the drawing. The current commands are displayed in the command list area.

   The **View Display** page can be used to configure view display.
3. In the **Drawing Rep Tool** dialog box, type a drawing representation name in the **Name** box or use the default.
4. In the **Command Definition** area of the dialog box, select **Display** or **Erase** to change the view display:
   - **Display**—Turns on the display of specified drawing views
   - **Erase—** Turns off the display of drawing views

An erased view is replaced by a green outline containing the name of the view. To prevent erased views from being highlighted, clear **Highlight Erased Views** in the Environment dialog box.

You can switch between **Display** and **Erase** as you specify cumulative commands.

5. Change the selection of views:
   - Select **All** to select all the views in the current drawing.
   - Select **Individual** to specify a particular view in the current drawing. The selection tool is automatically activated, and the GET SELECT menu appears, allowing you to select an on screen view. You can use **Navigate Sheets** to navigate to select a view on another sheet. When you select a view, its name appears in the box.
   - Select **Sheet** to specify the sheet(s) on which to apply the commands. Type the sheet numbers in the sheet box. Enter a sheet number or multiple numbers separated by a comma (1,3), or specify a range (5-12).
   - Specify views of a particular type by selecting **Type**, and then select a view name from the **Type** list.

   The **Type** list includes the most frequently used view types:

   General

   Projection

   Detailed

   Auxiliary

   X-section

   Broken

   Partial

   Exploded

   You can select more than one type in any applicable combination to refine the selection. For example, select Projection to select all projection views; then select X-section to specify only cross-sectional projection views.

   Select **Individual** to select an unlisted view type such as Graph, Revolved, or Of Flat Ply.

   - Select **Model** to select all views associated with the specified drawing model. Click the selection tool and select a drawing model from the list of available models (models belonging to the current drawing).
6. Each command that you define is listed in the command list area. When a rep is executed, all the commands are performed in sequential order, as they were entered. Control the command list as follows:

- After you select an option to configure the drawing, click **Add** to add the command to the command list. **Add** has the same effect as **Execute** unless the **Delay View Command(s) Until Execute** check box is selected.
- To insert a command into the command list, select a command in the list, then select an option, then click **Insert**. The new command is inserted into the command list above the selected command. When the **Delay View Command(s) Until Execute** check box is selected, **Insert** does not execute until you click **Execute**.
- To delete a command from the command list, select a command and click **Delete**. The command is removed from the command list. When the **Delay View Command(s) Until Execute** check box is selected, **Delete** does not execute until you click **Execute**.

7. At any time, or when all the commands are listed to configure the drawing representation, click **Execute**. The system applies all the commands listed, in the order in which they are listed, to the current drawing. When all listed commands have been applied, **Execute** is not available unless **Delay View Command(s) Until Execute** is selected.
8. Click the **Drawing Display** tab.

   The **Drawing Display** page can be used to automatically retrieve a specific sheet, go to a previously named view, or center a specific view. It also controls the behavior of tables with repeat regions.

   Use the **Drawing Display** options to control the cosmetic components of a drawing. In the Drawing Location area of the dialog box, do one of the following:

   - Select from the **Go to Sheet** list of available sheets in the current drawing to specify the sheet where to load the drawing representation (for example, load to sheet 6 of a 10-sheet drawing). When you retrieve a drawing representation, the specified sheet of the drawing is displayed. Only specified sheets are loaded when you retrieve the drawing representation.
   - To specify a named pan/zoom location for the drawing, select the name of a view from the **Go to Named Pan/Zoom State** list. This list contains the same view names that are on the **View** > **Saved Views** list (the views that you previously defined).

   When you next retrieve the drawing representation, the drawing zooms to the selected view. This allows you to make a minor change in a particular small part of the drawing without having to see the entire drawing.

   - Select **Go to Center of View** to specify the location for the drawing representation. Centering a view keeps the drawing from repainting other views. Selecting **Go to Center of View** automatically clears **Go to Sheet** and **Go to Named Pan/Zoom State**.

     Select the view to go to from the list of views and models, or use the GET SELECT menu to select a view, and select the middle of the view. When you next retrieve the drawing representation, the drawing centers the specified view.

9. In the **Table Preferences** area, specify table control:
   - **As Is**—Leaves BOM tables in their current state
   - **Frozen**—Freezes BOM tables—repeat region calculations are frozen so any changes to the BOM assembly will not be reflected in the table
   - **Updating**—Updates BOM tables—repeat regions automatically calculate changes to BOM assembly information
10. When you have finished configuring the drawing representation, click **OK** to accept the commands. The **Drawing Rep** menu appears.

    After you have created a user-defined drawing representation, the entire **Drawing Rep** menu appears, and you can execute, copy, redefine, or delete a drawing representation, or display information about it in a printable window.
11. Save the drawing. Saving the drawing also saves the drawing representation.

To Create a New Drawing Rep While Opening a Drawing

You can create a new drawing representation upon retrieving a drawing. This procedure lets you go straight to the **Drawing Rep Tool** dialog box in the **No Views** drawing representation state. No drawing models are retrieved, and **Delay View Command(s) Until Execute** is selected by default.

> **Note: Delay View Command(s) Until Execute** is selected by default for creating a drawing representation upon retrieval. This is a safeguard against accidentally retrieving the models into session while defining the drawing representation.
>
> With **Delay View Command(s) Until Execute** selected, you can specify several commands to configure the rep before executing any of them. Each command is added to the command list. The commands are not executed, and the system does not retrieve data, until you click **Execute**.
>
> When the **Delay View Command(s) Until Execute** check box is cleared, each command is executed immediately.

1. Click **File** > **Open**.
2. Select a drawing from the File Open dialog box (click just once to select the drawing, not to open it) and click **Open Rep**. The Open Rep dialog box opens, listing the two default drawing representations, All Views and No Views.
3. Do not select a drawing representation, but select the **Create New Drawing Rep** check box, and click **OK**. The Drawing Rep Tool dialog box opens.
4. **Note:** Selecting **Create New Drawing Rep** is a convenient way to access the Drawing Rep Tool dialog box without retrieving any models. The system retrieves the selected drawing, with none of its views displayed and its tables set to frozen. The models are not retrieved into session.

To Configure a Drawing Representation

You can configure a drawing representation to control view display and drawing display in the following ways:

- Display or erase all views
- Display or erase a particular view
- Display or erase views of a particular type, for example, all cross-sectional views
- Display or erase all views associated with a particular model
- Display or erase views on a specified sheet
- Load the drawing to a specified sheet
- Load the drawing to a pan/zoom state
- Load the drawing to the center of a specified view
- Set tables to frozen or updating

A drawing representation is never active—it is always executed. The drawing is always in a stage of configuration and can be changed at any time. Creating a drawing representation means that you are specifying one or more commands to configure the current drawing. As you specify commands, you can apply each command to the drawing immediately, or you can define several commands and then run them at once. A delay option is provided to allow you to configure the drawing without retrieving data until the configuration is complete.

The order in which you define commands is important. The system compiles a list of commands and runs them in the order in which you define them. The commands interact with each other cumulatively. For example, you can turn all views off (command #1) and then turn on all general views (command #2).

Drawing representations behave differently than simplified representations in Part and Assembly modes in that if one representation is executed after another representation, the drawing will display the results of both representations. For example, you could erase all views on sheet #1; then erase views on sheet #2; and then display all views. Each command is applied and the last command, when implemented, counteracts the earlier commands.

The Drawing Rep Menu

Use the **Drawing Rep** menu commands together with the **Drawing Rep Tool** dialog box to create and work with a drawing representation.

When you first access the menu, only the **Create**, **Execute**, and **Done-Return** commands are available.

After you have created a drawing representation, the other commands are also available.

The **Drawing Rep** menu contains the following commands:

Create—Opens the **Drawing Rep Tool** dialog box and allows you to configure a drawing representation for the current drawing. The dialog box automatically fills in the appropriate commands and updates the selections to the current state of the drawing.

Execute—Opens the **Select Rep** dialog box and allows you to select a rep to execute from the list of existing representations. Select a rep to execute, and click **OK**. The system executes the selected rep for the current drawing. That is, it executes the commands required to configure a drawing as defined by the drawing representation. To save the drawing representation, you must save the drawing.

Copy—Copies the selected user-defined drawing representation to a new drawing representation with an updated name.

Redefine—Displays the selected user-defined drawing representation in the **Drawing Rep Tool** dialog box so that you can modify it.

Delete—Deletes the selected user-defined drawing representation.

Info—Displays a printable window containing information about the selected rep.

Done-Return—Exits the **Drawing Rep** menu and returns to the **ADV DWG OPTS** menu.

The Drawing Rep Tool Dialog Box

The Drawing Rep Tool dialog box is divided into two pages and opens with the View Display page visible. The **View Display** page allows you to control view display configurations, and the **Drawing Display** page allows you to control sheet, pan/zoom, and table configurations. The following options are available for both pages:

Name—Displays a default name for the drawing representation that you can change

Navigate Sheets—Allows you to navigate through the sheets of a drawing to display the current sheet

OK—Accepts all the drawing configuration settings, closes the Drawing Rep Tool dialog box, and returns to the Drawing Rep menu

Execute—Executes the active command or applies all the listed commands to the drawing. When there is no active command, **Execute** is not available as an indication that all listed commands have been applied.

Default Drawing Representations

Pro/ENGINEER automatically creates two default drawing representations named All Views and No Views for all newly created drawings.

To create the default drawing representations for a previously saved drawing, retrieve the drawing and save it. The All Views and No Views default drawing representations are created. These representations are listed in the **Select Rep** dialog box that opens when you are executing a drawing representation using the **DRAWING REP** menu or in the **Open Rep** dialog box when you are executing a drawing representation while retrieving a drawing.

- **All Views**—Turns on all views to display the full drawing, keeps tables updating, and resets the pan/zoom setting.

  All the models are retrieved.

- **No Views**—Turns off all views on all sheets of the drawing, keeps tables frozen, and resets the pan/zoom setting.

  No model is in session.

  You can use a No Views representation to create detail items such as notes and symbols that have no attachment to the model. You can retrieve the drawing showing only the notes and work on them without loading any models.

To Execute a Drawing Representation

You can execute a new, All Views, or No Views drawing representation. An All Views rep configures the drawing to turn on all views to display the full drawing, keeps tables updating, and resets the pan/zoom setting. A No Views drawing representation configures the drawing to turn off all views on all sheets of the drawing, keeps tables frozen, and resets the pan/zoom setting. When you load the drawing, no model is loaded.

1. Click **Tools** > **Drawing Representations**. The Menu Manager opens.
2. Click **Execute**.
3. Select the name of the drawing representation, and click **OK**. The system executes the selected drawing representation.

To Copy a Drawing Representation

After you have saved a drawing representation, you can copy the rep and save it with different properties.

1. Click **Tools** > **Drawing Representation**. The menu manger opens.
2. Click **Copy**. The **Select Rep** dialog box opens, listing the saved representations.
3. In the **Select Rep** dialog box, select a rep to copy from the list of existing representations, and click **OK**. You can copy only user-defined representations.
4. The system copies the selected drawing representation and opens the **Drawing Rep Tool** dialog box. The drawing representation name is automatically updated, the dialog box is populated with the same commands as those of the source rep, and the current commands are displayed in the command list area.
5. Rename and modify the new drawing representation.

To Redefine a Drawing Representation

1. Click **Tools** > **Drawing Representation**. The Menu Manager opens.
2. Click **Redefine**. The **Select Rep** dialog box opens, listing the saved representations.

3. In the **Select Rep** dialog box, select a rep to redefine from the list of existing representations, and click **OK**. You can redefine only user-defined representations.
4. The system opens the selected rep in the **Drawing Rep Tool** dialog box.
5. Redefine the selected rep.

To Delete a Drawing Representation

1. Click Tools > **Drawing Representations**. The Menu Manager opens.
2. Click **Delete**. The **Select Rep** dialog box opens, listing the saved representations.
3. Select a rep to delete, and click **OK**. You can delete only user-defined representations.
4. The system deletes the selected rep.

To Obtain Information About a Drawing Representation

After you have saved a drawing representation, you can obtain information on screen about the rep, using the **Info** command on the **Drawing Rep** menu

1. Click **Tools** > **Drawing Representations**. The Menu Manager opens.
2. Click **Info**. The **Select Rep** dialog box opens, listing the saved representations.
3. In the **Select Rep** dialog box, select a rep from the list of existing representations, and click **OK**.
4. An information window displays such information as the name of the drawing representation, the commands that control view display, the commands that control the loading of the drawing. Click
   File > **Save** to write the information to a file named drawingrep.inf, or choose
   File > **Save As** to write the information to a designated file.

To Execute a Drawing Representation While Retrieving a Drawing

You can execute a drawing representation directly into session while retrieving a drawing, using **Open Rep** in the File Open dialog box.

1. Click **File** > **Open**.
2. Select a drawing from the File Open dialog box (click just once to select the drawing, not to open it) and click **Open Rep**. The Open Rep dialog box opens, listing previously saved drawing representations and the two default drawing representations, All Views and No Views.
3. Select the name of the drawing representation you want to execute, and click **OK**.
4. The system retrieves the drawing and configures the drawing to the selected drawing representation state.

Note: If the drawing is already in session and you close the drawing and then access the **Open Rep** dialog box, clicking **Cancel** brings up the drawing, which defaults to its previous state (either retrieves or does not retrieve models).

Tip: Erasing Views by Menu or by Drawing Rep Tool Dialog Box

The **Erase** and **Display** options in the **Drawing Rep Tool** are functionally distinct from the **Erase View** and **Resume View** commands on the **Views** menu so that you can prevent previously erased views from reappearing when a drawing is retrieved. The **Views** menu opens when you click **View** > **Drawing Display** > **Drawing View Visibility**.

When you erase a view using the **Erase** option in the **Drawing Rep Tool**, you must use the **Display** option in the **Drawing Rep Tool** to display the view again; any views that you erased using the **Erase View** menu command remain erased.

When you erase a view using the **Erase View** menu command, you must use the **Resume View** menu command to display the view again.

Running Drawing Programs

About Creating Drawing Programs

Using the **Tools** > **Drawing Program** commands, you can create a *drawing program* to define how a drawing will adapt to a change in the state of its model. Changes might occur when you re-execute the model with new Pro/PROGRAM inputs, or replace it with another instance from a family of models. Typically, you can distinguish one sequence, or state, from another by the values of some of the parameters in the model.

A drawing program is intended to be used to adapt a drawing to a part or assembly program. It contains logic statements that control the drawing layout and perform various detail functions. For example, a part drawing might include a detailed view of a particular feature, such as a keyway. If you suppress that feature, the system should erase that detailed view, and move the other views to fill the space. If you make a state such as `no_detail_view` that erases the detailed view and organizes (moves) the other views appropriately, the program queries the system to determine if the keyway is suppressed. If so, the drawing displays the `no_detail_view` state. If not, the drawing displays the detailed views of the model.

A drawing program has two portions: *states* and *program text*. A state is a named sequence of familiar procedures, such as showing a dimension or moving a view, that you perform on a drawing to define how it should appear. It is a record of modifications that you would like to make to the drawing. To create a state, you type a name and then record various detail commands. You can *play back* these commands to determine what the drawing state actually does, and then edit it, if necessary. As you create a drawing, you can create additional states and delete others.

You can place dimensions, dimension breaks, and dimension clips on snap lines during the creation of a drawing program state.

Once you have defined the drawing states, you can create a drawing program, as shown in the preceding figure. The drawing program is a text file, embedded inside the drawing, which contains lines of text that set certain states for the drawing, depending on the values of the conditional expressions that you use. You can use IF statements, drawing parameters, and assignment statements to set previously defined drawing states. The program first searches the drawing for a drawing parameter; if it does not find a parameter, it searches the default model for a model parameter. If the drawing program recognizes the drawing parameter, it designates it with the postfix ":d" (that is, drawing_attribute:d). If it is a model parameter, it designates it with the model number.

- Commands to execute a drawing state take the following form: SET STATE *name_of_state.*
- You can nest IF statements in the following form:

```
IF <expression>
    ELSE IF <expression>
    ELSE
    ENDIF.
```

An expression is a logical expression that you could use in a part relation. It could contain drawing attributes (as found in the SET UP menu) or the feature suppressed function (that is, FEAT_SUPPRESSED (model_name, feat_id) to determine if a feature is suppressed; and FEAT_SUPPRESSED (as*sembly_name, comp_id*) to determine if a component is suppressed). For example:

```
IF FEAT_SUPPRESSED (bolt,15)
SET STATE no_detail_view
```

In this case, the system checks to see if feature ID 15 of the model named "bolt" is suppressed. If so, it sets the state to `no_detail_view`; otherwise, it does nothing.

- Assignment statements take the following form: *var_name = expression,* where *var_name* is a variable name
- Comment lines take the following form: `/*` *This is a comment line.*

  Note: The words IF, ELSE, ENDIF, SET, STATE, FEAT_SUPPRESSED are reserved. You cannot use them as variable names in a drawing program. Also, if using the equal sign (=) in an IF statement, use "==". If using a statement to set a parameter value, use "=".

Example: Drawing Program Text

```
IF FEAT _SUPPRESSED (shaft, 12)
SET STATE no_detail_view

ELSE SET STATE all_detail
```

To Create a Record of Modifications

1. Click **Tools** > **Drawing Program**. The Menu Manager opens.
2. Click **Define States** > **Create State**.
3. Type the name of the state. Press Enter.
4. Click **Record Cmds**.
5. From the DWG COMMANDS menu, choose commands to move, erase, and show items in the drawing; then save these changes (later, you can edit the state and highlight those items).
6. To undo any of the modifications that you have made, choose **EDIT STATE** > **Undo Cmds**. The system places the cursor on the command that you last executed, and highlights in the drawing window the item that was acted on. If it highlights a command line in red, it means that the system has *undone* it (the *name* field is not always filled in because not all items have names).
7. After you create the state, the system resets the drawing to its default display. Click **Done** to activate the state and make the changes.

The EDIT STATE Menu

- **Record Cmds**—Displays the DWG COMMANDS menu, a compound menu of possible commands and object types upon which to operate. After choosing an action and item type, you can modify items in the drawing. The following table presents those commands that you can record (marked with an "X"). In addition to those commands included in the table as not being recordable, you cannot record the following: moving leader jogs or dimension jogs; modifying the model grid; erasing, showing, and moving cross-section arrows and view notes; erasing or moving ordinate dimensions; and modifying filled draft cross sections.
- **Play Cmds**—Resets the display of the drawing to its original display (before starting the state), and plays each saved command in the state, pausing in between. You can stop playback by selecting the stop sign icon in the message area. The system skips commands that do not affect any displayed sheet.
- **Undo Cmds**—Removes commands. Displays the Undo State Commands text-tool window listing commands in the state.
- **Layer States**—Displays the DPM LAYER menu. You can blank, display, or set to normal the display status of any drawing layer. You can also leave the status unchanged. Drawing programs can *only* control the status of drawing layers and cannot affect model layers.
- **Set Cur Sheet**—Switches to another sheet of the drawing to modify it. You can also display another drawing sheet by choosing **Window** > **New** from the Pro/ENGINEER menu bar. This command does *not* affect the current state in any way.

Commands That You Can Record (marked with an "X")

DETAIL ITEMS	Move	Edit Attachment	Erase	Show	Show View	Switch Sheet	Create
Dimensions	X	X	X	X	X		X
Reference Dimensions	X	X	X	X	X		X
Notes	X	X	X	X	X	X	X
Balloon Notes	X	X	X	X	X	X	X
BOM Balloons	X	X	X	X	X		
Symbols	X	X	X	X	X	X	
Gtols	X	X	X	X	X	X	
Surf Finish Symbols	X	X	X	X	X	X	
Datum Targets	X		X	X			
Set Datums	X		X	X			
Axes	X		X	X			
Draft Entities	X					X	
Draft Datums	X					X	
Draft Axes	X					X	
Draft Groups	X					X	
Tables	X					X	
Views	X		X	X		X	

The UNDO COMMANDS menu displays these commands:

- **Next**—Executes the command at the next line and moves the cursor to that line.
- **Prev**—Unexecutes the command at the previous line and moves the cursor to it.
- **Run**—Selects a line in the text-tool window. The system executes or unexecutes commands until that line.
- **Toggle Cur**—Undoes the command at the current line. The system updates the drawing window to this, and highlights the command line. If you have already undone the current command, this command restores it.
- **Pick/Toggle**—Selects a command line in the text-tool window. If you choose a command that you have not undone, it undoes it. If you choose a command that you have undone, it restores it.

To Create Detail Items in a Drawing State

To create detail items such as dimensions, notes, and balloon notes in a drawing state, choose **Create** from the DWG COMMANDS menu. Items that you create in one state are not visible in any other state or outside of the drawing program.

To Redefine a Drawing State

To *redefine a state*, click **Tools > Drawing Program > Define States > Edit State**. From the **PRGM STATES** menu, choose the state to edit. If the drawing program is not using the state, Pro/ENGINEER plays it quickly; otherwise, it rolls the program to the point where this state is executed, and displays the **EDIT STATE** menu.

To Remove a Drawing State

To *remove a state*, choose **Delete State** from the DRAW STATES menu. You can delete any state that the drawing program is not currently using.

To Call a User-Defined Function

To call in a user-defined function that is already registered in a Pro/TOOLKIT application, choose **User Func** from the DWG COMMANDS menu.

To Run the Drawing Program (Execute a State)

Along with the EDIT PROGRAM menu, the DRAW PROGRAM text-tool window opens, listing the lines of the program. Using the commands in the EDIT PROGRAM menu, you can interactively add lines to the program, remove lines from it, step forward and backward, and modify it. As you make each change, the system updates the display of the drawing.

Notes:

- You cannot modify a value that the program is driving (such as the value of a drawing user attribute, the sheet on which a draft entity should be). However, you can modify a value that the program is not driving if the program moves the view.
- If you delete an item that the drawing program controls, the system automatically removes from the state the command in the program that controls that item.
- If you convert a view to a snapshot, the system removes from their states all commands in the drawing program that control the view and its subordinate items.
- The drawing program can only modify the drawing, not the models of the drawing.

When the program appears, the system positions the text-tool cursor after the last line executed. As you move through the program, you can change the display by choosing commands from the SET STATE menu. After you exit the DRAW PROGRAM menu, the system executes the program in its entirety and redisplays the drawing. It automatically updates the drawing program to reflect changes made to parts when you retrieve a drawing, switch to Drawing mode, or regenerate by choosing **Draft** from the REGENERATE menu (you can use **Model** *only* if the model changes).

After you use one of the editing commands, the system reinterprets the entire text of the program, and re-executes the program up to the cursor.

- If the interpreting is successful, it converts all parameter names, model names, and keywords to uppercase. In addition, the model postfix of a parameter appears automatically when needed (for example, if the drawing has more than one model).
- If the interpreting is unsuccessful, the system informs you that an error has occurred and highlights the error line. It cannot execute any line that is not interpreted, so if you move the text-tool cursor into an uninterpreted area of the program, it does not execute any states and does not change the drawing. Until you correct the error, the system does not execute the program and make the changes that the program has made to the drawing.

The Edit Program Menu

To create a drawing program, choose **Edit Program** from the DRAW PROGRAM menu. The EDIT PROGRAM menu displays the following commands:

- **Insert Line**—Prompts you to type text lines into the program. To finish, type a carriage return.
- **Delete Line**—Deletes a range of lines from the program.
- **Edit Line**—Modifies the current line in the message area.
- **Screen Edit**—Edits the program directly in the text-tool window.

- **File Edit**—Edits the drawing program using the system editor.
- **Set State—**Displays a run-time menu of all drawing states. You can select a state from the menu, or create a new state interactively.
- **Next**—Moves to the next executable line of the program.
- **Prev**—Backs up one line, undoing the modification that line caused to the drawing. If you get to this line by using the **Jump** command, you back up to the line on which the jump was started.
- **Run**—Executes the program forward or backward until reaching a certain line.
- **Jump**—Sets the current line as a specified line number without executing any line. You can only jump to executable lines.
- **Switch Dim**—Changes the data display to symbols or numerical values. This command does not change the program or the drawing.
- **Set Cur Sheet—**Sets the current sheet of the drawing for display purposes. You can display another drawing sheet by choosing **Window** from the Pro/ENGINEER menu bar, followed by **New**. This command does not change the program or the drawing.

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