

Pro/ENGINEER[®] Wildfire[®] 4.0

**Pro/ENGINEER Fundamentals
Help Topic Collection**

Parametric Technology Corporation

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Pro/ENGINEER Fundamentals

Fundamentals Overview

The Fundamentals Help describes basic functions for all areas of Pro/ENGINEER Wildfire. Refer to these topics for information on using the user interface, on configuring Pro/ENGINEER for Fundamentals, and on performing basic operations.

Tasks for Fundamentals

Working with the User Interface

Using the Main Window

Working with Files

Using Systems Colors

Using Related Applications

Working with the Model

Using the Model Tree

Viewing the Model

Making Selections

Searching for Entities

Creating Cross Sections

Regenerating Models

Integrating Changes

Working with Core Functions

Analyzing Model Parameters

Managing Annotation Features and Annotations

Setting Up Model Properties

Working with Relations and Parameters

Constructing Chains and Surface Sets

Getting Productive

Using the Layer Tree

Using Family Tables

Using Configuration Options

Customizing Pro/ENGINEER

Getting Information

Using Trail Files and Training Files

Configuring Pro/ENGINEER Fundamentals

About Configuring Pro/ENGINEER Fundamentals

You can customize the way you configure Pro/ENGINEER by entering `config.pro` configuration file options and their values in the **Options** dialog box (**Tools > Options**).

Pro/ENGINEER Fundamentals Help provides a list of configuration options arranged in alphabetical order. Each option contains the following information:

- Configuration option name.
- Default and available variables or values. All default values are in italic.
- Brief description and notes describing the configuration option.

To Set Pro/ENGINEER Fundamentals Configuration Options

1. Click **Tools > Options**. The **Options** dialog box opens.
2. Click the **Show only options loaded from file** check box to see currently loaded configuration options or clear this check box to see all configuration options.
3. Select the configuration option from the list or type the configuration option name in the **Option** box.
4. In the **Value** box type or select a value.
Note: The default value is followed by an asterisk (*).
5. Click **Add/Change**. The configuration option and its value appear in the list. A green status icon confirms the change.
6. When you finish configuring, click **Apply** or **OK**.

System Color Options

In the system colors file (`system_colors_file`) within the `config.pro` file, you can provide default RGB values (<0.000000 0.000000 0.000000>) for the display of various graphics. See the next table. The three decimal values (from 0–100) specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue. The RGB values match those in the **R**, **G**, and **B** boxes in the **Color Editor** dialog box. To access this dialog box, **View > Display Settings > System Colors** and click a color in the **System Colors** dialog box.

Configuration Option	Graphics Affected	User Interface Option
system_background_color	background of the graphics window	Background
system_curves_color	curve entities	Curves
system_dimmed_menu_color	unavailable menu items	Highlight - Secondary
system_edge_high_color	edges	Highlight - Edge
system_geometry_color	solid wireframe entities	Geometry
system_hidden_color	hidden, solid wireframe entities	Hidden Line
system_highlight_color	primary highlight color	Highlight - Primary
system_letter_color	datum entity labels or tags	Letter
system_section_color	sections	Sketched Curve
system_sheetmetal_color	sheetmetal parts	Sheet Metal
system_volume_color	Manufacturing volumes	Manufacturing Volume

add_lower_level_comps_to_layer

no, yes

Activates the **Favor Current Layer** option, which allows lower-level components to be added to top-level layers.

af_copy_references_flag

no, yes

Controls whether the **Backup Refs** column in the **Annotation Feature** dialog box is displayed. This column specifies whether the system creates backup references during creation or redefinition of an Annotation Element.

allow_confirm_window

yes, no

Displays a confirmation window when you exit Pro/ENGINEER.

angular_tol

0 0.000000 <integer tolerance>

Sets default angular tolerance dimensions. The integer sets the number of decimal places, and the tolerance is the *actual* tolerance value. For example, *6 0.000025* sets a tolerance to 6 decimal places, and the default tolerance value is 0.000025.

For integer dimensions, the integer value is 0, and the tolerance is an integer. For example, *0 1* sets a tolerance of 1 for integer dimensions.

A value affects only models created after the tolerance option is specified in the configuration file.

auto_ae_param_file

<full path>

Specifies the location of the file in which Annotation Element parameters are automatically added each time you create an annotation element.

autohide_copied_group_af

yes, no

Copies a feature group containing Annotation features and automatically hides from display the Annotation features in the new group.

auto_regen_views

yes, no

Regenerates the drawing display whenever you change from one window to another.

yes—Regenerates both the drawing and the drawing view.

no—Regenerates the drawing but not the view unless you select **View > Repaint**, **View > Update**, or **Edit > Regen**.

autoplace_single_comp

yes, no

Activates the **Component Placement** dialog box in drag-and-drop operations.

bell

no, yes

Rings the keyboard bell when prompts appear.

blank_layer

0

Displays the specified layer as empty, or blank, when you begin a Pro/ENGINEER session.

bmgr_pref_file

<path>

Specifies the location of the graph preference file for the Graph Tool settings, such as axis, line weights, label fonts, and so forth. Pro/ENGINEER uses the settings in the text file to render your graphs to your specifications. After you create the text file, you add the `bmgr_pref_file` option to your `config.pro` file.

A sample graph preference file with the possible values follows:

```
X_Axis_Color 5.019608e-01f,5.019608e-01f,1.000000e+00f
X_Axis_DisplayLabel 1
X_Axis_GridColor 5.019608e-01f,5.019608e-01f,0.000000e+00f
X_Axis_GridEnabled 1
X_Axis_GridStyle 2
X_Axis_LabelColor 1.000000e+00f,1.000000e+00f,1.000000e+00f
X_Axis_LabelEnabled 1
X_Axis_LabelFont graphtool_font
X_Axis_LabelFontHeight 1.500000e-01f
X_Axis_Thickness 4
X_Axis_TickColor 1.000000e+00f,1.000000e+00f,1.000000e+00f
X_Axis_TickFont graphtool_font
X_Axis_TickFontHeight 1.000000e-01f
X_Axis_TickHorizontal 1
Y_Axis_Color 1.000000e+00f,0.000000e+00f,0.000000e+00f
Y_Axis_DisplayLabel 1
Y_Axis_GridColor 5.019608e-01f,5.019608e-01f,0.000000e+00f
Y_Axis_GridEnabled 1
Y_Axis_GridStyle 2
Y_Axis_LabelColor 1.000000e+00f,1.000000e+00f,1.000000e+00f
Y_Axis_LabelEnabled 1
Y_Axis_LabelFont graphtool_font
Y_Axis_LabelFontHeight 1.500000e-01f
Y_Axis_Thickness 2
Y_Axis_TickColor 1.000000e+00f,1.000000e+00f,1.000000e+00f
```

`Y_Axis_TickFont` `graphtool_font`

`Y_Axis_TickFontHeight` `1.000000e-01f`

`Y_Axis_TickHorizontal` `1`

bom_format

`<file path>`

Specifies the location of the BOM format file for a customized BOM.

browser_favorite

`<full directory path name>`

Specifies the directory for the local file system that appears in the Folder navigator in the Pro/ENGINEER browser.

button_name_in_help

`no, yes`

Specifies whether to display Help text associated with a menu command or a button.

clearance_triangulation

`none, low, medium, high`

Specifies the quality of surface triangulation used to obtain initial points for clearance and interference calculations.

clock

`yes, no`

Determines whether a clock is displayed while Pro/ENGINEER is working on a task.

color

`yes, no`

Determines whether the model takes its assigned appearance.

`yes`—Displays the model in the assigned colors.

`no`—Displays the model in white for nonshaded display styles and in light gray for shaded display styles.

color_ramp_size

`-1`

Specifies the number of shades in a color ramp for the display of multicolor, shaded models of simulation or analysis results or both. Your system graphics must support 256 colors and color maps that compress.

color_resolution*0.100000*

Specifies minimum allowable deviation between user-defined colors. Colors having RGB values within the tolerance of existing colors cannot be created. Decreasing this setting allows the definition of colors similar in RGB value.

color_windows*one_window, all_windows*

Specifies in which windows the model displays in color.

all_windows—Displays the model in color in all windows.

one_window—Displays the model in color only in the graphics window. In auxiliary windows, the model appears in the default system color.

compress_output_files*yes, no*

Specifies whether to store object files in a compressed format. Compressed files are slower to read and write, but are 30–50% smaller and are fully compatible across systems.

Note: If you want to copy a compressed object file from a UNIX system to a Windows NT or Windows 95 machine or vice versa, use the operating-system command `rcp -b` from the Windows machine.

conf_mouse_anim*no, yes*

Displays the presenter's mouse movements on the screen during a shared Pro/ENGINEER session.

create_numbered_layers*no, yes*

Creates the default layers numbered 1 through 32.

def_layer*layer_<item_type>*

Adds item types to the specified default layer name. The variables, or item types, follow:

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_feat, layer_gtol, layer_hole_feat, layer_intchg_func, layer_intchg_simp, layer_nogeom_feat, layer_note, layer_parameter_dim, layer_part_refdim, layer_point, layer_protrusion_feat, layer_quilt, layer_refdim, layer_rib_feat, layer_ribbon_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_dec_places

`2`

Sets the default number of decimal places (0-14) for nonangular dimensions in all model modes. It does not affect the decimal places as modified using **Num Digits**. For Sketcher, use the option `sketcher_dec_places`.

default_dim_num_digits_changes

`yes, no`

Sets the default number of digits in a dimension to the last entered value. If you set this option to `no`, the system defaults to the value specified for the configuration option `default_dec_places`.

default_geom_analysis_type

`quick, saved, feature`

Sets the default analysis type in geometry analysis tools. Some geometry analysis tools such as Offset, Draft, Reflection, and Shadow do not have the Feature analysis type. For such tools, Pro/ENGINEER sets the default analysis type to Quick even if you set this configuration option to `feature`.

default_layer_model

`<model name>`

Specifies the name of the model that is used to drive the rule-based layer.

default_placement_surfacefinish

`normal_to_Entity, with_leaders, on_entity, free`

Sets the default placement type for surface finish annotations within the **Surface Finish** dialog box.

normal_to_Entity—Attaches the symbol directly to a surface of the model such that the annotation is placed normal to the selected surface and in the annotation plane. However, if the annotation plane is not normal to the selected entity, then the surface finish annotation too is not normal to the entity.

with_leaders—Attaches the symbol to model surfaces using leaders.

on_entity—Attaches the symbol directly to a surface of the model.

free—Places the symbol without attaching it to model geometry.

disable_search_path_check

no, yes

Controls whether the search path is checked for name conflicts when creating, renaming, or copying models. A check ensures that only models with unique names are in session.

display

shade, wireframe, hiddenvis, hiddeninvis

Displays the model with all lines as specified. The display of lines is reflected in plotter, DXF, and IGES files.

shade—Displays all surfaces shaded except for drawing models.

wireframe—Displays all lines in white.

hiddenvis—Displays hidden lines in grey.

hiddeninvis—Removes hidden lines from the display.

display_annotations

yes, no

Controls the display of annotations in the graphics window in 3D models.

display_axes

yes, no

Displays the datum axes.

display_axis_tags

yes, no

Displays the name tags for the datum axis.

display_coord_sys

yes, no

Displays the datum coordinate systems.

display_coord_sys_tags

yes, no

Displays the name tags for the coordinate system.

display_full_object_path

no, yes

Displays the object's file name (including its object-type suffix and its version number) and its file path in the window title and Model Info display.

no—Displays only the object's name.

yes—Displays the object's full name and its file path.

display_layer

0, <layer ID>

Displays the specified layer when you begin a Pro/ENGINEER session.

display_plane_tags

yes, no

Displays the datum tags.

display_planes

yes, no

Displays the datum planes.

display_point_tags

yes, no

Displays datum point tags.

display_points

yes, no

Displays datum points and their names.

display_tol_by_1000

no, yes

Displays tolerances for nonangular dimensions multiplied by 1000.

display_z_extension_lines

yes, no

Controls the creation of Z-extension lines in driven dimensions.

dm_http_compression_level

0

Sets the data compression factor (0–9, 0 = no compression) for data exchange with a Windchill server. Higher compressions can speed up uploads for a client over a slow network.

edge_display_quality

normal, high, very_high, low

Controls the display quality of an edge for a wireframe and for hidden-line removal by varying the tessellation.

normal—Provides a normal quality of edge display.

high—Increases tessellation by a factor of 2.

very_high—Increases tessellation by a factor of 3.

low—Decreases tessellation compared to normal, thus speeding up the display of an object.

enable_popup_help

yes, no

Enables pop-up Help in the dialog boxes.

fast_highlight

no, yes

Improves the performance of highlighting and reorienting large assembly models, regardless of display style (wireframe, hidden line, no hidden line, and shaded).

no—Uses standard highlight.

yes—Uses fast highlight. When you select components, the model geometry (solid, surface) and associated annotations, curves, and cables are highlighted. Datums of a selected component do not highlight unless they are also selected. For models of modest size and complexity, the slight increase in performance may not warrant the additional memory required to support this option.

file_timestamp_format

%d-%Mmm-%yy %Hh:%mi:%ss %PM

Controls the format of timestamps when you view a listing of files and directories in Web pages generated by Pro/ENGINEER in the Pro/ENGINEER browser and in dialog boxes. The format for the value of the configuration option is a string consisting of

up to seven part: the year, month, and date, the hours, minutes, seconds, and the AM or PM designator. You can specify the parts in any order.

The default value is `%dd-%Mmm-%yy %Hh:%mi:%ss %PM` in most languages. However, in Japanese-language sessions, the default value is `%yyyy/%mm/%dd %hh:%mi:%ss`.

You can use the following formats when specifying the value of the timestamps:

`%yyyy`—4-digit year (for example 2005)

`%yy`—2-digit year (for example 05)

`%MMM`—Month (for example SEP)

`%Mmm`—Month (for example Sep)

`%Month`—Full month name (for example September)

`%mm`—Month number, padded to 2 digits (for example 09)

`%m`—Month number (for example 9)

`%dd`—Date, padded to 2 digits (for example 05)

`%d`—Date, no padding (for example 5)

`%Hh`—Hour, 12-hour format (for example 12)

`%hh`—Hour, 24-hour format (for example 00)

`%PM`—AM or PM

`%mi`—Minutes

`%ss`—Seconds

file_open_default_folder

`working_directory`, `in_session`, `my_documents`, `pro_library`, `workspace`, `commonsplace`

Sets default directory from which to open a file when using **File > Open**.

`working_directory`—Searches the working directory.

`in_session`—Searches objects in session.

`my_documents`—Searches the **My Documents** folder.

`pro_library`—Searches the Pro/Library directory in Pro/LIBRARY.

`workspace`—Searches the Workspace in PDM application.

`commonsplace`—Searches the Commonsplace in Pro/INTRALINK.

`default`—Searches for the `My Documents` folder on Windows or for the working directory on UNIX. Subsequently, when you click **File > Open**, Pro/ENGINEER opens the directory where the previous **File Open** dialog box was closed. In a linked session with a PDM application, searches the active workspace.

flip_arrow_scale*1.000000*

Sets a scale factor for enlarging the flip arrow for the direction for feature creation.

force_new_file_options_dialog*no, yes*

Forces the use of the **New File Options** dialog box when you click **File > New**. The **Use default template** check box in the **New** dialog box becomes unavailable.

full_hlr_for_cables*full, partial, none*

Determines whether hidden lines are shown or hidden with cables.

full—Removes the hidden lines from view when cables hide other geometry (only available when Hidden line is active).

partial—Cables hide other noninterfering cables, except when cables are routed together between the same locations.

none—Hidden lines are displayed, so the display process is faster.

general_undo_stack_limit*50*

Sets the number of undo or redo operations. If the number of operations exceeds 50, the first operation in the stack of operations is removed first, and so on.

global_appearance_file*<directory name>*

Sets the default directory for the `global.dmt` file. Use the full path to avoid problems.

highlight_geometry*yes, no*

Specifies whether items you select on the Model Tree are highlighted in the graphics area.

highlight_layer_items*yes, no*

Specifies whether items you select in the Layer Tree highlight in the graphics area.

info_output_format

html, text

Sets the default format type for presentation of system information.

info_output_mode

both, choose, file, screen

Sets the default method of presenting system information for audit trails, BOMs, names, and models. All other output displays on the screen.

both—Displays the output and writes it to a file.

choose—Displays the **INFO OUTPUT** menu, so you can choose the method of output.

file—Writes the output to a file.

screen—Displays the output.

kbd_cmd_abbreviation

off, on

Enables or disables the use of abbreviations when you enter commands from the keyboard.

kbd_selection

no, yes

Enables or disables the keyboard for selecting locations in the graphics area. If set to *no*, use of the mouse is required.

linear_tol

0 0.000000-1

Sets the default linear tolerance dimensions. The integer sets the number of decimal places, and the tolerance is the actual tolerance value. For example, *6 0.000025* sets a tolerance to 6 decimal places, and the default tolerance value is 0.000025.

For integer dimensions, the integer would be 0 and the tolerance would be an integer. For example, *0 1* sets a tolerance of 1 for integer dimensions.

Any modifications to these options affect only new models that are created after the option modification.

lods_enabled*no, yes*

Uses level of detail (lod) in shaded models during dynamic orientation (panning, zooming, and spinning). During runtime, you can override this setting by clicking **Tools > Level of Detail > Environment**.

maintain_limit_tol_nominal*no, yes*

Maintains the nominal value of a dimension regardless of the changes that you make to the tolerance values.

mdl_tree_cfg_file*<path>*

Specifies the Model Tree configuration file to be loaded when you start Pro/ENGINEER.

measure_dec_places*-1, <integer>*

Controls the number of digits displayed after the decimal point for the measure analysis tools. If you do not change the default of *-1*, Pro/ENGINEER uses the setting of the *measure_sig_figures* configuration option.

measure_sig_figures*6*

Sets the number of significant figures in the results when you use the **Model Size** dialog box from the **Info** menu. The maximum value is 11.

menu_font*<variables>*

Specifies the font used in the Pro/ENGINEER menu bar, menus, and all their children. Provide the comma-delimited variables in any order, for example, *italic bold, 24, times* or *24,times, italic bold* have the same effect. Any omitted variable uses the standard setting.

menu_mapper_location*<directory path>*

Sets the path to the Menu Mapper. When you click **Help > Menu Mapper**, Pro/ENGINEER launches the Menu Mapper from the specified location.

The default path is

http://www.ptc.com/community/proewf3/menu_mapper/en/index.htm.

mesh_spline_surf

no, yes

Displays the blue mesh surface lines.

model_notes_as_labels

no, yes

Determines whether model notes display as full text (default) or labels.

model_tree_start

yes, no

Displays the Model Tree with its model (default).

native_kbd_macros

no, yes

Specifies support of keyboard macros in a native language, for example, German, rather than only English.

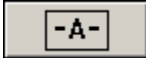

number_user_colors

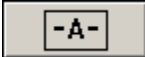
200

Specifies the maximum number of user-defined colors that are available within the **Appearance Editor** and the **Entity Colors** dialog boxes.

old_style_set_datum

yes, no

Specifies whether to display  in the **Datum** or **Axis** dialog box. The  button allows you to create set datum tags with the old-style display.

Note: Even if the value of `old_style_set_datum` configuration option is set to *no*, the  appears in the **Datum** or **Axis** dialog box when modifying a pre-existing old-style set datum.

online_resources_location

<directory path>

Sets the path to Online Resources. When you click **Help > Online Resources**, Pro/ENGINEER launches the Online Resources page from the specified location.

The default path is `http://www.ptc.com/community/proewf3/newtools/index.htm`.

open_simplified_rep_by_default

no, yes

Opens the **Open Rep** dialog box by default if you click **Open** on the **File Open** dialog box.

orientation

trimetric, isometric, user_default

Establishes the initial default view position, or orientation. After you set the configuration options for *x_axis* and *y_axis*, the system defaults to the user-defined values. To override the orientation at runtime, click **View > Orientation > Standard Orientation**.

trimetric—Orients the model trimetrically.


isometric—Orients the model isometrically.


user_default—Orients the model in the position specified in the configuration options *x_axis* and *y_axis*. If you do not define these options, the system defaults to *trimetric*.

orientation_style

dynamic, anchored

Sets the default viewing style irrespective of whether the Orient mode is enabled or disabled, that is, you are outside Orient mode or in the Orient mode, respectively. When Orient mode is enabled, you may change the viewing style as required.

dynamic—The Orientation Center is displayed as . Orientation is updated as the pointer moves. The model spins freely about the Orientation Center.

anchored—The Orientation Center is displayed as . The orientation is updated as the pointer moves. Model rotation is controlled by the direction and distance the pointer is moved from its initial position. The Orientation Center changes color at each 90-degree interval. When the pointer returns to the original position, the view is reset to where you started.

override_store_back

no, yes

Stores all retrieved objects in the current working directory.

no—Stores objects in their original directories. If you do not have write permission to the original directory, the configuration option `save_object_in_current` takes effect.

yes—Stores objects in the current working directory.

parenthesize_ref_dim

no, yes

Encloses reference dimensions in parentheses. If set to *no*, the reference dimensions are followed with the text "REF."

pick_aperture_radius

7.000000

Specifies the size of the area about the mouse when making selections. Units are 1/1000 of screen size.

plot_names

yes, no

no—Gives plot files, except PostScript plots, the extension `plt`.

yes—Gives all plot files descriptive extensions:

`hp`—For Hewlett-Packard

`hp2`—For Hewlett-Packard `hpgl2`

`cal`—For Calcomp

`ver`—For Versatec

`ger`—For Gerber photoplotters

`ps`—For PostScript (including color)

preferred_save_as_type

`*.prt *.igs *.set *.vda *.neu *.stp *.ntr *.ct *.cat *.stl *.iv *.obj
*.slp *.unv *.wrl *.enm *.evs *.mdb *.edn *.emp *.evs *.edp *.gbf *.asc
*.facet *.sat *.model *.ed*x_t *.ed*.jpg *.shd *.eps *.tif*.pic *.zip`

Sets the order of the file types to customize the **Type** list in the **Save a Copy** dialog box.

prehighlight

yes, no

Highlights selectable items beneath the pointer before selection. Prehighlighting provides a visual check to confirm that you will select the intended item.

prehighlight_tree

no, yes

Specifies whether the selectable items beneath the pointer on the Model Tree or the Layer Tree are highlighted before you select them.

pro_colormap_path

<directory path>

Specifies the directory path for a color map (.map) file to be loaded from disk. Use the full path to avoid problems.

pro_crosshatch_dir

<directory name>

Specifies a default directory for your crosshatch library in which you can save crosshatching patterns for later retrieval. Use the full path of the default directory.

pro_editor_command

<command>

Enables an editor other than the system editor when the optional editor has been specified as the value for the options `drawing_file_editor` or `relation_file_editor`.

The command specified will be executed as it is typed in the `config.pro` file. If the command does not open a new window, you can start the editor in the system window used to start Pro/ENGINEER.

pro_material_dir

<directory name>

Sets the default directory for the part material library. Use the full path to avoid problems. For example, `/home/users/library/material`.

pro_plot_config_dir

<directory name>

Sets the directory of your user-defined plotter configuration file. Use the full path to avoid problems. For example, `/home/users/plot_dir`.

pro_unit_length

`unit_inch`, `unit_foot`, `unit_mm`, `unit_cm`, `unit_m`

Sets the default units for new objects.

pro_unit_mass

`unit_pound`, `unit_ounce`, `unit_ton`, `unit_gram`, `unit_kilogram`, `unit_tonne`

Sets the default units for mass for new objects.

prompt_on_erase_not_disp

no, yes

Displays a prompt so you can choose whether to save undisplayed objects before they are erased. This option is used with the **Erase > Not Displayed** command on the **View** menu.

no—Erases all undisplayed objects without a prompt.

yes—Prompts you to choose whether to save undisplayed objects.

prompt_on_exit

no, yes

Prompts you whether to save objects when you exit a Pro/ENGINEER session. Your objects are not saved unless you set the option to *yes*.

propagate_change_to_parents

no, yes

Determines which parent models to save when the option `save_objects` is set to `changed` or `changed_and_specified`.

no—Saves only parent models that have actually been changed.

yes—Saves any model that is a parent of a changed model.

provide_pick_message_always

no, yes

Displays a description of an item in the message area after each selection.

no—Displays descriptions for an item only while querying

yes—Displays descriptions for items in all case, even when Query is not used. Queries include preselection highlighting, **Next** and **Previous**, and selections.

quick_print_drawing_template

<path and drawing template name>

Specifies the path and name of a drawing template to be used by the **File > Quick Print** command.

quick_print_plotter_config_file

<path name to the plotter .pcf file>

Specifies the path and name of the default plotter configuration file (*.pcf) to be used by the **File > Quick Print** command.

If `plotter` is set to `ms_print_mgr`, no other options are necessary. If it isn't set to `ms_print_mgr`, adding to the plotter configuration file the values for `plot_file_dir` and `plotter_command` is recommended.

read_famtab_file_on_retrieve

no, yes

no—Ignores `filename.ptd`.

yes—Creates and saves `filename.ptd` and uses that file on generic retrieval.

regen_backup_using_disk

no, yes

Specifies whether the current model is backed up before each regeneration.

relation_file_editor

<editor>

Sets the editor used to edit relations.

relation_text_trail_output

yes, no

Includes or excludes changes in the text area of the **Relations** dialog box in the trail file.

relation_tool_mapkey_behavior

Increment, full_output

Controls how Pro/ENGINEER runs mapkeys in the **Relations** dialog box.

Increment—Records the text set added or removed in the text area and saves it to the mapkey. Subsequently, on running the mapkey, adds or removes text incrementally to the current text set in the text area.

full_output—Records the original text set in the text area and saves it to the mapkey. Subsequently, on running the mapkey, the current text set in the text area is replaced with the recorded text set.

Note: You cannot record mapkeys if the value of `relation_text_trail_output` is set to `no`. However, you can run a mapkey that was recorded with this configuration option set to `yes`, regardless of this options's setting when you run the mapkey.

relations_num_const_units

yes, no

Checks for units in a relation, issues a warning if units are missing, and prompts you to apply units.

If you want to add a relation to nonsolid models, such as layouts and bulks, you must always specify units for numeric constants. The setting of this configuration option is ignored for nonsolid models.

relations_units_sensitive

yes, no

Checks for units when solving relations.

yes—Takes units into account. If units are missing, a warning appears.

no—Ignores units.

rename_drawings_with_object

none, part, assem, both

Controls whether the system automatically copies drawing files associated with the part or assembly object types. The drawing files must have the same name as their objects.

none—Excludes the associated drawing when saving copies.

part—Copies the associated drawings of a part.

assem—Copies the associated drawings of an assembly and its components.

both—Copies the associated drawings for both parts and assemblies.

To ensure that only objects with unique names are in session, use the default *no* for `disable_search_path_check`. The system checks the search path for objects with duplicate names.

restricted_gtol_dialog

yes, no,

Controls the restrictions in the **Geometric Tolerance** dialog box when picking certain GTOL types.

yes—Adheres to standards.

no—Drops all restrictions.

restricted_val_definition

<file name>

Specifies the location and name of the external file that contains definitions of the restricted-value parameters. Use the full directory path and name.

retain_display_memory

yes, no

Determines whether the display of an object on the screen is kept in memory when you quit the window. The default *yes* speeds up the retrieval of objects.

retrieve_data_sharing_ref_parts

no, yes, ignore_missing

Retrieves the referenced parts for dependent features with shared data, such as Inheritance, External Copy Geometry, External Shrinkwrap, and External Merge.

no—Ignores referenced parts in the retrieval.

yes—Prompts the user to accept each referenced part during the retrieval.

ignore_missing—Skips any missing referenced part, sends a message to that effect, and continues the retrieval process.

save_clipped_view

no, yes

Specifies whether to save the model in the view clipped state.

save_display

no, yes

Displays the geometry and detail items, such as solid dimensions, in View-Only mode. Use this option to decrease model retrieval time. To override this setting during runtime, click **Tools > Environment**, and then select or clear **Save Display** under **Default Actions**.

save_instance_accelerator

saved objects, none, explicit, always

Determines how instances are saved with the Family Tables of solid parts.

saved objects—Saves instance accelerator files if,

- The instance is modified in one of the following ways.
 - Feature redefinition
 - Feature rerouting
 - Reference replacement
 - Feature reordering
 - Component replacement
 - Object integration
- The instance verification status is not set to Failed.

- The instance verification status is set to Unverified. In this case, when creating the accelerator file, Pro/ENGINEER displays a message stating that the accelerator file was created for the nonverified instance.

none—Does not save instance accelerator files.

explicit—Saves instance accelerator files only when you explicitly save instances.

always—Always saves instance accelerator files, regardless of whether you are saving an instance explicitly or through a higher-level object.

You can override this configuration option at runtime by clicking **File > Instance Operations**, and clicking another command on the associated **INST DBMS** menu.

save_model_display

shading_lod, wireframe, shading_high, shading_low

Sets the amount of graphical data stored in the assembly and part files. By default, the system always stores wireframe data in both parts and assemblies. The only information that the system saves in the assembly `.asm` file is the display setting of components that are intersected by assembly features. The system also saves the tessellated data for the low, medium, high, and lod options. The display setting at the time the assembly is saved is stored in the part and assembly files.

shading_lod—Saves all levels of tessellation according to the setting in the **View Performance** dialog box.

wireframe—Stores wireframe data in both parts and assemblies for a wireframe of the components without shading.

shading_high—Saves the most detail in a finer tessellation for a shaded version of the components.

shading_low—Saves the least detail in a coarser tessellation for a shaded version of the components.

save_modified_draw_models_only

yes, no

Determines whether the system saves the model after you have changed it. If set to *no*, the system saves the model every time that you store the drawing.

save_object_in_current

no, yes

Saves the object retrieved from a directory where you do not have write permission. See the `override_store_back` option. The objects to be saved are set by the `save_objects` configuration option.

no—Does not save the object.

yes—Saves the object in the current directory.

save_objects

changed_and_specified, *all*, *changed*, *changed_and_updated*

Determines when an object and its dependent objects, such as a part used in an assembly, are stored.

changed_and_specified—Saves the top-level object plus any modified, dependent objects.

all—Stores all objects.

changed—Stores only modified objects.

changed_and_updated—Stores changed and modified objects.

save_unchanged_pdm_object

as_ref, *as_copy*

Determines how to save an object retrieved from a Pro/PDM database. The object is saved in the current working directory.

as_ref—Saves the object as a reference only, that is, as a pointer to the Pro/PDM database.

as_copy—Saves the unchanged object.

search_path

<directory path>

Specifies an ordered list of directories in which to search for object or file retrieval. These directories, along with the current (working) directory and any directories in the `search.pro` file (see the configuration file option `search_path_file`) make up the Pro/ENGINEER search path.

Separator Characters and Search Paths

You must enclose in quotation marks any search path with a separator character (space, comma, or semicolon) in a directory name, for example:

- Windows: `search_path "C:\Program Files\proe2001\models"`
- UNIX: `search_path "/home/user/smith/odd,dir;name"`

Note: For Windows NT, omit the last backslash (\) from the path, or enclose the path in quotation marks, or add a trailing space after the backslash.

Relative and Absolute Paths

The directory path names can be relative or absolute. You can use special characters, such as "." in UNIX and Windows, in specifying a relative path name.

Relative path names are initially resolved relative to the startup directory. If you subsequently reload the configuration file, the system reevaluates the relative path

names relative to the current working directory and appends the new directories (if any) to the search path (the previous path remains in place).

It is better, therefore, to specify the full path names always (in other words, from root) so as to avoid problems if you change working directories or use the same configuration file in another startup directory.

Using More Than One Path

The option can have several path names on a single line, separated by commas, semicolons, or spaces. Whichever delimiter you choose, you must then use consistently. The option can appear any number of times in the configuration file, so it is not necessary to have more than one path name to a line. If objects with the same name are stored in more than one search-path directory, the system retrieves the first one that it finds, regardless of which object is the most recent.

Previously Defined Environment Variables

Search paths may also include previously defined environment variables. This is done by preceding the variable with \$ in the search path definition. For example, the environment variable `OBJ_TYPE` can be used as follows:

```
search_path /partlib/$OBJ_TYPE/objs
```

search_path_file

<path>

Specifies the location of the `search.pro` file, which contains a list of directory path names. These directories, along with the current working directory, and any directories specified by the configuration option `search_path`, make up the Pro/ENGINEER search path.

In the `search.pro` file, you specify an individual directory path on each line, starting with the first line in the file. Blank lines and comment lines (which begin with `!`) are permitted.

You can specify either the path for the `search.pro` file or just the path to the directory containing that file. In the latter case, the system looks for `search.pro` in that directory. Use the full path rather than a relative one to avoid problems if you change working directories or use the same configuration file in another startup directory.

Separator Characters and Search Paths

You must enclose in quotation marks any search path with a separator character (space, comma, or semicolon) in a directory name, for example:

- Windows: `search_path "C:\Program Files\proe2001\models"`
- UNIX: `search_path_file "/home/user/smith/odd,dir;name/search.pro"`

Note: For Windows NT, omit the last backslash (`\`) from the path, or enclose the path in quotation marks, or add a trailing space after the backslash.

sel_insts_on_comp_retrieval*no, yes*

Prompts you to choose an instance when you retrieve instances of a family of assemblies and the table-driven components used in the assembly instances are themselves generics.

select_on_dtm_edges*all_modes, sketcher_only*

Specifies the method of selecting a datum plane. If you use Query frequently, set this option to *sketcher_only*.

all_modes—Click the visual boundary of the datum plane.

sketcher_only—Click the tag of the datum plane in modes other than Sketcher.

set_trail_single_step*no, yes*

Enables a trail file to be single-stepped by pressing ENTER.

shade_mfg_ref_models*yes, no*

Determines whether to process reference parts when generating the shaded image.

no—Does not shade reference parts (to save time).

yes—Shades reference parts.

shade_surface_feat*yes, no*

Displays surface features with shading.

shade_with*curves, no*

Displays datum curves on shaded objects.

show_dim_sign*no, yes*

Shows positive or negative values for dimensions. Dimensions for coordinate systems and datum point offsets always show negative or positive values, even if this option is set to *no*.

no—Displays positive dimensions by creating the geometry to the opposite side, if you enter a negative value for the dimension.

yes—Displays negative dimensions by creating the geometry to the same side, if the dimension you modify is negative, and if you enter a negative value.

show_selected_item_id

no, yes

Specifies whether to display IDs of the items listed in the **Pick From List** dialog box for the queried model geometry or dimensions.

show_shaded_edges

no, yes

Controls the brightness of edge color compared to the surfaces to which they belong when shading objects.

no—Displays the edges of the same color.

yes—Displays brighter edges.

spherical_map_size

256x256, 512x512, 1024x1024

Specifies the size of the texture image of the spherical map (resolution) to be used for realtime rendering. Increasing the image size affects performance but improves quality of the image.

Note: This option is valid only in the OpenGL graphics mode and for graphics cards that do not support cubic environment mapping.

spin_center_display

yes, no

Determines whether the spin center symbol is displayed.

To override the display setting during runtime, click **Tools > Environment**, and then select or clear **Spin Center** under **Display** in the **Environment** dialog box.

spin_rate_zoom_sensitive

no, yes

Controls the sensitivity of the model to rotation.

yes-Reduces the sensitivity of the model to rotation.

spin_with_orientation_center

yes, no

Displays the orientation center while reorienting the model.

spin_with_part_entities*no, yes*

Displays datum features during the dynamic spinning of a model.

spin_with_silhouettes*no, yes*

Displays silhouette lines during the dynamic spinning of a model.

start_model_dir

<directory path>

Provides the full path to the directory containing start parts and assemblies. For example:

```
start_model_dir C:\Users\Johndoe\Pro\Start_Models
```

```
start_model_dir/users/johndoe/pro/start_models
```

system_background_color

<0.000000 0.000000 0.000000>

Defines the background color of the graphics window. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue.

system_colors_file

<path>

Specifies the full path within the `config.pro` file that sets the default color of the graphics. To change the colors in session, click **View > Display Settings > System Colors**, and then choose a tab in which to change a system color.

In the system colors file, you can define default RGB values (<0.000000 0.000000 0.000000>) for the various graphics. See the next table. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue. The RGB values are identical to those in the **R**, **G**, and **B** boxes in the **Color Editor** dialog box. To access this dialog box, click a color in the **System Colors** dialog box.

Configuration Option	Graphics Affected	User Interface Option
system_background_color	background of the graphics window	Background
system_curves_color	curve entities	Curves
system_dimmed_menu_color	unavailable menu	Highlight -

Configuration Option	Graphics Affected	User Interface Option
	items	Secondary
system_edge_high_color	edges	Highlight - Edge
system_geometry_color	solid wireframe entities	Geometry
system_hidden_color	hidden, solid wireframe entities	Hidden Line
system_highlight_color	primary highlight color	Highlight - Primary
system_letter_color	datum tags	Letter
system_section_color	datum entity labels or tags	Sketched Curve
system_sheetmetal_color	sheet metal parts	Sheet Metal
system_volume_color	Manufacturing volumes	Manufacturing Volume

system_curves_color

```
<0.000000 0.000000 0.000000>
```

Defines the default color for curve entities. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines medium blue.

system_dimmed_menu_color

```
<0.000000 0.000000 0.000000>
```

Defines the default color of unavailable menu items. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue.

system_edge_high_color

```
<0.000000 0.000000 0.000000>
```

Defines the default color in which edges are highlighted. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue.

system_geometry_color

```
<0.000000 0.000000 0.000000>
```

Defines the default color of solid, wireframe entities. The three decimal values specify in order the percentage of red, green, and blue in the resulting color. For example, 0 0 49 defines a medium blue.

system_hidden_color

```
<0.000000 0.000000 0.000000>
```

Defines the default color for the display of hidden, solid wireframe entities. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue.

The system ignores this option if `use_part_color_for_hidden_lines` is set to `yes`.

system_highlight_color

```
<0.000000 0.000000 0.000000>
```

Defines the default color for the primary highlight color. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue.

system_letter_color

```
<0.000000 0.000000 0.000000>
```

Specifies the default RGB color for datum tags. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 specifies a medium blue.

system_section_color

```
<0.000000 0.000000 0.000000>
```

Specifies the default color for sections. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue.

system_sheetmetal_color

```
<0.000000 0.000000 0.000000>
```

Specifies the default RGB value for sheetmetal parts. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 specifies a medium blue.

system_volume_color

```
<0.000000 0.000000 0.000000>
```

Specifies the default color for Manufacturing volumes. The three real numbers from 0–100 specify a percentage of red, green, and blue, respectively, in the resulting color. For example, 0 0 49 defines a medium blue.

tangent_edge_display

solid, no, centerline, phantom, dimmed

Determines how edges between tangent surfaces are displayed.

solid—Displays edges as solid lines.

no—Does not display edges.

centerline—Displays edges in centerline font.

phantom—Displays edges in phantom font.

dimmed—Displays edges in a dimmed system color.

template_drawing

c_drawing.drw

Specifies the file name of the default drawing template.

template_solidpart

<filename>

Specifies the file name of the default drawing template for a part.

tol_display

no, yes

Displays dimensions without or with tolerances.

tol_mode

limits, nominal, plusminus, plusminussym, plusminussym_super

Sets the default display for dimension tolerances. Groups brought in from other models carry the tolerance display mode that was in effect when they were created.

limits—Displays dimensions as upper and lower limits.

nominal—Displays dimensions without tolerances.

plusminus—Displays dimensions as a nominal with plus-minus tolerances (the positive and negative values are independent).

plusminussym—Displays dimensions as a nominal with a single value for both the positive and negative tolerances.

plusminussym_super—Same as *plusminussym*. However, displays the plus-minus tolerance value at the half-way point of the dimension text and extending upwards.

tolerance_class*fine, medium, coarse, very_coarse*

Sets the default tolerance class for ISO-standard models. The system uses the tolerance class in conjunction with the dimension value when retrieving tolerances for general or broken-edge dimensions.

tolerance_standard*ansi, iso*

Sets the tolerance standard used when creating the model.

tolerance_table_dir

<directory path>

Sets the default directory for user-defined tolerance tables for ISO-standard models. All Holes and Shafts tables overwrite existing tables when loaded.

trail_delay*0*

Sets a delay in seconds between steps in a trail file.

trail_dir

<directory path>

Creates the trail file in the specified directory rather than in the startup directory.

use_8_plotter_pens*no, yes*

Specifies whether to support up to 8 plotter pens. The initial default is 4 pens.

use_part_color_for_hidden_lines*no, yes*

Uses a dimmed, user-defined part color for hidden lines and ignores the setting for `system_hidden_color`.

use_software_linefonts*no, yes*

no—Plots lines using the line font that most closely resembles the font used in Pro/ENGINEER.

yes—Plots the exact line style used in Pro/ENGINEER, dot by dot, dash by dash, and space by space.

warn_if_iso_tol_missing

no, yes

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

no—Pro/ENGINEER does not display a warning message.

yes—Pro/ENGINEER displays a warning message if a tolerance value in the selected table is missing.

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**.

web_browser_history_days

20

Stores history records for the Pro/ENGINEER browser for the number of days specified.

web_browser_homepage

<http://www.ptc.com/community/proewf/newtools/index.htm>

Specifies the path of the home page for the Pro/ENGINEER browser.

windows_scale

1.000000

Scales Pro/ENGINEER windows with a given coefficient from 0.050000 through 1.000000. A value of 0.85 is usually adequate to allow dynamic menus to display to the right of the Pro/ENGINEER window.

x_angle

0.000000

Sets a default view orientation in degrees for models. The default orientation depends on which option—*x_angle*, *y_angle*, or *orientation*—is last in the configuration file. If none is used, the default is **trimetric**. See the *orientation* option.

Note: If these variables are in the configuration file, the settings appear in the **Orientation** dialog box: choose **View > Orientation**, select **Preferences** under **Type**, and look under **Default Orientation**.

y_angle

0.000000

Sets a default view orientation in degrees for models. The default orientation depends on which option—`x_angle`, `y_angle`, or `orientation`—is last in the configuration file. If none is used, the default is **trimetric**. See the `orientation` option.

Note: If these variables are in the configuration file, the settings appear in the **Orientation** dialog box: choose **View > Orientation**, select **Preferences** under **Type**, and look under **Default Orientation**.

User Interface Basics

Working with Pro/ENGINEER Windows and Dialog Boxes

About the Pro/ENGINEER Main Window

Each Pro/ENGINEER object that you open appears in its own Pro/ENGINEER window. You can perform many operations from the top-level menus in multiple windows without canceling pending operations. Only one window is active at a time, but you can still perform some functions in the inactive windows. To activate a window, click **Window > Activate** or press **CTRL+A**.

The Pro/ENGINEER main window consists of a navigation area, the Pro/ENGINEER browser, the menu bar, toolchests, and information areas. The main window also contains the graphics window, where your model is displayed.

Navigation Area

The navigation area includes the Model Tree and Layer Tree, Folder browser, Favorites, and Connections.

Pro/ENGINEER Browser

The Pro/ENGINEER browser provides access to internal and external Web sites.

Menu Bar

The menu bar contains commands for creating, saving, and modifying models, and for setting your Pro/ENGINEER environment and configuration options.

You can customize the menu bar by adding, removing, copying, or moving commands, or by adding icons to or removing them from the menu items.

Note: Commands not appropriate to the active window are unavailable or not visible.

Toolchests

The toolchests, at the top, right, and left of the Pro/ENGINEER window, can contain toolbars, buttons, and menus. You can customize the content and location of the toolchests using the **Customize** dialog box.

Information Areas



Each Pro/ENGINEER window has a message area and a status bar. In addition, screen tips appear when you pass the pointer over menu names, menu commands, toolbar buttons, and certain dialog box items.

Dashboard

The dashboard contains a message area in which one-line messages related to work in a window are displayed. Use the standard scroll bars in the message area to review past messages.

Status Bar

When applicable, the status bar displays the following information:

- Warning and error shortcuts related to **Tools > Console**
- Number of items selected in the current model
- Available selection filters
- Model regeneration status,  , which indicates that the model must be regenerated, or  , which indicates that current processes have been halted
- Screen tips

About the Menu Manager

The Menu Manager is a series of cascading menus that you use to perform some tasks within Pro/ENGINEER. The **Menu Manager** menus change depending on the mode. Some of the **Menu Manager** menu options are duplicated in the menu bar menus.

Customizing the Menu Manager

You can customize the Menu Manager by creating a `menu_def.pro` file and storing it in your working directory.

Note: If a menu default entry has already been defined for a particular menu in the loadpoint directory `menu_def.pro` file, any conflicting definitions in the local `menu_def.pro` file (in the working directory) are ignored.

About the Dashboard

The dashboard is a context-sensitive area at the bottom of the Pro/ENGINEER window that guides you through the modeling process. As you select geometry in the graphics window and set your preferences, the dashboard narrows your available options, so you make only targeted modeling decisions. The dashboard consists of the dialog bar, slide-up panels, message area, and the control area.

Dialog Bar

You perform most of your modeling tasks in the graphics window and the dialog bar. When you activate a tool, the dialog bar displays commonly used options and collectors.

Slide-up Panels

To perform advanced modeling actions or retrieve comprehensive feature information, use the slide-up panels. Click one of the tabs above the dialog bar, and its slide-up panel opens. Because both the tabs and their corresponding slide-up panels are context sensitive, different tabs and panel elements are displayed, depending on the current modeling context. Default values are provided in some cases.

To open another panel, click its tab. To close a panel, click its tab and the panel slides back into the dashboard.

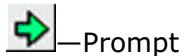
Message Area

As you work with your model, Pro/ENGINEER confirms your actions and guides you through the modeling process with text messages in the message area below the dialog bar. The message area contains all messages for the current modeling session. To find an earlier message, scroll the message list or drag the sash to expand the message area.

Text messages address two conditions:

- System functions
- Modeling actions

Each message is preceded by an icon to indicate the category of the message:



—Prompt



—Informational



—Warning



—Error



—Critical

The message window continues to display messages even if you pause a tool and the dashboard is unavailable.

Control Area

The control area of the dashboard contains the following elements:





—Pauses the current tool, temporarily returning you to the default system state in which you can make selections. Any features that you create while the original tool

is paused are placed in a Group in the Model Tree along with the original feature when completed.



—Resumes a paused tool.



—Activates Verify mode for the feature displayed in the graphics window. To stop Verify mode, click  again or click . The check box, when selected, activates dynamic preview, which you can use to view your changes to the model as you perform them.



—Complete the tool with the current settings.




—Cancels the current tool.


To Change the Position of the Dashboard

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens.
2. Click the **Options** tab.
3. Under **Dashboard position**, click **Above graphics area** or **Below graphics area** (the default).
4. Save the new settings in the `config.win` using one of the following methods:
 - Select **Automatically save to** (selected by default), and then accept the default `config.win` file name. You can also enter a new file name or navigate to the `config.win` file in which you want to save the settings.
 - Click **File > Save Settings**. In the **Save Toolbar Configuration File** dialog box, you can accept the default file name and path, or you can type a new file name or navigate to the `config.win` file in which you want to save the new settings. Path names can contain up to 260 characters.
5. Click **OK** to close the **Customize** dialog box and accept your changes.

About Entering Data

When you are prompted to enter data in the message area, a dialog box, or a slide-up panel, you can either accept the default value or enter a new value. Complete the data entry in one of the following ways:

- Press ENTER.
- Click  or **OK**.
- Middle-click.

Note: You can usually cancel data entry operations by pressing ESC or clicking **Cancel** or .

Cutting and Pasting Text in Windows

You can use the mouse or keyboard to cut, copy, and paste text from a system window, a Pro/ENGINEER window, or a message window. However, you cannot cut and paste either prompt text in the message area or online Help text.

Entering Arithmetic Expressions

When you are prompted for a numeric value, you can enter an arithmetic expression involving numbers, functions, operations, and parameter symbol names. For example:

```
Enter value [2.53]: 2*((3/8)/7)+ceil(d5)
```

The following basic arithmetic operators are supported:

- + addition
- - subtraction
- / division
- * multiplication
- () grouping, for example, $d0 = (d1 - d2) * d3$
- ^ exponentiation

Working with Fractions

Dimension entry and display can be in decimal or fractional format. If you are working with fractions instead of decimal values, you must enter the number as an equation. For example, enter the value $1 \frac{1}{8}$ as $1+1/8$. This value appears as $1-1/8$. You can create fractions automatically using configuration file options, or you can work in decimal format and convert specific dimensions to fractions.

About Dialog Boxes

Most dialog boxes support a set of related functions. For example, the view reorientation, zooming, panning, spinning, and spin-center functions are all available in the **Orientation** dialog box (**View > Orientation > Reorient**).

Many dialog boxes are also dynamic, and options change as you make selections. For example, the **Orientation** dialog box changes depending on the selection under **Type**.

As a rule, a function in the dialog box is available if:

- The function is appropriate for the state of the window or object or for the product type (Part, Assembly, Sketcher, Layout, and so on).
- The software required for using the function is installed. For example, if Pro/INTRALINK is installed, certain buttons on the **File Open** dialog box are available.

Performing Default Actions

Most dialog boxes in Pro/ENGINEER have a default button, denoted by a black border. You can perform the default action quickly in three different ways:

- Click the button.
- Middle-click within the Pro/ENGINEER window.
- Press ENTER.

Using Accelerators and Mnemonics

Accelerators and mnemonics are keyboard shortcuts to commands.

Using Accelerators

An accelerator is a key sequence for activating a menu command. To use an accelerator, press and hold the appropriate keys, such as CTRL or ALT, and then press the appropriate letter key. The key sequences for accelerators appear next to the commands on the menus. For example:

CTRL+P	Print
CTRL+S	Save
CTRL+Z	Undo
CTRL+ALT+A	Select All

Using Mnemonics

Mnemonics are letters that you use as shortcuts to traverse menus in the main menu bar. Mnemonics reduce mouse motion and provide a fast alternative if you prefer keyboard input.

Mnemonics are indicated by an underlined letter in the menu name. For example, the mnemonic for **File** is F, and the mnemonic for **Open** is O.

Press and hold ALT, press F.	The File menu appears
Press and hold ALT, press F, then O.	The File Open dialog box opens.

Shortcut Menus

Pro/ENGINEER provides shortcut menus of frequently used commands appropriate to the current context. You can access a shortcut menu by right-clicking.

Note: To open a shortcut menu in the graphics window, press the right mouse button for a short time.

Shortcut menus are accessible in the following areas:

- Graphics window

- Model Tree
- Some dialog boxes with lists of items
- Toolchests
- Message area
- Some status bar items
- Any area where you can perform an object-action operation.

Selection

About Query and Preselection Highlighting

You must select design items (datums or geometry) to be able to work with your model. This selection process usually begins with a query. You query design items to locate a specific item upon which to perform an action. As you move your pointer over the model and query, geometry and datums are highlighted and tooltips provide additional identification. This preselection highlighting enables you to accurately target the design item that you want to select. After you locate the item, you select it and begin designing.

Query

Query model geometry by placing your pointer over the model geometry and right-click. With each right-click, Pro/ENGINEER cycles through each geometrical item that is located directly under the pointer from top (closest to the pointer) to bottom. The queried item name is displayed in the Status bar. You can continue to query until you have located the item that you want to select. Remember that querying is very helpful if another item is above the item that you want to select.

Another way to query items is to use the **Next** and **Previous** shortcut menu commands or the **Pick From List** dialog box. The **Pick From List** dialog box contains a list of items that are located directly under the pointer, organized from top to bottom. Subassemblies are listed after the part-level components in the list. You can select the item from the **Pick From List** dialog box and begin designing your model.

Note:

- Pro/ENGINEER can display IDs of the items listed in the **Pick From List** dialog box for the queried model geometry or dimensions. Set the `show_selected_item_id` configuration option to `yes` to display these IDs (default is `no`).
- Pro/ENGINEER can display a message in the message area that indicates each queried or selected geometry item. You must set the `provide_pick_message_always` configuration option to `yes` to display these messages (default is `no`).

Preselection Highlighting in Graphics Window

As you move your pointer over an item or query an item, Pro/ENGINEER highlights the item. This type of highlighting is called preselection highlighting and provides a visual confirmation of the design item enabling you to accurately target the item that you want to select. As you move the pointer, Pro/ENGINEER highlights the top-most item located directly under the pointer. As you query, each item is highlighted from top (closest to the pointer) to bottom.

Note: Preselection highlighting is enabled by default. If you disable it, you must use a different selection method.

Preselection Highlighting in Model Tree and Layer Tree

When preselection highlighting is enabled using **Show > Preselection highlighting** and you move your pointer over selectable Model Tree or Layer Tree items, that is, the tree nodes, then the tree nodes below the pointer appear underlined in the current text color indicating preselection highlighting of the nodes. The geometry corresponding to the selectable tree items below the pointer is displayed in the graphics window using preselection highlight presentation.

When preselection highlighting is enabled by using both **Show > Preselection highlighting** and **Edit > Select > Preferences** and you move your pointer over selectable items in the graphics window, then the corresponding tree nodes appear underlined, indicating preselection highlighting of the nodes.

A Note About ToolTips

As you move the pointer over model geometry, a tooltip displays the name of the preselection highlighted or queried item in the Status Bar and in the graphics window. These Tooltips provide additional item identification. You can disable the graphic window tooltips by setting the `enable_popup_help` configuration option to `no` (default is `yes`).

About Selection

You must select a design item (datums or geometry) in order to work on your model. You can select an item before or after a feature tool is activated. To select an item, you place your pointer over the item in the graphics window. If another item is above it, you can query the item. After the item preselection highlights, click it. You can select multiple items by pressing the CTRL key as you click. Pro/ENGINEER builds a list of selected items or a "selection set" and indicates the number of items in the selection set in the Selected Items area located on the Status bar. For example, if you select three items, `3 selected` is displayed in the Selected Items area. You can double-click the Selected Items area to open the **Selected Items** dialog box. This box contains the names of all items in the selection set. You can view the selection set and remove selected items.

If you are selecting while working in a feature tool, each tool has specific selection requirements that must be met. These requirements are governed by filters and collectors. To make querying and selecting easier, Pro/ENGINEER provides filters that narrow the range of selectable items. These filters are located in the Filter box on the

Status bar. After you select items and open a feature tool, Pro/ENGINEER places the selected items in collectors.

Note:

- Preselection highlighting is enabled by default. If you disable it, you must use a different selection method. Refer to the topic under *See Also* for more information.
- By default, when you select an item in the Model Tree or Layer Tree, geometry corresponding to the selected tree item is highlighted in the graphics window. You can disable this highlighting of geometry using **Show > Highlight Geometry** in the Model Tree or Layer Tree.

Clearing a Selection

As you select items, you may want to clear the selection of an item from the selection set, chain, or surface set. You can clear a selection in the following ways:

Working outside of a tool

- Holding down the CTRL key, click individual items to clear each one. For example, individual items in a surface set.

Note: To clear the selection of the individual items from the ends of a chain, hold down the SHIFT key and click each item. To clear the selection of the entire chain, hold down the CTRL key and click the chain.

- Remove the item using the **Selected Items** dialog box.
- Clear the entire selection set, chain, or surface set by clicking in a vacant area in the graphics window.
- Right-click the Selected Items area and click **Clear** on the shortcut menu to clear the entire selection set.

Working inside of a tool

- Clear a selected item or clear all items in the active collector either by using the **Clear** shortcut menu command from the graphics window, or the **Remove** or **Remove All** shortcut menu commands from within the collector itself.
- Holding down the CTRL key, click individual items to clear each one from the collector. For example, individual items in a chain or surface set, or the entire chain or surface set.
- Right-click the Selected Items area and click **Clear** on the shortcut menu to clear the selected items.

Working with a Selection

After you finish selecting, you can begin working on the selection set by doing any of the following:

- Activate a feature tool by clicking a toolbar button enabling you to directly manipulate the item in the graphics window, or to use the Dashboard. Note that all selections inside a feature tool are governed by the requirement of that tool.

For example, you may only need to select one item to satisfy a Primary collector but multiple items for another collector.

- Right-click and use the shortcut menu commands to perform an action on the selection set.
- Use the menus on the Menu bar to perform an action on the selection set.

Note: If you cancel a feature tool, Pro/ENGINEER restores the selection set that existed immediately before you opened the tool. This enables you to perform modeling actions on the selection set without having to recreate it.

About Filters

Pro/ENGINEER provides various filters to help you select items. These filters are located in the Filter box on the Status Bar. Each filter narrows the item types that you can select enabling you to easily locate the item. All filters are context-sensitive, so that only those filters that are valid for the geometrical context or that are required to meet feature tool requirements are available. Additionally, Pro/ENGINEER automatically selects the best filter according to the context. Remember that you can always explicitly change the filter by simply selecting another one from the Filter box.

Smart Filter

Pro/ENGINEER automatically selects the Smart filter, if it is available. The Smart filter enables you to select the most common types of items that are valid for the current geometrical context. This filter works on two levels: As you begin the selection process, the Smart filter allows you to select general, higher-level items, such as features or components. After you select the higher-level items, the Smart filter automatically narrows the selection scope enabling you to select more specific items, such as edges or faces. This entire process occurs automatically and results in less time explicitly changing filters and more time modeling in the graphics window.

As with any other filter, simply select a different filter from the Filter box, if you do not want to use the Smart filter.

Note: The Smart filter is not available if you disable preselection highlighting (enabled by default).

About Collectors

As you work with your model while in a feature tool, all selections are governed by the requirements of the tool. So, you must select a certain number and type of required items to create the feature. Pro/ENGINEER uses filters and collectors to guide you in making correct selections.

After you select the items, they are collected as references (not the actual geometry) and placed in collectors. For example, you select a surface on your model as the primary placement reference for a hole and then activate the Hole tool.

Pro/ENGINEER automatically opens the tool, collects the reference and places it the **Primary** placement collector. Now, you need to select secondary placement

references to satisfy the **Secondary references** collector (and to constrain the hole feature). At this time, you satisfied the basic hole requirements and you are free to close the tool and create the hole. However, you will want to finish designing the hole to your specifications.

So, collectors function as storage bins for selected references that your feature tool requires. They also help you in making correct selections.

Collector Types

Pro/ENGINEER provides the following types of collectors:

- **Summary Collectors**—Indicate that the required reference or references that you selected. Summary collectors are usually located on the dialog bar and always indicate the selection frequency and type. For example, you select 2 surface sets, or two edges and a curve, and the Summary collector indicates *2 Surface Sets, or 3 Selected*. Remember that a Summary collector is always associated with its Detail collector, however not all feature tools have Summary collectors.
- **Detail Collectors**—Contain detailed information about the required reference or references that you selected. Detail collectors are usually located on the slide-up panels and always contain detailed information such as feature name, type, and values. For example, you select a surface on your model as the primary placement reference for a hole. The **Primary** collector in the Placement slide-up panel contains *Surf:F5 (PROTRUSION)*. All feature tools have Detail collectors.

Note: Collectors are only available after a feature tool activates.

About Working with Collectors

As you work in a feature tool, you must actively work with collectors. You can do the following:

- **Activate a collector**—You activate a collector in order to work with it. You can click in a Detail collector to activate it or use a shortcut menu command. You can also activate a Summary collector by clicking in it. Remember that if you activate a Detail collector, the associated Summary collector (if available) also activates and vice versa.
- **Select new references**—After you activate a collector, simply select new items to add to the existing ones or to replace them.
- **Change values, types, and other parameters**—You can change reference values, type, and other parameters by working in the Detail collector.
- **Clear the active collector**—You can clear all references from the active collector using the following methods:
 - While working in the graphics window, activate the collector and right-click. After the shortcut menu appears, select **Clear**. Pro/ENGINEER clears all references from the active collector.

- Simply, place your pointer in the collector and right-click. After the shortcut menu appears, select **Remove All**. Pro/ENGINEER removes all references from the collector. Note that multiple references must be present in the active collector to use **Remove All**.
- **Remove one reference from the active collector**—You can remove one reference from the active collector using the following methods:
 - Press CTRL and click each selected item on the model. Pro/ENGINEER removes the associated reference from the active collector.
 - Place your pointer in the active Detail collector, select the item to remove and right-click. After the shortcut menu appears, select **Remove**. Remember that your pointer must be in the Detail collector before you right-click.
- **Report reference information**—You can report detailed information for the selected reference. Simply select the reference from the active Detail collector and right-click. After the shortcut menu appears, select **Information**. Pro/ENGINEER opens the browser and displays reference information that can include feature element data and dimensions, layers, and parent and child feature data. Remember that your pointer must be in the active Detail collector before you right-click. Note that you cannot report reference information about chains or surface sets.

To Select Items

This topic assumes that preselection highlighting is enabled (default setting). If you disable it, you must use a different selection method. Refer to *About Disabling Preselection Highlighting* under *See Also* for more information.

1. Place your pointer directly over the item that you want to select and right-click. Pro/ENGINEER prehighlights each queried item that is directly under your pointer from top (closest to the pointer) to bottom. The queried item name displays in the Status bar and as a tooltip.
2. After you locate the item, click it. The selected item highlights in the graphics window and in the Model Tree. Pro/ENGINEER builds a selection set and indicates 1 Selected in the Selected Items area on the Status bar.
3. If you want to add an additional item to the selection set, press CTRL and move the pointer over the item or use query the item, then click. Both selected items highlight in the graphics window and in the Model tree. The Selected Items area indicates 2 Selected. Repeat this step to add more items.
4. If you want to select an existing Group feature, click an item contained in the Group feature, right-click and use **Select Group** from the shortcut menu. Pro/ENGINEER selects the Group feature and adds it to the selection set.
5. If you want to remove an item from the selection set, place your pointer over the highlighted item and press CTRL+click. Pro/ENGINEER clears the item and removes it from the selection set.

6. With the selected items highlighted, right-click and use the shortcut menu commands or open a feature tool to perform an action on the selected items.

Note:

- You must press (and hold) CTRL to select multiple items. Otherwise, Pro/ENGINEER clears all selected items from the selection set and replaces them with the last selected item.
- All selections inside a feature tool are governed by the requirement of that tool. For example, you may only need to select one item to satisfy one collector but multiple items for another collector.

Tip:

- Before you begin selecting items, make certain that the correct filter is selected.
- You can always click a vacant area in the graphics window to clear the selection set. You can also use **Clear** from the shortcut menu.
- If you are not in a feature tool and you select only one item, and the item surface (not the outline) highlights, click a vacant area in the graphics window to clear the selection. Then, start selecting again.
- You can also view the selection set or remove items from the selection set by double-clicking the Selected Items indicator on the Status bar. After the **Selected Items** dialog box opens to display the selection set, simply select the items to remove and click **Remove**.
- You can right-click the Selected Items area and click **Clear** on the shortcut menu to clear the entire selection set.
- Pressing CTRL+double-click selects or removes an item and activates Edit enabling you to change dimension values and properties for the selected items.

Selection Actions

You can use various actions to select items (geometry and datums). The table below lists the main selection actions:

Action	Description
Click	Selects individual items to add to the selection set or tool collector.
Double-click	Activates the Edit mode enabling you to change dimension values or properties for the selected items. Note: For actions inside of a feature tool, refer to the documentation for the tool.
CTRL+click (press and hold CTRL as you click)	Selects additional items to be included in the same selection set or tool collector.

Action	Description
	Clears a selected item and removes it from the selection set or from the tool collector.
CTRL+double-click (press and hold CTRL as you double-click)	Combines CTRL+click and Double-click in one action. Note: For actions inside of a feature tool, refer to the documentation for the tool.
SHIFT+click (press and hold SHIFT as you click)	After you select an edge or curve, activates the chain construction mode. After you select a solid surface or quilt, activates the surface set construction mode.
Right-click	Activates the shortcut menu.
SHIFT+right-click (press and hold SHIFT as you right-click)	Queries through possible chains based on the anchor that you selected.

About Disabling Preselection Highlighting

Preselection highlighting provides a visual confirmation of the design element enabling you to accurately target the element that you want to select. Pro/ENGINEER enables preselection highlighting by default. However, you can disable it, if necessary.

Disabling Preselection Highlighting

You disable preselection highlighting by clearing the **Preselection highlighting** check box in the **Selection Preferences** dialog box (**Edit > Select > Preferences**). Note that you can also set the `prehighlight` configuration option to `no` (default is `yes`).

Selection Behavior

If you disable preselection highlighting, the Pro/ENGINEER selection behavior changes. To select items, you must use the following selection method:

- **Pick**—You select items directly from the model. You can activate the Pick mode by using the **Pick** command from the **Edit** menu (**Edit > Select > Pick**) or from the graphics window shortcut menu. Pro/ENGINEER selects this method by default if you disable preselection highlighting.
- **Query**—You select items that are listed in the **Pick From List** dialog box. You can activate the Query mode by using the **Query** command from the **Edit** menu (**Edit > Select > Query**), or from the graphics window shortcut menu.

Remember that after you finish your selection using **Query**, Pro/ENGINEER returns to the Pick selection mode.

Note: If you disable preselection highlighting, you cannot use the Smart filter.

To Select with Preselection Highlighting Disabled

After you disable preselection highlighting, you can use one of two following selection methods to select items.

1. If you want to use the Pick selection method to select items, place your cursor in the graphics window, right-click and select **Pick** from the shortcut menu, or from the **Edit** menu (**Edit > Select > Pick**). You can now select items directly from the graphics window using the standard selection capabilities, such as pressing CTRL to select multiple items. Pro/ENGINEER builds a selection set and indicates the number of items in the set on the Status bar. Notice that you cannot query items by using the Pick selection method.

Note: Pro/ENGINEER selects this method by default if preselection highlighting is disabled.

2. If you want to use the Query selection method to select items, place your cursor in the graphics window, right-click and select **Query** from the shortcut menu, or from the **Edit** menu (**Edit > Select > Query**). Do the following:
 - Click the area on the model over which the item you want to select is located. The **Pick From List** dialog box appears listing all items that are directly under your pointer from top (closest to the pointer) to bottom.
 - Right-click to query the item. Pro/ENGINEER highlights the queried item in the graphics window and in the **Pick From List** dialog box.
 - Middle-click to select the item. Pro/ENGINEER builds a selection set and indicates *1 Selected* on the Status bar. The **Pick From List** dialog box also closes and the selected item highlights in the graphics window and on the Model Tree. Notice that you can also click **OK** from the **Pick From List** dialog box to select an item.
 - If you want to select additional items, right-click and select **Query** from the shortcut menu. Then, press CTRL and click the additional items from the Model Tree. Note that you cannot select additional items in the graphics window or in the **Pick From List** dialog box when using this selection method.
 - In the graphics window, right-click and use the shortcut menu commands to perform the preferred action on all selected items.

Note: After you select an item, Pro/ENGINEER returns to the Pick selection method (the default method). You must select **Query** from the shortcut menu to continue using this method.

Tip: After the **Pick From List** dialog box appears, you can move your pointer over a new location and click to change the items on the **Pick From List** dialog box.

3. Right-click and use the shortcut menu commands or open a feature tool to perform an action on the selected items.

Note: If you disable preselection highlighting, you cannot use the Smart filter.

Tip:

- Before you begin selecting items, make certain that the correct filter is selected.
- You can always click a vacant area in the graphics window to clear any selections.
- You can also view the selection set or remove items from the selection set by double-clicking the Selected Items indicator on the Status bar. After the **Selected Items** dialog box opens to display the selection set, simply select the items to remove and click **Remove**.

Chains and Surface Sets

Chains

About Chains

Chains consist of multiple edges or curves that are related, for example, by a common vertex or by tangency. You select these related edges or curves and place them into a group or chain. Chains enable you to efficiently perform modeling actions on all of the selected edges or curves in that chain or on multiple chains at one time.

You can construct chains and use them at any time (within a tool or before entering a tool) during your modeling session. Remember, anytime you want to construct a chain, you must first select a reference, and then press (and hold) SHIFT to activate the chain construction mode. Pro/ENGINEER provides ToolTips, messages, and labels that act as a visual aid to guide you through the chain construction process.

To modify chains, you use the **Chain** dialog box. This dialog box contains a list of chains that you constructed using the active part and chain attributes and options.

Chain Types

You can create the following types of chains:

Non-rule based

- **One-by-One Chain**—Chain of individual edges, curves, or composite curves that you select. You can also use a One-by-One chain if you want to model additional edges or curves along with other chains that you constructed. You usually create a One-by-One chain if the elements that you need to select are not part of the same feature, such as a datum curve, or if the elements exist across multiple features. Note that some applications may place other conditions on the resulting chain, such as tangency.
- **Intent Chain**—An intent chain is one that is defined and preserved by the event that created it rather than by the specific entities that it contains. For a simple extrusion with a section composed of entities forming a single closed loop, all the

edges generated by the section define an intent chain. If you add or remove entities from the loop, the intent chain is automatically updated as well as any feature that references it.

For example, if you select the edges resulting from the intersection of two solid extrusions as an intent chain to create a round, then the intent chain and the round are automatically updated if you change the sections for the extrusions.

Rule-based

- **Partial Loop Chain**—Chain that begins at a start-point, follows an edge, and ends at the end-point of the edge or curve segment that you select (also known as a From-To chain). You can construct a curve, surface, or quilt boundary Partial Loop chains.
- **Tangent Chain**—Chain defined by the selected item and the extent to which adjacent entities are tangent.
- **Complete Loop Chain**—Chain that contains an entire loop of curves or edges that binds the curve, quilt, or solid surface to which it belongs, or binds a portion of it as defined by two curves or two edges.

Note: Although points and vertices are considered to be chains by some applications, this is an exception. Most tools discard points and vertices when transferred along with chains into the primary collector. You cannot explicitly convert points and vertices outside of a feature tool into chains, nor can they be extended or modified in any way as chains.

To Construct a Chain

1. Select an edge or curve on your model to establish the anchor. Pro/ENGINEER highlights the selected edge or curve.
2. Press the SHIFT key and move the pointer over to an adjacent segment. The preview of the one-by-one chain consisting of the anchor and the adjacent segment is displayed.
3. Holding down the SHIFT key, right-click and do one of the following actions:
 - Click **Pick From List** on the shortcut menu. The **Pick From List** dialog box opens which contains all the possible chains that you can construct using the anchor and the adjacent segment. Select the desired chain from the list. The preview of the chain is displayed in the graphics window. Click **OK** to accept the selected chain and close the **Pick From List** dialog box.
 - Click **Next** or **Previous** on the shortcut menu to preview the next or previous chain available for selection without opening the **Pick From List** dialog box. Click to accept the selected chain.
 - Right-click to query all possible chains associated with the anchor. Valid edges and curves for each possible chain are highlighted and a ToolTip identifies the chain type, for example *Tangent*, *Surface Loop*, or *One-by-One*. Note that the pointer must be near the selection and not over a handle to be able to query possible chains. If you are constructing a From-To chain, place the pointer over

the edge or curve where the chain will terminate and right-click to query through the possible chains until you locate the From-To chain.

Click the highlighted item associated with the chain type that you want to create. Pro/ENGINEER constructs the chain and highlights it.

Note: If you are working with a tool closed, the Selected Items area indicates the selection and the **Selected Items** dialog box contains the chain. If you construct a chain from inside a tool, the active collector contains the chain and the `Chain` label is displayed on the model.

4. If you are constructing a One-by-One chain, holding down the SHIFT key, click on the anchor again, and select additional edges or curves to include in the chain. Pro/ENGINEER appends and highlights the chain. Alternatively, holding down the SHIFT key, move the pointer over to an adjacent segment. The one-by-one chain consisting of the anchor and the adjacent segment are pre-highlighted. Click to accept the one-by-one chain. Select other segments to add to the one-by-one chain.
5. If you want to construct additional chains during the same workflow, release SHIFT, hold down the CTRL key and click an edge or curve on your model to select an anchor for the new chain. Release CTRL repeat step 2 through step 5.
6. Open a feature tool or continue working in a tool to use the chain to create a feature.

Tip:

- You can remove an entire chain from the selection set by holding down the CTRL key and clicking the chain to remove it.
- After you close a tool, Pro/ENGINEER restores the selection set that you constructed before opening the tool. You can continue to use your chains in other tools, without having to reconstruct them.
- When you construct a Partial Loop chain, that is, a From-To chain using direct selection in the graphics window, Pro/ENGINEER remembers the first selected edge or curve as the anchor reference and the second selection as the extent reference. If the extent reference is a one-sided edge and is the same as the anchor, then when you query this edge, **Surface loop from to** appears once in every query cycle. If the extent reference is a two-sided edge and is the same as the anchor, then when you query this edge, **Surface loop from to** appears twice in every query cycle.
- You can use a continuous intent edge to construct a One-by-one chain.
- To exclude a continuous intent edge or a composite curve located at the end of a chain, query the edge or curve and select it.

To Modify Chain Types and References

Note: This topic discusses chain modification while working inside of a tool and using the **Chain** dialog box. If you want to modify a chain while working outside of a tool, you can use the SHIFT key to clear the selection of individual items from the ends of

a chain or use the CTRL key to remove an entire chain from the selection set. Refer to *About Selection* under *See Also* for more information about clearing a selection.

1. Construct a chain and open the tool in which to work with the chain, or select a feature that contains an existing chain that you want to modify, right-click, and select **Edit Definition** from the shortcut menu.
2. Locate the Detail collector that contains the chain that you want to modify. Detail collectors are located on the slide-up panel. Note that you cannot use the Summary collector (located on the dialog bar) to modify a chain.
3. Click **Details**. Pro/ENGINEER pauses the tool, opens the **Chain** dialog box, and displays the attributes for the chain.
4. If the slide-up panel Detail collector contains more than one chain, select the chain to modify from the chain list in the **Chain** dialog box. The chain attributes are displayed.
5. Click the **References** tab to modify the chain type and references.
6. Click **Standard** to change the selected chain to a One-by-One chain or modify this chain type. The **References** tab contains the following option.

References collector—Contains the references for the selected chain. References consist of one or more adjacent edges or curves, or a single-piece Intent chain. To modify a reference, click the reference to activate the collector and then select a new reference from the model, select additional references, or remove the reference. Notice that you can also use the **References** shortcut menu command from inside the graphics window to activate this collector.

Note:

- All references in the collector are ordered according to the internal direction of the chain with the start point at the top.
 - You can remove references only at the top and bottom of the list. Additionally, if the collector contains only one reference, this reference cannot be removed. However, you can select another reference to replace it.
7. Click **Rule-based** to change the selected chain to a Tangent, Partial Loop (From-To), or Complete Loop chain, or to modify these chain types. The **References** tab now contains the following:

Anchor collector—Contains the anchor reference for the selected chain. Anchor references consist of an edge or a curve. To modify the anchor, click the reference to activate the collector and select a new anchor reference from the model. Notice that you can also use the **Anchor** shortcut menu command from inside the graphics window to activate this collector.

Note: You can only replace the anchor reference, you cannot remove it.

Rule—Displays the following chain types enabling you to modify them:

- **Tangent**—Constructs a tangent chain by using all edges that are tangent to their neighboring edges and tangent to the anchor, if possible. Note that if an anchor reference is used it must be an edge.
- **Partial loop**—Constructs a Partial Loop (From-To) chain by using all loop reference items (curves or edges that partially bind the quilt or solid surface to which it belongs) located between the anchor (start-point) and the extent (end-point) references. The **Extent Reference** collector and the **Flip** button appear. Note that you must select an anchor to use the **Partial Loop** option.
- **Complete loop**—Constructs a Complete Loop chain by using an entire loop of reference items (curves or edges that completely bind the curve, quilt, or solid surface to which it belongs). The **Loop Reference** collector appears. Note that you must select an anchor to use the **Complete loop** option.

Extent Reference collector—Contains the extent (end-point) reference for a Partial Loop chain. Extent references consist of an edge or a curve that can belong to a valid loop reference (solid surface, quilt, or curve). To modify the chain, click the reference to activate the collector and then select a different extent reference from the model. Notice that you can also use the **Extent Reference** shortcut menu command from inside the graphics window to activate this collector.

Note: You must select the **Partial Loop** chain type to use this collector.

Range—Displays the **Flip** button. The **Flip** button changes a portion of the loop reference in a Partial Loop (From-To) chain. Note that you must select the **Partial Loop** chain type, and more than one Partial Loop chain must be formed by the selected anchor, extent, or loop reference in order to use this button.

Loop Reference collector—Contains the loop reference for the **Complete Loop** chain. Consists of a quilt or a solid surface loop reference. To define a new loop range, click the reference to activate the collector and then select a new loop reference from the model. Notice that you can also use the **Loop Reference** shortcut menu command from inside the graphics window to activate this collector.

8. Click **OK**. Pro/ENGINEER saves the changes, closes the **Chain** dialog box, and resumes the tool.

Tip:

- You can construct a chain by clicking **Add** in the **Chain** dialog box and selecting chain references from the model.
- To remove a chain from the Chain list, select the chain and click **Remove** in the **Chain** dialog box.
- Remember that you can hold down the CTRL key and select a reference to either add an additional reference or to clear a selected reference.
- To clear references in the active collector, click **Remove** or **Remove All** on the shortcut menu that appears when you right-click in the collector. Alternatively,

click **Clear** on the shortcut menu that appears when you right-click in the graphics window.

- To easily locate references in the **Chain** dialog box, place your pointer over the reference in a collector. Pro/ENGINEER dynamically highlights the reference on the model.
- Notice that as you modify a chain in the **Chain** dialog box, Pro/ENGINEER dynamically displays the changes in the graphics window.
- When the **Chain** dialog box is open, you can perform Undo or Redo operations if the tool or feature supports Undo or Redo operations.

To Trim or Extend a Chain

Note:

- This topic discusses chain modification while working inside of a tool and using the **Chain** dialog box. If you want to modify a chain while working outside of a tool, you can use the SHIFT key to clear the selection of individual items from the ends of a chain or use the CTRL key to remove an entire chain from the selection set. Refer to *About Selection* under *See Also* for more information about clearing a selection.
 - You can trim or extend open-ended chains (One-by-One, Tangent, and Partial Loop).
 - You can trim closed-loop chains but you cannot extend them.
1. Construct the chain and open the tool in which to work with the chain, or select a feature that contains an existing chain that you want to modify and click **Edit Definition**.
 2. Locate the Detail collector that contains the chain that you want to modify. Detail collectors are located on the slide-up panel. Note that you cannot use the Summary collector (located on the dialog bar) to modify a chain.
 3. Click **Details**. Pro/ENGINEER pauses the tool, opens the **Chain** dialog box, and displays the attributes for the chain.
 4. If the slide-up panel Detail collector contains more than one chain, select the chain to modify from the chain list in the **Chain** dialog box. The chain attributes are displayed.
 5. Click the **Options** tab. If you have selected a closed-loop chain, then the **Start Point** check box appears selected by default.
 6. For closed-loop chains, if required, modify the start point using the **Start Point** collector and the **Flip** button. Clear the **Start Point** check box. The options under **Length Adjustment** become available.
 7. To trim or extend End 1 or End 2 of a chain, use one of the following options under **Length Adjustment**:

- **Value**—Trim or extend a chain using a value. Type a distance value and press ENTER, or select a most recently used value from the list. Note that negative values trim a chain. You can also drag the handle on your model to adjust the value. To add a relation to a feature for the extend values of chain ends when creating or redefining a feature, perform the following steps:
 - a. Type an expression as the value of for a length adjustment. For example, $d1*0.5$.
 - b. Press ENTER. If the expression returns a positive value, then Pro/ENGINEER prompts you for confirmation about whether you want to add the expression as a relation to the feature.
 - c. Click **Yes** to create the relation in the feature. This relation controls the extend value. The **Length Adjustment** options list becomes read-only and displays the value derived from the relation as a read-only value.

Note: After you have created the feature, you can modify the relation using the **Relations** dialog box that opens when you click **Tools > Relations**.
 - **Trim at Reference**—Trim a chain using a reference. Valid references consist of items that intersect the chain (surfaces, planes, curves, edges, axes, and so forth), or datum points or vertices that lie on the chain. This option activates the Trim at Reference collector enabling you to select a reference up to which the chain is trimmed. Notice that you can also use the **Trim at Reference** shortcut menu command from inside the graphics window to activate this collector.
 - **Extend to Reference**—Extends a chain using a reference. Valid references consist of items that intersect the chain (surfaces, planes, curves, edges, datum points, and so forth). This option activates the Extend to Reference collector enabling you to select a reference up to which the chain is extended. Notice that you can also use the **Extend to Reference** shortcut menu command from inside the graphics window to activate this collector.
8. To trim a closed-loop chain, click **Excluded** and select an edge or curve from the start point of the chain.
9. Click **OK**. Pro/ENGINEER saves the changes, closes the **Chain** dialog box, and resumes the tool.

Tip:

- Notice that as you modify a chain in the **Chain** dialog box, Pro/ENGINEER dynamically displays the changes in the graphics window.
- To clear references in the active collector, click **Remove** or **Remove All** on the shortcut menu that appears when you right-click in the collector. Alternatively, click **Clear** on the shortcut menu that appears when you right-click in the graphics window.
- To easily locate references in the **Chain** dialog box, place your pointer over the reference in a collector. Pro/ENGINEER dynamically highlights the reference on the model.

To Add Relations for Extend Values of Chain Ends

1. To add a relation to a feature for extend values of chain ends when you create or redefine a feature using the graphics area, right-click the feature on the Model Tree and click **Edit** on the shortcut menu.
2. Double-click the dimension value in the graphics area and specify the length adjustment value as an expression. For example, $d1*0.5$.
3. Press ENTER. If the expression returns a positive value, then Pro/ENGINEER prompts you for confirmation about whether you want to add the expression as a relation to the feature.
4. Click **Yes** to create the relation in a feature. Pro/ENGINEER derives the length adjustment value from the relation and displays it as a read-only value in orange. The relation controls the length adjustment value of the chain.

Note: You can also add a relation to a feature for extend values of chain ends using the **Chain** dialog box when creating or redefining a feature.

About Validating Chains Dynamically

When you create or edit chains Pro/ENGINEER dynamically validates the chain. If the chain is valid the **Dynamic validation** box is automatically selected. When dynamic validation is enabled, adding new references to the active collector or removing references from the active collector does not make the active chain invalid. Note that you can disable dynamic validation at any time.

If the chain is invalid the **Dynamic validation** box is automatically cleared. When dynamic validation is disabled, you can add new references to the active collector or remove references from the active collector without restrictions. However, this may make the active chain invalid. If the chain becomes invalid then the **Dynamic validation** box becomes unavailable.

To Append or Exclude Items from a Chain

Note:

- This topic discusses chain modification while working inside of a tool and using the **Chain** dialog box. If you want to modify a chain while working outside of a tool, you can use the SHIFT key to clear the selection of individual items from the ends of a chain or use the CTRL key to remove an entire chain from the selection set. Refer to *About Selection* under *See Also* for more information about clearing a selection.
 - You can append items or exclude items in rule-based chains. However, all tools may not support the appending and excluding of items.
1. Construct the chain and open the tool in which to work with the chain, or select a feature that contains an existing chain that you want to modify and click **Edit Definition**.

2. Locate the Detail collector that contains the chain that you want to modify. Detail collectors are located on the slide-up panel. Note that you cannot use the Summary collector (located on the dialog bar) to modify a chain.
3. Click **Details**. Pro/ENGINEER pauses the tool, opens the **Chain** dialog box, and displays the attributes for the chain.
4. If the slide-up panel Detail collector contains more than one chain, select the chain to modify from the chain list in the **Chain** dialog box. The chain attributes are displayed.
5. Click the **Options** tab.
6. Click in the **Appended** or **Excluded** collector to append items or exclude items, respectively. Clicking the collector activates the collector. Notice that you can also use the **Appended Items** or **Excluded Items** shortcut menu command from inside the graphics window to activate this collector.
7. Hold down the CTRL key and from the graphics window, select one or more edges or curves from your model to append to or remove from the chain. The **Appended** collector does not allow you to select an excluded edge or curve. Similarly, the **Excluded** collector does not allow you to select an appended edge or curve.

Note:

- Pro/ENGINEER dynamically validates the chain you are creating or editing. If the chain is valid the **Dynamic validation** box is automatically selected. When dynamic validation is enabled, adding new references to the active collector or removing references from the active collector does not make the active chain invalid.
 - If the chain is invalid the **Dynamic validation** box is automatically cleared. When dynamic validation is disabled, you can add new references to the active collector or remove references from the active collector without restrictions. However, this may make the active chain invalid. If the chain becomes invalid then the **Dynamic validation** box becomes unavailable.
 - You can disable dynamic validation at any time.
8. Click **OK**. Pro/ENGINEER saves the changes, closes the **Chain** dialog box, and resumes the tool.

Tip:

- Notice that as you modify a chain in the **Chain** dialog box, Pro/ENGINEER dynamically displays the changes in the graphics window.
- To clear references in the active collector, click **Remove** or **Remove All** on the shortcut menu that appears when you right-click in the collector. Alternatively, click **Clear** on the shortcut menu that appears when you right-click in the graphics window.

- To easily locate references in the **Chain** dialog box, place your pointer over the reference in a collector. Pro/ENGINEER dynamically highlights the reference on the model.

To Modify the Start Point or Direction of a Chain

Note: This topic discusses chain modification while working inside of a tool and using the **Chain** dialog box. If you want to modify a chain while working outside of a tool, you can use the SHIFT key to clear the selection of individual items from the ends of a chain or use the CTRL key to remove an entire chain from the selection set. Refer to *About Selection* under *See Also* for more information about clearing a selection.

1. Construct the chain and open the tool in which to work with the chain, or select a feature that contains an existing chain that you want to modify and click **Edit Definition**.
2. Locate the Detail collector that contains the chain that you want to modify. Detail collectors are located on the slide-up panel. Note that you cannot use the Summary collector (located on the dialog bar) to modify a chain.
3. Click **Details**. Pro/ENGINEER pauses the tool, opens the **Chain** dialog box, and displays the attributes for the chain.
4. If the slide-up panel Detail collector contains more than one chain, select the chain to modify from the chain list in the **Chain** dialog box. The chain attributes are displayed.
5. Click the **Options** tab.
6. To change the start point and direction of an open-ended chain (One-by-One, Tangent, and Partial Loop), click **Flip**. Pro/ENGINEER reverses the chain start point and direction. Skip to step 9.
7. To change the chain start point location of a closed-loop chain (One-by-One or Complete Loop), click the **Start Point** collector. After the collector is activated, select any vertex from the chain on your model as the new start point location. Notice that you can also use the **Start Point** shortcut menu command from inside the graphics window to activate this collector.
8. To change the chain direction of a closed-loop chain, click **Flip**. Pro/ENGINEER reverses the chain direction.
9. Click **OK**. Pro/ENGINEER saves the changes, closes the **Chain** dialog box, and resumes the tool.

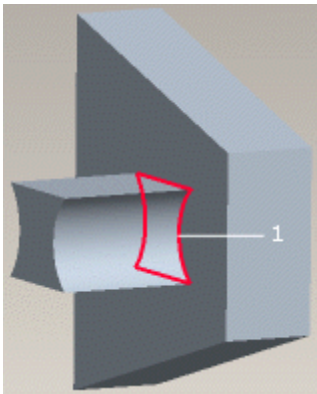
Tip:

- Notice that as you modify a chain in the **Chain** dialog box, Pro/ENGINEER dynamically displays the changes in the graphics window.
- To clear references in the active collector, click **Remove** or **Remove All** on the shortcut menu that appears when you right-click in the collector. Alternatively, click **Clear** on the shortcut menu that appears when you right-click in the graphics window.

- To easily locate references in the **Chain** dialog box, place your pointer over the reference in a collector. Pro/ENGINEER dynamically highlights the reference on the model.
- To remove the Start point handle, right-click the handle and click **Remove** on the shortcut menu. The End point handle appears. Similarly, to remove the End point handle, right-click the handle and click **Remove** on the shortcut menu. The Start point handle appears.

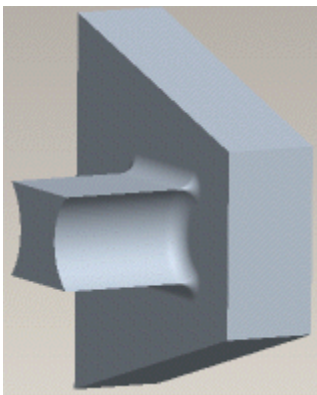
Example: Intent Chains

1. The following figure shows an intent chain. All edges of an intent chain can be selected in a single click.

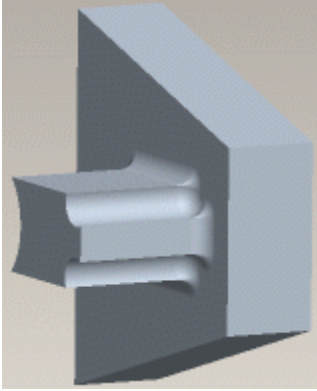


1 Intent chain

2. In the following figure the edges of the protrusion are rounded using the intent chain.

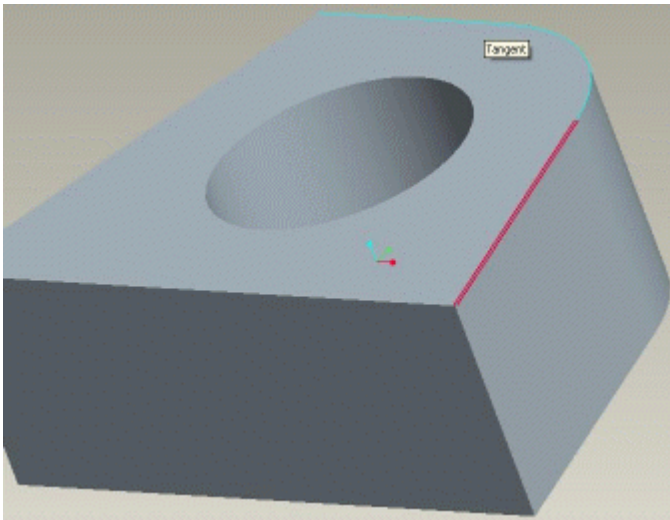


3. In the following figure the geometry has changed, but the intent, that is, a round about the bottom, has not changed. The intent chain and the round are updated automatically.



Example: Tangent Chain

In the following example, an edge tangent to the anchor is selected resulting in a tangent chain.



Surface Sets

About Surface Sets


Surface sets consist of multiple surfaces that you select and place into a group. This group or surface set enables you to efficiently perform modeling actions on the selected surfaces at one time. You can select surfaces and use surface sets at any time (within a tool, or before entering a tool) during your modeling session. Remember that when you want to construct a surface set, you must first select a reference and then press (and hold) the SHIFT key to activate the surface set construction mode.



Note: When working with an assembly inside a tool, you can select surface sets from the same model only.

To modify surface sets, you use the **Surface Sets** dialog box. This dialog box contains a list of surface sets that you constructed using the active part, the number of surfaces in each surface set, attributes for each surface set, and excluded

surfaces. Pro/ENGINEER provides ToolTips, messages, and labels that act as a visual aid guiding you through the surface set construction process.

When using a dashboard tool or a tool that uses the Menu Manager, you can dynamically preview the surface set geometry after every action. To disable dynamic preview, clear the **Dynamic preview** checkbox in the **Surface Sets** dialog box.

Note: When using a dashboard tool, the state of the checkbox in  controls the initial setting of the **Dynamic preview** checkbox in the **Surface Sets** dialog box.

For example, if you change  to  before opening the **Surface Sets** dialog box, then the **Dynamic preview** checkbox appears as not selected when you subsequently open the **Surface Sets** dialog box.

Surface Set Types

You can construct the following types of surface sets:

Non rule-based

- **Individual Surface Set**—Contains individual solid surfaces or quilts that you select. You can also use a Individual surface set if you want to model additional surfaces along with other surface sets that you constructed. Note that if you want to exclude surfaces from a Individual surface set, you must remove the surface from the set. You cannot add them to the Excluded surface set.
- **Intent Surface Set**—Contains intent surfaces. You can add intent surfaces to the Excluded surface set or use the intent surfaces as bounding surfaces in Seed and Boundary Surface set.
- **Excluded Surface Set**—Contains all surfaces that you excluded from one or more surface sets. You construct the excluded surface set when you do not want to perform a modeling action on one or more individual surfaces that belong to a surface set. Note that you cannot add unwanted surfaces from a Individual surface set to an Excluded surface set.

Rule-based

- **All Solid Surface Set**—Contains all solid surfaces of the active part.
- **Quilt Surface Set**—Contains quilt surfaces that you selected from the active part.
- **Loop Surface Set**—Contains two-sided edges that form a loop around a surface.
- **Seed and Boundary Surface Set**—Contains all surfaces between and including the selected seed and boundary surfaces.

Note: Most tools require that all selected surfaces must belong to the same solid part geometry or quilt.

To Construct an Individual Surface Set

An Individual surface set is simply a selection set containing of one or more solid or quilt surfaces.

1. Select a surface on your model. Pro/ENGINEER highlights the selected surface. Notice that if you are working outside of a tool, the Selected Items area indicates the selection and the **Selected Items** dialog box contains the surface references. However, if you construct a Individual surface set from inside a tool, the active collector contains the surface set and the `Individual Surfaces` label displays on the model.
2. If you want to add additional surfaces to the selection set, press CTRL and select them. Note that most tools require that all selected surfaces for the surface set belong to the same solid part geometry or quilt.
3. Open a feature tool or continue working in a feature tool to use the surface set to create a feature.

To Construct an All Solid Surface Set

1. Select a solid surface on your model. Pro/ENGINEER highlights the selected surface.
2. Right-click and select **Solid Surfaces** on the shortcut menu. Pro/ENGINEER constructs the surface set and adds all solid surfaces to the set. Note that if you are working outside of a tool, the Selected Items area indicates the selection and the **Selected Items** dialog box contains `All Solid Surfaces`. However, if you construct a surface set from inside a tool, the active collector contains the surface set and the `All Solid Surfaces` label is displayed on the model.
3. Open a feature tool or continue working in a feature tool to use the surface set to create a feature.

Note:

- When working in a tool, if the surface sets collector or if surface sets and the **Anchor** collector are empty, then the Part filter is available. You can use the Part filter to select a part from the Model Tree or graphics area. You can also use the Part filter to select the components from the Layer Tree, irrespective of the layer status.
- When working in a tool, if the surface sets and the **Anchor** collector are empty, then the Part filter is available. To construct an All Solid surface set select the Part filter and then select a part with solid surfaces from the graphics area or the Model Tree. When you select a part, by default, the **Anchor** collector displays the first solid surface in the part and continues to remain active, the **All solid surfaces** option is selected, and the Part filter becomes unavailable.

Holding down the CTRL key and selecting the anchor surface clears the selection of the All Solid surface set.

- When working in a tool, if the surface sets collector is empty, then the Part filter is available. To construct an All Solid surface set select the Part filter and then select a part that has solid surfaces from the graphics area or the Model Tree.

To Construct a Quilt Surface Set

Note:

- The active part must contain one or more quilt to construct a Quilt surface set.
 - If you constructing a Quilt surface set while working in a tool, activate the surfaces collector and follow step 2. through step 4.
1. Select a feature from the model. Pro/ENGINEER highlights the feature.
 2. Query each feature and locate a quilt. Notice that you can also use the **Quilts** filter to easily locate quilts.
 3. Select a quilt. Pro/ENGINEER adds the selected quilt surface to the selection set. If you are working outside of a tool, the Selected Items area indicates the selection and the **Selected Items** dialog box contains the quilt reference. If you are working from inside a tool, the active collector contains the Quilt surface set and the `Quilt Surfaces` label appears on the model.
 4. If you want to add additional quilts to the selection set, press CTRL and repeat step 1. through step 3. Note that most tools require that the selected surfaces belong to the same solid part geometry or quilt.
 5. Open a feature tool. Pro/ENGINEER constructs the Quilt surface set, and the surfaces collector contains the Quilt surface set. The `Quilt Surfaces` label appears on the model.

Tip: You can place your pointer over an item and right-click, or use **Next** and **Previous** from the shortcut menu to query each feature. You can also use the **Pick From List** to select features.

To Construct a Loop Surface Set

1. Select a surface on your model to establish the anchor. Pro/ENGINEER highlights the selected surface. Note that most tools require that the surface belongs to the same solid part geometry or quilt.
2. Press SHIFT and move your pointer over the boundary of the selected surface. Valid surfaces highlight and a ToolTip displays each surface name.
3. Select a two-sided edge that belongs to the anchor surface to complete the Loop Surface boundary. Pro/ENGINEER constructs and highlights the Loop surface set. Notice that if you are working outside of a tool, the Selected Items area indicates the selection and the **Selected Items** dialog box contains the `Loop Surfaces`. However, if you construct a surface set from inside a tool, the active collector contains the surface set and the `Loop Surfaces` label displays on the model.
4. If you want to construct an additional Loop surface set during the same workflow, release SHIFT. Then, press CTRL and click a surface on your model to establish an anchor for the new surface set. Release CTRL and repeat step 2 through step 4.
5. Open a feature tool or continue working in a feature tool to use the surface set to create a feature.

Tip:

- You can use **Next**, **Previous**, or **Pick from List** from the shortcut menu to select edges.
- To open the **Selected Items** dialog box, simply double-click the Status area next to the Filter box.

To Modify a Loop Surface Set

You must be inside a tool to modify Loop Surface sets.

1. In the graphics window, place your pointer over a `Loop Surfaces` label and right-click.
2. Click **Activate Set** from the shortcut menu .
3. Press SHIFT and move your pointer over the boundary of the selected surface. Valid edges and surfaces highlight. A ToolTip displays the names of valid edges.
4. Select the required edge to change the loop surface and complete the Loop Surface set. Pro/ENGINEER highlights the Loop Surface set.

Note: A Loop Surface set closes if a new set is added, the tool is closed, or if you activate another collector inside of a tool.

Tip:

- To remove a Loop Surface set, right-click on the label and click **Remove Set**.
- You cannot modify the anchor surface of a Loop surface set.
- You can use **Next**, **Previous**, or **Pick from List** from the shortcut menu to select edges.
- Inside a tool the selected items are not available for surface collection.

To Construct an Intent Surface Set

An Intent surface set is simply a selection set containing one or more intent surfaces.

Note: If you constructing an Intent surface set while working in a tool, activate the surfaces collector and follow step 2. through step 4.

1. Select a feature from the model. Pro/ENGINEER highlights the feature.
2. Query each feature and locate an intent surface.
3. Select an intent surface. Pro/ENGINEER adds the selected intent surface to the selection set. If you are working outside of a tool, the Selected Items area indicates the selection and the **Selected Items** dialog box contains the intent surface reference. If you are working from inside of a tool, the active collector contains the Intent surface set and the `Intent Surfaces` label appears on the model.

4. If you want to add additional intent surfaces to the selection set, press CTRL and repeat step 1. through step 3. Note that most tools require that the selected surfaces belong to the same solid part geometry or quilt.
5. Open a feature tool. Pro/ENGINEER constructs the Intent surface set, and the surfaces collector contains the Intent surface set. The *Intent Surfaces* label appears on the model.

Tip:

- If you are working in a tool, you can also use the **Intent Surface** filter to easily locate intent surfaces.
- You can place your pointer over an item and right-click, or use **Next** and **Previous** from the shortcut menu to query each feature. You can also use the **Pick From List** to select features.

To Construct a Seed and Boundary Surface Set

1. Select a surface on your model to establish the surface anchor (seed) surface. Pro/ENGINEER highlights the selected surface.
2. Hold down the SHIFT key and move your pointer over the model. Valid surfaces highlight and a ToolTip displays the surface name.
3. Select a surface that belongs to the same solid part geometry or quilt as the anchor surface. This establishes the boundary surface. Pro/ENGINEER constructs and highlights the Seed and Boundary surface set. Notice that if you are working outside of a tool, the Selected Items area indicates the selection and the **Selected Items** dialog box contains *Seed and Boundary Surfaces*. However, if you construct a surface set from inside a tool, the active collector contains the surface set and the *Seed and Boundary Surfaces* label displays on the model.

Note: You can also use a Loop surface set as a boundary for constructing a Seed and Boundary surface set. To construct a Loop surface set and use it as a boundary for constructing the Seed and Boundary surface set, hold down the SHIFT key and select a two-sided edge that belongs to the selected boundary surface to complete the Loop surface boundary.

4. Release SHIFT. The Seed and Boundary surface set highlights. This set includes the seed surface and all the surfaces between the seed surface and the boundary surfaces but not the boundary surfaces. Notice that boundary surface is not highlighted.
5. If you want to include additional boundary surfaces to the set during the same workflow, repeat step 2 through step 4. Note that most tools require that the surfaces belong to the same solid part geometry or quilt.
6. If you want to construct an additional Loop surface set during the same workflow, release SHIFT. Then, press CTRL and click a surface on your model to establish an anchor for the new surface set. Release CTRL and repeat step 2. through step 4.

7. Open a feature tool or continue working in a feature tool to use the surface set to create a feature.

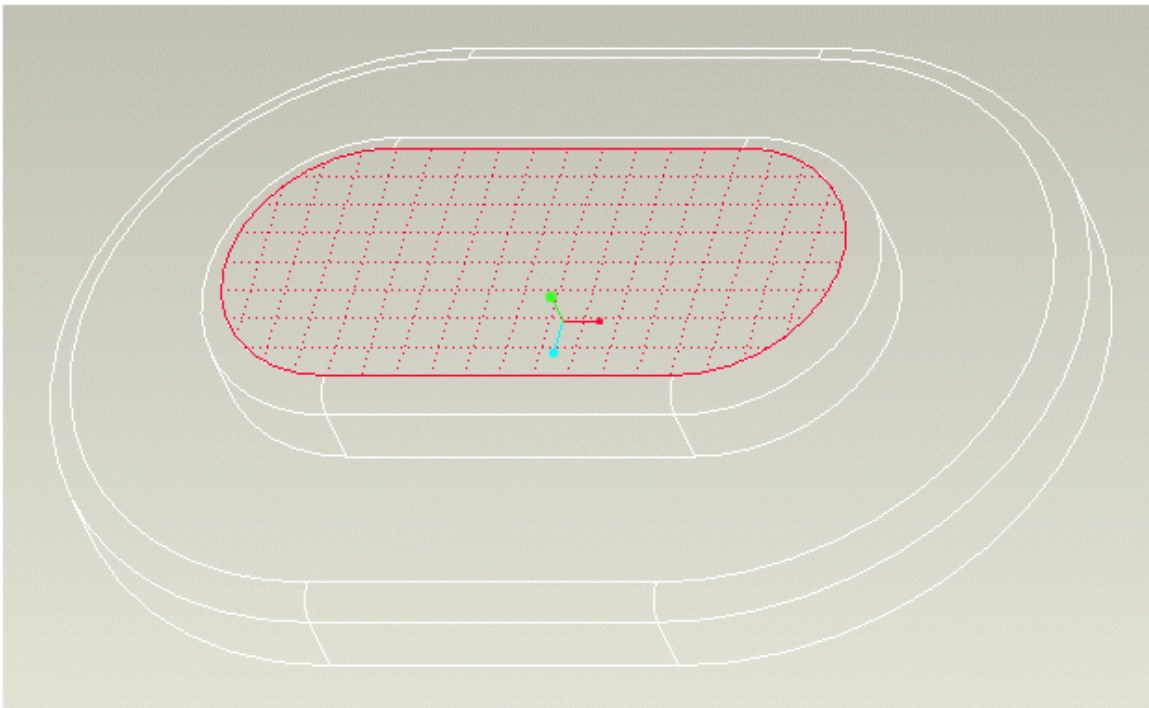
Tip:

- You can use **Next**, **Previous**, or **Pick from List** from the shortcut menu to select edges.
- To open the **Selected Items** dialog box, simply double-click the Status area next to the Filter box.
- In Seed and Boundary surface sets you can use Intent surfaces as bounding surfaces.

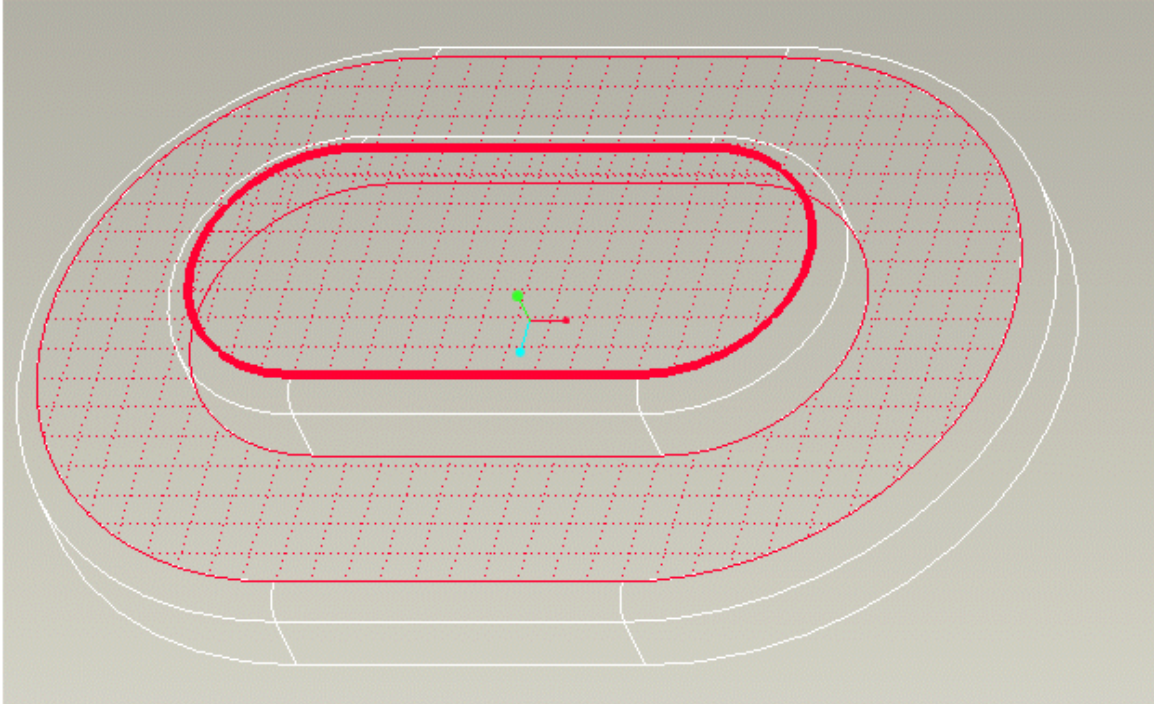
Example: Constructing a Seed and Boundary Surface Set

This example shows how to construct a Seed and Boundary surface set using a Loop surface set as a boundary.

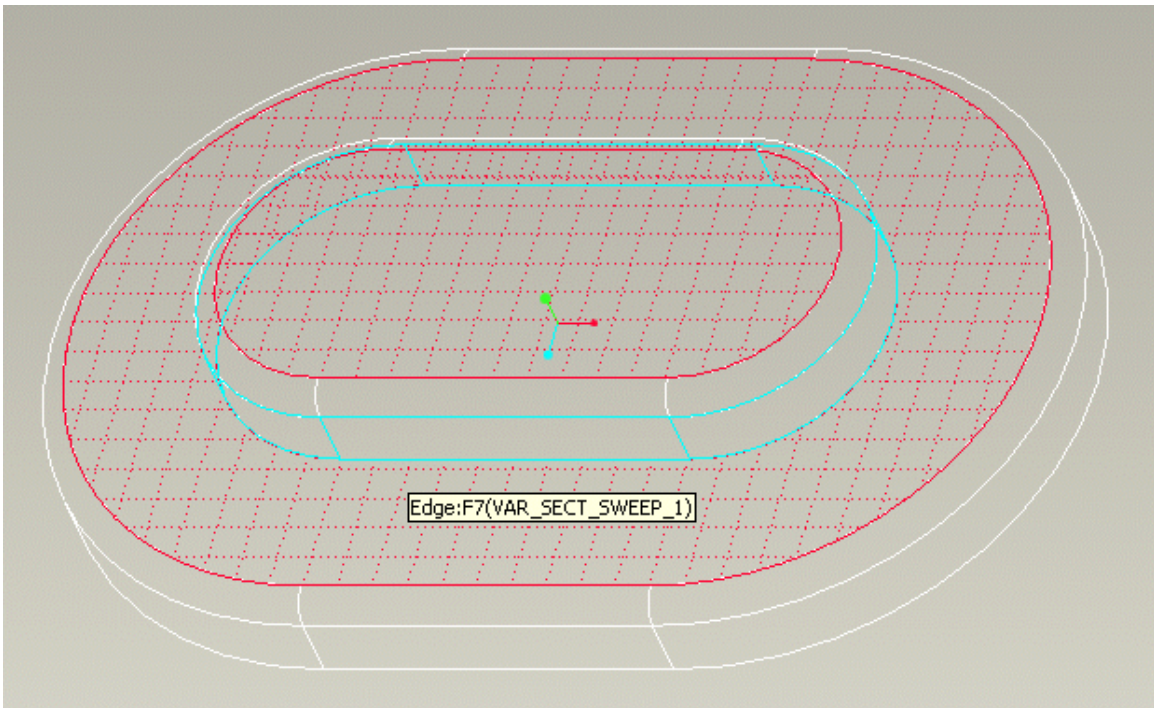
Select a seed surface on the model to establish the surface anchor (seed) surface. Pro/ENGINEER highlights the selected surface as shown below.



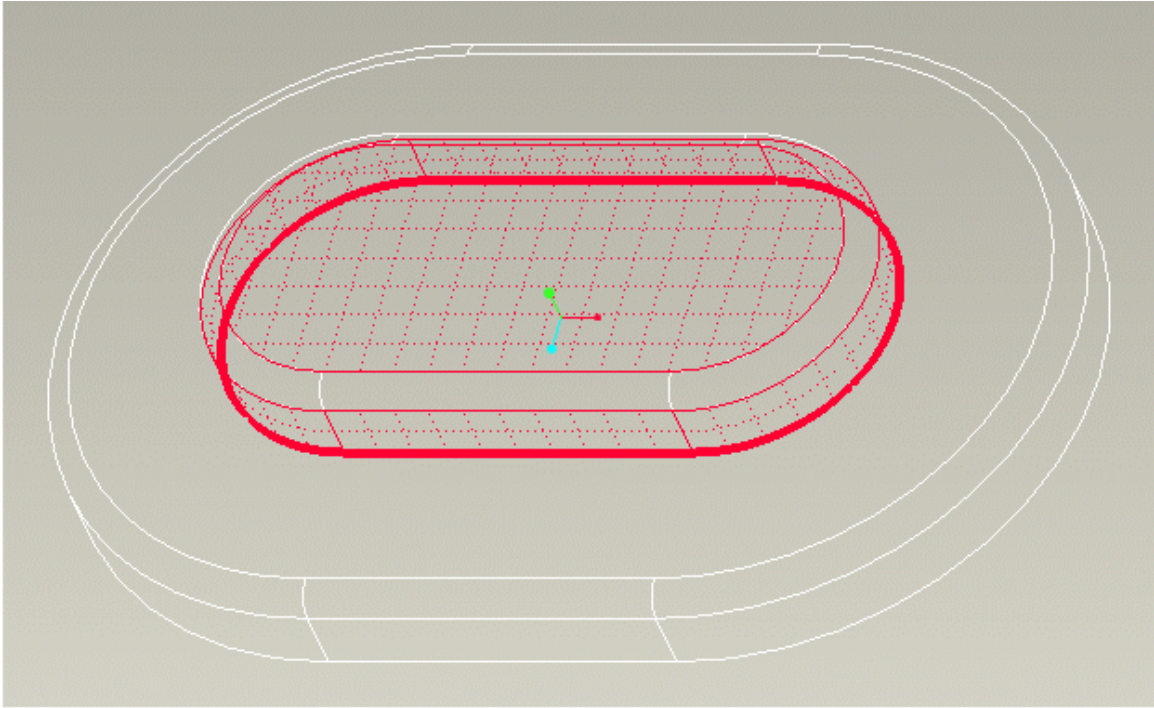
Hold down the SHIFT key and move the pointer over the model. Valid surfaces are highlighted and a ToolTip displays the surface name. Select a boundary surface as shown below.



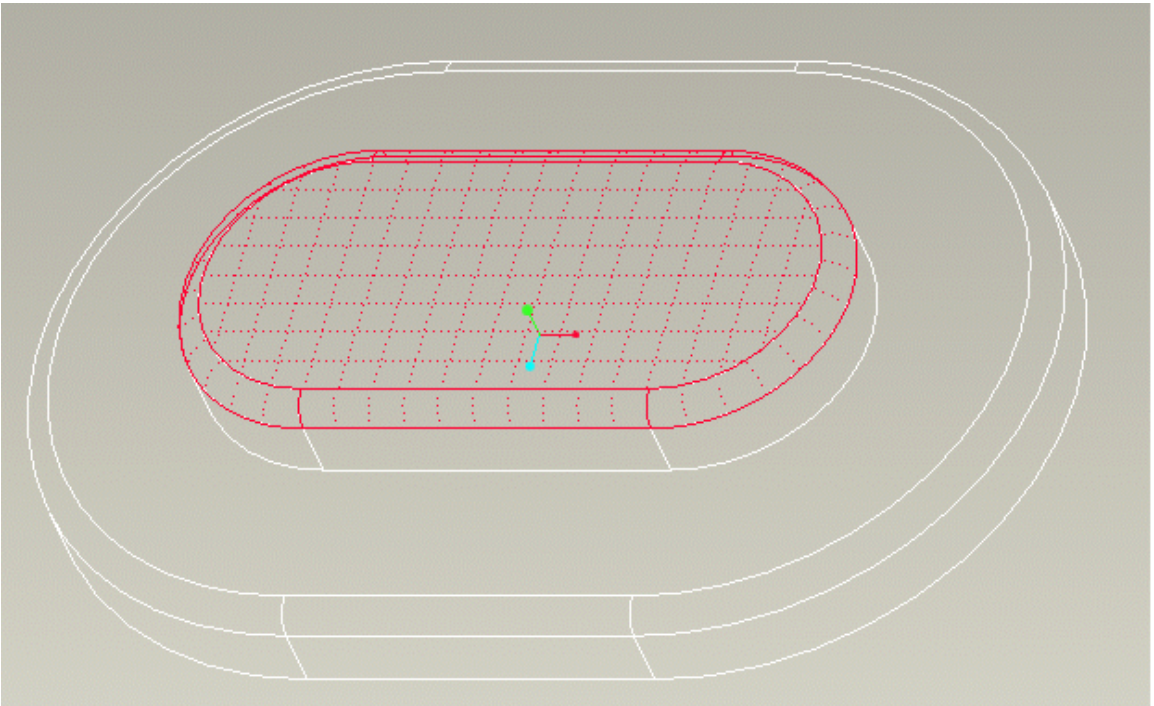
Hold down the SHIFT key and move the pointer over the desired Loop edge of the selected boundary. The Loop surface is highlighted as shown below.



Select the Loop edge that belongs to the selected boundary surface to complete the Loop surface boundary. The Loop surface set is highlighted as shown below.



Release the SHIFT key. The Seed and Boundary surface set is created as shown below.



To Modify a Seed and Boundary Surface Set

You must be inside a tool to modify Seed and Boundary Surface sets.

1. In the graphics window, place your pointer over the *Seed and Boundary Surfaces* label and right-click.
2. Click **Activate Set** from the shortcut menu.
3. Press SHIFT and select the required boundary surface or surfaces to add or remove them from the Seed and Boundary Surface set. Pro/ENGINEER highlights the seed and boundary surfaces along with the valid surfaces. A ToolTip displays the names of valid surfaces.

Note:

- You cannot modify the anchor surface of a Seed and Boundary surface set.
- A Seed and Boundary surface set closes if a new set is added, the tool is closed, or if you activate another collector inside of a tool.

Tip:

- To remove a boundary surface from the set, right-click on the label and click **Remove Set**.
- You can use **Next**, **Previous**, or **Pick from List** from the shortcut menu to select edges.
- Inside a tool the selected items are not available for surface collection.

To Exclude Surfaces from Surface Sets

You can use this procedure to exclude surfaces either before opening a tool or while you are working in a tool.

1. Construct one or more surface sets using the active part. Pro/ENGINEER highlights the surfaces.
2. Press CTRL and select one or more highlighted surfaces that you want to exclude from perform a modeling action. Pro/ENGINEER constructs the Excluded surface set and adds the excluded surfaces to it. Notice that the Selected Items area indicates the selection and the **Selected Items** dialog box contains *Excluded Surfaces*.
3. Open a feature tool if one is not already open, and use the surface sets to create a feature. Notice that in a tool, the active collector contains the Excluded surface set and the *Excluded Surfaces* label appears on the model.

Tip: You can use **Next**, **Previous**, or **Pick from List** from the shortcut menu to select edges.

To Modify Rule-based Surface Sets

Note: This topic discusses surface set modification while working inside of a tool and using the **Surface Sets** dialog box. If you want to modify a surface set while working outside of a tool, you can use the CTRL key to include individual surfaces in the set. You can also clear the entire surface set from the selection set. Refer to *About Selection* under *See Also* for more information about clearing a selection.

1. If you are working in a tool that uses the Menu Manager and not the dashboard, construct the surface set and open a tool. Place your pointer over a surface set label in the graphics window, right-click, and select **Surface Sets** on the shortcut menu. The **Surface Sets** dialog box opens. Skip to step 5.
2. If you are working in a dashboard tool, construct a surface set and open a tool in which to work with the surface set, or select a feature that contains an existing surface set that you want to modify, right-click, and select **Edit Definition** on the shortcut menu.
3. Locate the Detail collector that contains the surface set that you want to modify. Detail collectors are located on the slide-up panel. Note that you cannot use the Summary collector (located on the dialog bar) to modify a surface set.
4. Click **Details**. Pro/ENGINEER pauses the tool and opens the **Surface Sets** dialog box. This dialog box contains a list of surface sets that were constructed using the active part, and their attributes. Notice that the surface set list contains set type and the number of surfaces contained in each set.
5. Do one of the following from the Surface Sets list:
 - Click **Neighbor Surfaces** to modify a Loop surface set. The **Anchor** and **Loop Edge** collectors appear.
 - Click **Bounded Set** to modify a Seed and Boundary surface set. The **Anchor** and **Bounding Surfaces** collectors appear if individual surfaces determine the boundary. The **Anchor**, **Boundary loop anchor**, and **Boundary loop edge** collectors appear if Loop surfaces determine the boundary.
 - Click **All Solid Surfaces** to modify a Solid surface set. The **Anchor** collector appears.
 - Click **Quilt Surfaces** to modify a Quilt surface set. The **Anchor** collector appears.
6. After you select a surface set from the list, the following options appear based on the surface set that you selected:
 - **Anchor collector**—Contains the anchor reference for the selected surface set. The anchor reference consists of a solid surface or quilt that belongs to the same solid part geometry or quilt.
 To modify the anchor reference, click the reference to activate the collector and select a different anchor from the model. Notice that you can also use the **Anchor** shortcut menu command from inside the graphics window to activate this collector.
Note: You can only replace the anchor reference, you cannot remove it.
 - **Rule**—Displays the following rule-based surface set types enabling you to modify the selected surface set:

Loop surfaces—Constructs a Loop surface set by using all edges that form a loop around an anchor surface. Note that the anchor reference cannot be a quilt or an intent surface if you want to use this option.

Seed and boundary surfaces—Constructs a Seed and Boundary surface set by using all surfaces between the seed surface and the bounding surfaces, including the seed surface and excluding the bounding surfaces. Displays the **Boundary** options and the **Include bounding surfaces** check box. Note that to use **Seed and boundary surfaces** option, the anchor reference cannot be a quilt or an intent surface.

All solid surfaces—Constructs an All Solid surface set by using all solid surfaces in the active part. The **Reference** collector appears and contains the surface reference.

- **Boundary**—Displays the type of boundary used to construct the Seed and Boundary Surface set.

Individual Surfaces—Constructs a Seed and Boundary Surface set with individual surfaces as the bounding surfaces. Activates the **Bounding Surfaces** collector.

Loop Surfaces—Constructs a Seed and boundary Surface set with a Loop surface set as the bounding surface. Activates the **Boundary loop anchor** collector.

- **Loop Edge collector**—Contains an edge that forms a loop around a surface. This edge reference determines the neighboring surfaces for a Loop surface set. A loop edge reference consists of a two-sided edge that belongs to the anchor reference surface.

To modify a loop edge reference, click a loop edge reference to activate the collector and select a different edge from the model. This new reference replaces the existing loop edge reference. Notice that you can also use the **Loop Edge** shortcut menu command from inside the graphics window to activate this collector.

Note: You must select the **Neighboring surfaces only** rule-based surface set type to use this collector.

- **Bounding Surfaces collector**—Contains one or more bounding surface references that determine the boundary for the Seed and Boundary surface set. A bounding surface reference consists of a solid surface or quilt that belongs to the same solid part geometry or quilt. You can use intent surfaces as bounding surfaces.

To modify a bounding surface reference, click the reference to activate the collector and either select a different surface from the model, hold down the CTRL key and select additional references, or remove a reference. Notice that you can also use the **Bounding Surfaces** shortcut menu command within the graphics window to activate this collector.

Note:

Surfaces removed from the Individual surface set are not considered

excluded surfaces.

You must select the **Seed and boundary surfaces** rule-based surface set type to use this collector.

- **Boundary loop anchor collector**—Contains a surface reference that is an anchor for the Loop surface set. The Loop surface set determines the boundary for the Seed and Boundary surface set. The anchor surface reference consists of a solid surface or quilt that belongs to the same solid part geometry or quilt and acts as the anchor surface for the Loop surface set.

To modify an anchor surface reference, click inside the collector to activate it and either select a different surface reference from the model or remove the surface reference. You can remove the surface reference by clicking **Clear** on the shortcut menu in the graphics window or by clicking **Remove** on the shortcut menu that appears when you right-click in the collector. You can also use the **Boundary loop anchor** command on the shortcut menu within the graphics window to activate this collector.

- **Boundary loop edge collector**—Contains an edge reference for the Loop surface set. The Loop surface set acts as a boundary for the Seed and Boundary surface set. This edge reference determines the neighboring surfaces for a Loop surface set. A loop edge reference consists of a two-sided edge that belongs to the anchor reference surface.

To modify the edge reference, click inside the collector to activate it and either select a different edge reference from the model or remove the edge reference. You can remove the edge reference by clicking **Clear** on the shortcut menu in the graphics window or by clicking **Remove** on the shortcut menu that appears when you right-click in the collector. You can also use the **Boundary loop edge** command on the shortcut menu within the graphics window to activate this collector.

- **Reference collector**—Contains the surface references for the Solid surface set. Note that this reference is only displayed and can only be modified by changing the anchor reference.
- **Include bounding surfaces check box**—Includes the bounding surfaces in a Seed and Boundary surface set.

7. Click **OK**. Pro/ENGINEER saves the changes, closes the **Surface Sets** dialog box, and resumes the tool.

Tip:

- You can always construct a new rule-based surface set by clicking **Add** and selecting the surface references from the model.
- To remove a surface set from the Surface Set list, select the surface set and click **Remove**. Note that **Individual Surfaces** and **Excluded Surfaces** cannot be removed.

- You can also remove a surface set from the graphics window by placing your pointer over the surface set label, right-click, and use **Remove Set** from the shortcut menu.
- To clear references in the active collector, click **Remove** or **Remove All** on the shortcut menu that appears when you right-click in the collector. Alternatively, click **Clear** on the shortcut menu that appears when you right-click in the graphics window.
- Remember that you can hold down the CTRL key and select a reference to either add an additional reference or to clear a selected reference.
- To easily locate a references in the **Surface Sets** dialog box, place your pointer over the reference in a collector. Pro/ENGINEER dynamically highlights the reference on the model.
- Notice that as you modify a surface set in the **Surface Sets** dialog box, Pro/ENGINEER dynamically displays the changes in the graphics window.
- When the **Surface Sets** dialog box is open, you can perform Undo or Redo operations if the tool or feature supports Undo or Redo operations.

To Modify Non Rule-based Surface Sets

Note: This topic discusses surface set modification while working inside of a tool and using the **Surface Sets** dialog box. If you want to modify a surface set while working outside of a tool, you can use the CTRL key to include individual surfaces in the set. You can also clear the entire surface set from the selection set. Refer to *About Selection* under *See Also* for more information about clearing a selection.

1. If you are working in a tool that uses the Menu Manager and not the dashboard, construct the surface set and open a tool. Place your pointer over a surface set label in the graphics window, right-click, and select **Surface Sets** from the shortcut menu. The **Surface Sets** dialog box opens. Skip to step 5.
2. If you are working in a dashboard tool, construct a surface set and open a tool in which to work with the surface set, or select a feature that contains an existing surface set that you want to modify, right-click, and select **Edit Definition** from the shortcut menu.
3. Locate the Detail collector that contains the surface set that you want to modify. Detail collectors are located on the slide-up panel. Note that you cannot use the Summary collector (located on the dialog bar) to modify a surface set.
4. Click **Details**. Pro/ENGINEER pauses the tool and opens the **Surface Set** dialog box. This dialog box contains a list of surface sets that were constructed using the active part, and their attributes. Notice that the surface set list contains set type and the number of surfaces contained in each set.
5. Do one of the following from the Surface Sets list:
 - Click **Individual Surfaces** to modify the Individual surface set. The **Included Surfaces** collector appears.

- Click **Intent Surfaces** to modify an Intent surface set. The **Anchor** collector appears.
 - Click **Excluded Surfaces** to modify the Excluded surface set. The **Excluded Surfaces** collector appears.
6. After you select a surface set from the list, the following options appear based on the surface set that you selected:
- **Included Surfaces** collector—Contains one or more surface references that you selected. Individual surfaces references consists of a solid surface or quilt that belongs to the same solid part geometry or quilt.
To modify a reference, click the reference to activate the collector and then either select a different surface from the model, press CTRL and select additional references, or remove a reference. Notice that you can also use the **Included Surfaces** shortcut menu command from inside the graphics window to activate the **Included Surfaces** collector.
Note: Surfaces removed from the Individual surface set are not added to the Excluded surface set. These surfaces are removed only from their respective surface set.
 - **Anchor** collector—Contains the anchor reference for the selected surface set. The anchor reference consists of a solid surface or quilt that belongs to the same solid part geometry or quilt.
To modify the anchor reference, click the reference to activate the collector and select a different anchor from the model. Notice that you can also use the **Anchor** shortcut menu command from inside the graphics window to activate this collector.
Note: You can only replace the anchor reference, you cannot remove it.
 - **Excluded Surfaces** collector—Contains one or more excluded surfaces that you selected.
To modify a reference, click the reference to activate the collector and then either select a different reference from the model, press CTRL and select additional references, or remove a reference. Notice that you can also use the **Excluded Surfaces** shortcut menu command from inside the graphics window to activate the **Excluded Surfaces** collector.
Note:
Surfaces removed from the **Excluded Surfaces** collector return to their respective surface set.
Surfaces removed from the Individual surface set are not add to the Excluded surface set. These surfaces are removed only from their respective surface set.
7. Click **OK**. Pro/ENGINEER saves the changes, closes the **Surface Sets** dialog box, and resumes the tool

Tip:

- You can always construct a new rule-based surface set by clicking **Add** and selecting the surface references from the model.

- To remove a surface set from the Surface Set list, select the surface set and click **Remove**. Note that **Individual Surfaces** and **Excluded Surfaces** cannot be removed.
- You can also remove a surface set from the graphics window by placing your pointer over the surface set label, right-click, and use **Remove Set** from the shortcut menu.
- To clear references in the active collector click **Remove** or **Remove All** on the shortcut menu that appears when you right-click in the collector. Alternatively, click **Clear** on the shortcut menu that appears when you right-click in the graphics window.
- Remember that you can press CTRL and select a reference to either add an additional reference or to clear a selected reference.
- To easily locate a reference in the **Surface Sets** dialog box, place your cursor over the reference in a collector. Pro/ENGINEER dynamically highlights the reference on the model.
- Notice that as you modify a surface set in the **Surface Sets** dialog box, Pro/ENGINEER dynamically displays the changes in the graphics window.

Drag and Drop

About Drag and Drop

Pro/ENGINEER lets you move files from browsers, other Pro/ENGINEER sessions and from Windchill Solutions to the Pro/ENGINEER graphics window and the Model Tree. When a file is dragged to an acceptable target, the pointer changes indicating the object can be placed there.

Links can also be dragged from browsers and Pro/ENGINEER navigators, and dropped in the embedded Web browser.

Graphics Window

Pro/ENGINEER files can be dropped in the graphics window from the embedded Web browser, a standalone browser or a file manager. Note that the Internet Explorer browser must be version 5.5 and later. The Netscape Navigator browser must be version 6.0 and later.

When you activate the Folder Navigator, you can drag a Pro/ENGINEER component from the embedded Web browser and drop it in the graphics window. The component opens in a separate window unless you drop it on an active assembly.

By default, when you drop a component on an assembly, Pro/ENGINEER attempts to automatically place it. If the component cannot be assembled automatically, the **Component Placement** dialog box opens to allow you to select references and constrain the part in the assembly. If you want to choose whether to assemble the component or open it in a new window, set the configuration option `autoplace_single_component` to `no`. When set to `no`, the **Confirm Retrieval** dialog box opens displaying the choice to open the component in a separate window or to

add it to the assembly. Refer to the help in Assembly for more information on placing components in assemblies.

You can drag a .zip file to the graphics window. The **File Open** dialog box opens and displays the files contained in the .zip . Click a Pro/ENGINEER file in the **File Open** dialog box to open it.

Model Tree


You can drag and drop components onto the Model Tree. If you have an assembly open, the dropped component is unplaced. To place the component, click **Edit Definition** from the Model Tree shortcut menu

Search Navigator

If you are connected to a Windchill server, files are listed in the embedded Web browser. You can drag a file from the Number window in the Search navigator to the graphics window. The details page for that object open in the embedded Web browser. You can then drag a part or assembly from the browser to the graphics window.

Tool Pausing

About Tool Pausing



When creating features, you may need to work in a tool that does not use the dashboard. Use  on the dashboard to suspend your current workflow and open another tool. When the current tool is paused, it becomes inactive, and you are temporarily returned to the default system state, in which you can select objects that satisfy other tool requirements. You can pause a tool explicitly or automatically.

To resume a paused tool, click .

To Pause a Tool


You can pause a tool either explicitly or automatically.

To Pause Explicitly

1. Click . The dashboard is now unavailable.
2. Open and use the tool for which you paused the original tool.
3. Click  to return to the tool.

To Pause Automatically

1. Select an asynchronous tool from the toolbar. The current tool is now unavailable, and the Datum creation tool interface automatically opens.

2. Click  when you complete or cancel the Datum tool to resume the paused tool.

Working with Handles

About Handles

Handles are graphical objects that you use to manipulate data in the graphics window. You can snap handles to existing geometric references or to user-defined grid increments. Changes to the geometry are displayed dynamically in the graphics window.

You can move a drag handle along a 2D or 3D trajectory.

Duplicating a Handle

In the Round, Draft, Extend, Offset Curve, and Datum Point feature-creation and Datum Point Offset Csys tools, you can widen your ability to manipulate geometry by duplicating a handle. By duplicating the handle, you change the feature type from constant to variable or another radius or point is added.

You can change the handle behavior by moving the dragger and pressing SHIFT or CTRL.

- SHIFT—Activates the Snap mechanism, if available.
- CTRL—Activates the Duplicate mechanism, if available.

To Duplicate a Handle

You can duplicate a handle when creating a Round, Draft, Extend, Offset curve, or Datum Point feature.

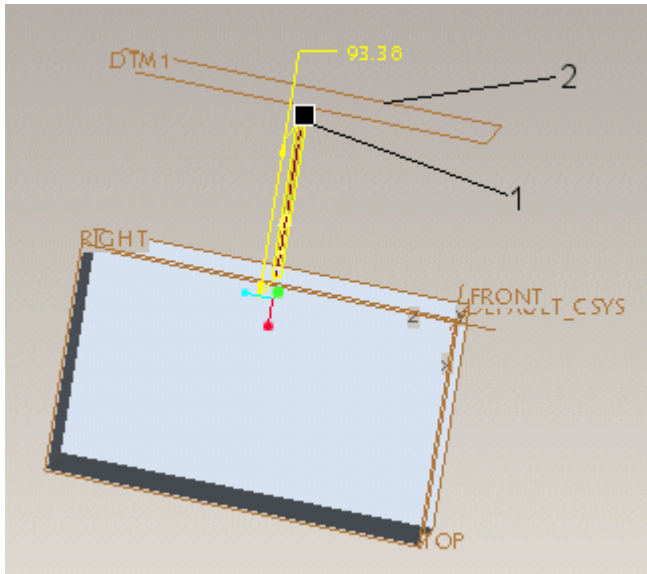
1. When creating the feature, right-click a handle to access the shortcut menu.
2. Click **Add Radius** (Round), **Add Angle** (Draft), **Add** (Extend and Offset curve), or **Duplicate** (Datum Point). The handle is duplicated. You can then drag the handle to place it.

Snapping a Handle

You can snap a handle to reference geometry, such as a datum plane, an edge, a point or vertex, or a surface.

To Snap a Handle

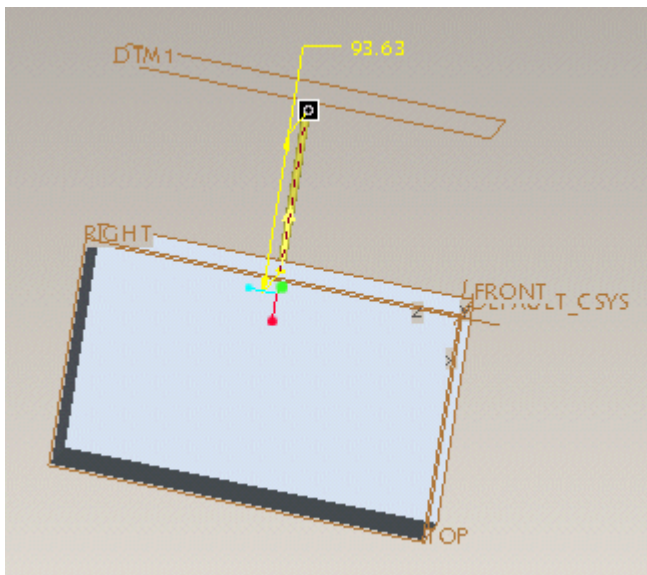
1. As you drag the drag handle toward the reference geometry to which you want to snap the handle, the drag handle appears as a black square, as shown in the next figure:



1 Drag handle

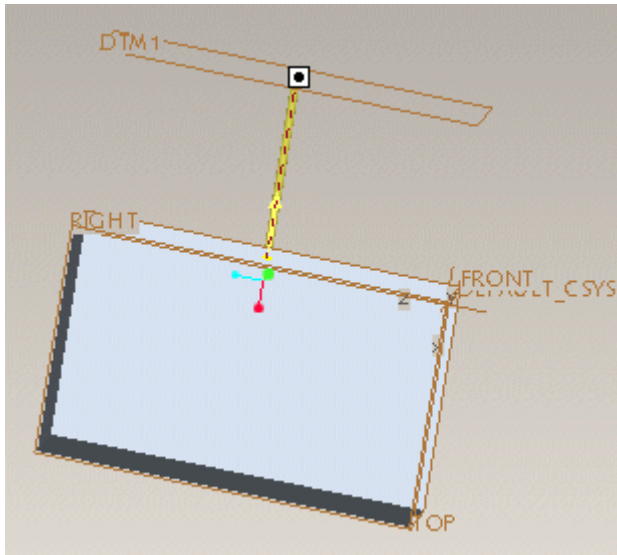
2 Reference datum plane

2. While dragging the drag handle, press and hold SHIFT. The drag handle appears as a black square containing a white circle, as shown in the next figure:



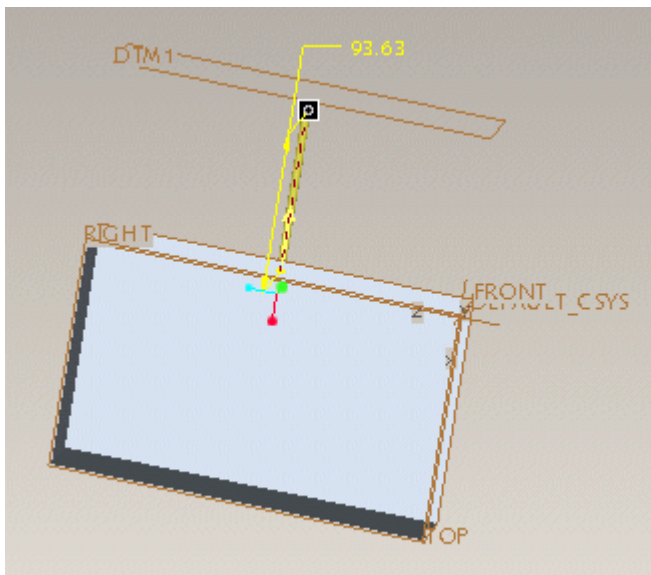
3. Move the pointer over the reference geometry. The reference geometry is highlighted.

4. Release the mouse button and SHIFT. The reference geometry is selected. The handle is snapped and appears as a white square containing a black circle, as shown in the next figure:



To Unsnap a Handle

1. Press SHIFT and move the pointer over the snapped handle. The reference geometry is highlighted and the drag handle appears as a black square containing a white circle, as shown in the following figure:



2. Drag the drag handle off of the reference geometry.
3. Release the mouse button and SHIFT. The handle is now unsnapped from reference geometry.

Using Verify Mode

About Verify Mode

Using Verify mode, you can determine whether a feature that you have defined can be created in Pro/ENGINEER. When you enter Verify mode, Pro/ENGINEER regenerates the feature in the graphics window according to the dimensions, references, and settings that you have defined. While you are in Verify mode, you cannot edit the feature or cancel feature creation.





If Pro/ENGINEER cannot regenerate the defined feature according to the current dimensions, references, and settings, you can exit Verify mode and change the feature definition or enter Fix Model mode to adjust other aspects of the model.

If Pro/ENGINEER can regenerate the defined feature, the resulting model, including the feature being created, is displayed. Exit Verify mode and either accept the feature or continue defining it.

While in Verify mode, you can activate the following commands from the **Analysis** menu:

- **Measure**
- **Model Analysis**

To Use the Verify Mode

1. Click  to enter Verify mode. The regenerated feature appears in the graphics window.
2. To exit Verify mode, click one of the following items:
 - Click  to toggle off Verify mode.
 - Click  or **Exit Verify** from the shortcut menu to resume the previous tool.
 - Middle-click or click  to accept the feature

Pro/ENGINEER Menus

The File Menu

Working with Pro/ENGINEER Files

About Pro/ENGINEER Files

The following requirements apply when creating and performing operations on Pro/ENGINEER files:

- File names have a 31 character limit. You cannot create or retrieve an object with more than a 31 character file name.
- You cannot use brackets, such as [], { }, or (), spaces and punctuation marks (.?!;) in file names.
- File names can contain hyphens and underscores, however, the first character in a file name cannot be a hyphen.
- Use only alphanumeric characters (see hyphen and underscore exception in previous bullet) in file names. Object files containing non-alphanumeric characters (such as @, #, and %) in their names do not display in, and are not retrievable from dialog boxes. You cannot save new object files with nonalphanumeric characters in their names.
- Use only lowercase characters for file names. An object or file on disk always saves with the name in lowercase.
- Directory paths can use special characters, such as tilde (~) and two periods (..) to specify relative paths. It is better, however, to always show the absolute path names (from the root) to avoid problems if you change working directories or use the same configuration file in another startup directory.
- A directory path can contain up to 260 characters.

File Format

Saving an object to disk creates a file with the following format:

`object_name.object_type.version_number`

For example, if you create and then save the part *bracket*, the filename for the part is `bracket.prt.1`.

About File Types

The majority of Pro/ENGINEER files are object files that store information about an object. Other files created in Pro/ENGINEER are object-dependent, temporary files that contain information that may have been output by you, or created as a quick way to retrieve data without regenerating the object, such as the `.ptd` file for Family Tables.

You can remove temporary files from the system (identified with an asterisk in the following table), but you must never delete object files.

In the following table of file-naming conventions, words like `sectionname`, `partname`, and `filename` indicate the name you enter at the prompt for sections, parts and data files, or the active object used to create the file (for example, `partname.inf`).

File Name	Description
<code>Asm#####.asm</code>	Default assembly name.
<code>Assemblyname.asm</code>	Assembly with name entered by you.

File Name	Description
<i>Assemblyname.als</i>	Assembly, Program design file.
<i>Assemblyname.bom*</i>	Bill of Materials file.
<i>Assemblyname.inf*</i>	File created when you choose Info > Assembly .
<i>Assemblyname.ref*</i>	File created when you are saving an assembly by using File > Save a Copy . It lists the components that you selected for renaming.
<i>Assemblyname.ptd*</i>	Family Table file for assembly.
<i>Assemblyname.tst*</i>	Records results of Verify operation on instances of <i>assemblyname</i> .
<i>Assemblyinstancename.xas</i>	Instance accelerator file for assembly instance.
<i>Assemblyname_a.wrl</i>	VRML export file for an assembly.
Color.map	Color palette of user-defined colors.
<i>Componentname.memb*</i>	File created when you choose Info > Component .
Config.pro	Configuration options file.
Curv#.dat*	Data file containing curvature information for curve, where # is curve internal ID.
Dgm#####.dgm	2D schematic diagram with default name.
<i>Diagramname.dgm</i>	2D schematic diagram created with name entered by you.
Datafile.ers*	Summary of drawing setup file errors generated upon reading a setup file with errors.
Dimdiff.info*	Tabulation of dimension and feature differences between two parts, or two versions of the same part. Created when you choose Analysis > Part Comparison .
<i>Directoryname.idx</i>	Instance index file for directory

File Name	Description
	<i>directoryname.</i>
Drw####.drw	Drawing with default drawing name.
<i>Drawingname.drw</i>	Drawing with name entered by you.
Feature.lst*	File created when you choose Info > Feature List .
Feature.inf*	File created when you choose Info > Feature .
<i>Filename_amp.dat*</i>	Assigned mass properties file.
<i>Filename.cbl</i>	Cable parameters file.
<i>Filename.con</i>	Connector parameters file.
<i>Filename.dat*</i>	Data files created for editing, such as relations data.
<i>Filename.dtl</i>	Setup file used by drawings, formats, and layouts.
<i>Filename.gph</i>	User-defined feature with name entered by you.
<i>Filename.ibl</i>	File containing data for creating a blend feature.
<i>Filename.inf*</i>	Information files created using the Info menu.
<i>Filename.lgh</i>	File containing lights data.
<i>Filename.ncl*</i>	File containing manufacturing CL data.
<i>Filename.pts</i>	File containing spline points coordinate data with name entered by you.
<i>Filename.scl</i>	File containing definitions for system colors.
<i>Filename.shd*</i>	Shaded view file.
<i>Filename.sym</i>	Symbol file storing geometry of drawing symbol.
<i>Filename.tph*</i>	File containing manufacturing toolpath information for model <i>filename</i> .

File Name	Description
<i>Formatname.frm</i>	Format used by drawings and layouts.
Iges_stats.dat*	A summary of processing an IGES file.
Lay####.lay	Layout with default layout name.
<i>Layoutname.lay</i>	Layout with name entered by you.
<i>Machinename.mac</i>	File containing data for a manufacturing machine.
<i>Manufacturename.mfg</i>	Manufacturing object with name entered by you.
<i>Markupname.mrk</i>	Drawing markup file. Created in Markup mode.
Menu_def.pro	Default menu options file.
<i>Modelname.wrl</i>	File created when you export a model to the VRML format.
Param.inf*	File created when you choose Info > Relations and Parameters .
<i>Partinstancename.xpr</i>	Instant accelerator file for part instance.
<i>Partname.bde</i>	File that lists the elements that exceed the aspect ratio bound of a meshed object.
<i>Partname.bdi</i>	File that lists the elements that have a Jacobian distortion index lower than preset bound for a meshed object.
<i>Partname.bdm</i>	File that lists the elements that have mid ratios higher than a preset bound for a meshed part.
<i>Partname.ger*</i>	Geometry misalignment data.
<i>Partname.pls</i>	Part, Program design file.
<i>Partname.prt</i>	Part with name entered by you.
<i>Partname.m_p*</i>	Part mass properties file.
<i>Partname.ptd*</i>	Family Table file for part.
<i>Partname.tst*</i>	Records results of Verify operation on instances of <i>partname</i> .

File Name	Description
<i>Partname_p.wrl</i>	VRML export file for part.
<i>Partname_pr#.wrl</i>	VRML resolution file for part.
<i>prt#####.prt</i>	Part with default part name.
<i>Rels.inf*</i>	Relations info file created when you choose Info > Relations and Parameters .
<i>Reviewref.info*</i>	File created when reviewing feature references during a regeneration failure.
<i>s2d#####.sec</i>	Sketch with default sketch name.
<i>Sectionname.sec</i>	Sketch with name entered by you.
<i>Spline.pts</i>	Default name for file containing spline points coordinate data.
<i>Toolname.tpm</i>	File containing data for a manufacturing tool.
<i>Training_file.txa</i>	Training file.
<i>Trail.txt*</i>	Default name for trail file.
<i>Xsectionname.m_p*</i>	Cross-section mass properties file.


Creating Files

About Creating New Files

You create models by building features. To create features, you must first begin a new part. You can create a new sketch, part, assembly, manufacturing model, drawing, format, report, diagram, layout, markup, or interchange file in Pro/ENGINEER using the **New** and **New File Options** dialog boxes.

When creating a new object, a template is automatically supplied for any object type supported by a template. You can accept the default template, select another, or browse to a file you want to use as a template.

To Create a New File

1. Click  or click **File > New**. The **New** dialog box opens.
2. Select the type of file to create. If **Sub-types** are available, they are listed.

Note: If you click through the file types, you can see the default name for each file type in the **Name** box. The default prefix indicates the file type. For example part prt0001, saves as file prt0001.prt, and assembly mfg0001 saves as file mfg0001.mfg.

3. Select a **Sub-type**, if available.
4. In the **Name** box, type a name for your file or use the default name. If you accept the default template or if **Use default template** is not available, go to step 10. To use another template go to step 5.
5. Clear **Use default template** and click **OK**. The **New File Options** dialog box opens.

Note:

- If an object type is not supported by a template, then **Use default template** is unavailable.
 - To see the **New File Options** dialog box by default for template-supported types, set the configuration option `force_new_file_options_dialog` to Yes.
6. Type the name of the template file, select a template file, or browse to a file and then select it as a template file. For each template two files are provided, one in metric (mmns) and one in nonmetric (inlbs).
 7. Assign parameter values or change parameter values, if required. All standard part and assembly templates contain **DESCRIPTION** and **MODELED BY** parameters, which are automatically passed into Pro/INTRALINK and Pro/PDM.
 8. Select **Copy Associated Drawings** to create a drawing of a new part automatically when you select a template containing a drawing with the same name. For example, if you select the template `inlbs_part_solid.prt`, and the template directory contains a corresponding drawing template `inlbs_part_solid.drw`, select **Copy associated drawings** to automatically create a drawing with the same name.

Note: **Copy associated drawings** is cleared by default. Use the configuration option `rename_drawing_with_object` to reset the default value.

9. Click **OK**. The Pro/ENGINEER graphics window opens and the object appears.

Note: In some cases another dialog box may open and require input before the Pro/ENGINEER graphics window opens.

To Copy into an Empty Part or Assembly

Use **Copy From** to import a part or assembly, including the geometry and features, into the active, empty, part or assembly. You can copy a part, only into a part, and an assembly, only into an assembly. The target part or assembly must not have a default template.

1. Click **File > New**. The **New** dialog box opens.

2. Clear **Use default template**.
3. Select **Part** and click **OK**. The **New File Options** dialog box opens.
4. In the **Template** section of the dialog box, select **Empty** and click **OK**. The Pro/ENGINEER window is displayed. A part appears in the Model Tree.
5. Click **File > Copy From**. The **Choose Template** dialog box opens.
6. Select the part that you want to copy and click **Open**. The part is displayed in the Pro/ENGINEER graphics window. The message *All existing parameters and layer information will be lost when the component is filled* appears. This message refers to the parameters and layer information in the target (empty) part.

Note: When using the **Copy From** command:

- The part you are copying from cannot have external dependencies, such as a Merge feature.
- The assembly you are copying from cannot have external dependencies, such as components.
- No family table information is copied.

Object Templates

You can use Pro/ENGINEER object templates to customize the default content of newly created objects. Object templates belong to one of two categories: model templates and drawing templates. Model templates are standard Pro/ENGINEER models that contain predefined features, layers, parameters, named views, and other attributes. Drawing templates are special drawing files that contain instructions to create drawing items, including views, tables, formats, symbols, snap lines, notes, parametric notes, and dimensions.

PTC-provided model templates include default datum planes, named views, default layers, default parameters, and default units. The PTC standard drawing templates include three orthogonal views and can be further customized.

The following Pro/ENGINEER object types support the use of templates:

- Solid parts
- Sheetmetal parts
- Bulk parts
- Design assemblies
- Mold manufacturing objects
- Cast manufacturing objects
- Drawings

Configuration Options for Object Templates

To control object template display and storage, use the following configuration options.

- `start_model_dir`—Specifies the directory in which to store templates.
- `force_new_file_options_dialog`—Forces automatic display of the **New File Options** dialog box
- `template_solidpart`, `template_sheetmetalpart`, `template_designasm`, `template_mfgcast`, `template_mfgmold`, `template_drawing`—Specifies the model file to use as the default template.

Opening Files

About Opening Files

You can open files across platforms, retrieve files from other directories, and retrieve earlier versions of a file. In addition, you can use wildcards to retrieve objects and to perform procedures to save time when retrieving graphics. To open a recently saved file, click its name at the bottom of the **File** menu.

You can retrieve files of all types from the current session (in memory) and from disk using the **File Open** dialog box (**File > Open**). You use this dialog box to:

- Navigate your disk and directory structure.
- Filter displayed file types.
- Preview Pro/ENGINEER files.
- Select and open a file.
- Import files into Pro/ENGINEER.
- Create or select simplified representations of files.


To Open a File

\$\$\$ProCmdModelOpen@CMD,ProCmdRecentFileOpen@CMD,file_open@DLG,file_open@In_Session,file_open@desktop_pb,file_open@mydocs_pb,file_open@workingdir_pb,file_open@network,file_open@System
Formats_shortcut,file_open@view_favorites\$\$\$

1. Click **File** and select one of the files at the bottom of the **File** menu. The four most recent files that you opened are listed. To open a file not on this list, go to step 2.

Note: When you create a file, you must save it before it appears in the most recent list on the **File** menu.

2. Click  or **File > Open**. The **File Open** dialog box opens. The directory in the address bar defaults to one of the following items:

- **My Documents** (Windows platforms only) if you have not set a working directory or previously saved an object to another directory in your current Pro/ENGINEER session.
 - The working directory that you set for your current session.
 - The directory you last accessed to open, save, save a copy, or back up your file.
 - The directory set with the `file_open_default_folder` configuration option.
 - The directory you set using **Tools > Address Default** in the **File Open** dialog box. This setting applies to the current session only and overrides the `file_open_default_folder` configuration option for the current session.
3. Locate the file to open in the default directory or select a different directory using a method below:
- Click a folder under **Common Folders**.
 - Click any directory in the address bar and then select another directory or a file. The path in the address bar changes as you select directories.
 - Click  to toggle the address bar to display a directory path that you can edit directly.
 - Click **Folder Tree** and select a folder to browse. To add a folder to the list of common folders select the folder, right-click, and choose **Add to Common Folders** from the shortcut menu.

Note:

- To narrow your search, select a file type from the **Type** box and a subtype from the **Sub-type** box. Only selected types are listed in the directory.
 - To quickly locate a file, type the file name into the **Search** box. The files in the current directory are filtered according to the typed text.
 - To change the view, organize, or perform other actions in the directory, click **View**, **Organize**, or **Tools**. You can also right-click a file in the directory and use the shortcut menu.
4. To open the file, double-click it or click **Open**. The object appears in the graphics window.

Note: To select a simplified representation of the object, select the object, click **Open Rep**, select a type of representation from the **Open Rep** dialog box, and click **OK**.

Transferring Files Using FTP

Pro/ENGINEER object files are saved in a binary format. This format can be transferred to and used on other hardware platforms. The files are guaranteed to be compatible with new Pro/ENGINEER software revisions.

If you use FTP (File Transfer Protocol) to transfer files, make sure that you set your FTP environment to transfer binary file types, and not ASCII files. When performing a file transfer using FTP, be aware that the Pro/ENGINEER file header may appear to be ASCII, but the file is actually in binary format.

Retrieving an Earlier Version of a File

Although Pro/ENGINEER automatically retrieves the latest saved version of an object, you can retrieve any previous version by entering the full file name with extension and version number (for example, `bracket.prt.5`).

If you do not know the specific version number, you can enter a number relative to the latest version. For example, to retrieve a part from two versions ago enter `partname.prt.-2`.

Note:

- As a delimiter for the version number, use a period (.).
- If an object is in the current session (in memory), Pro/ENGINEER ignores any version number you add when you are retrieving it.
- If you retrieve a file from a non-working directory, rename the file and then save it, the renamed file is saved to the directory from which it was originally retrieved. It is not saved in the current working directory.

You can also retrieve another version of a part already in memory as long as you give the part a new name. The part is retrieved into a subwindow under the new name.

Retrieving Files Across Platforms

To successfully retrieve Pro/ENGINEER files created on a UNIX system and transferred to a Windows system, the files must be transferred using a binary method. However, files transferred without using a binary method can be corrected.

If during file retrieval Pro/ENGINEER determines a correction is required, a dialog box opens giving you the opportunity to correct the error.

You can choose to cancel the retrieval process and correct no files or you can continue the file retrieval process, allowing Pro/ENGINEER to correct and retrieve the required files. If you choose to make the correction, all subsequent corrections in the current Pro/ENGINEER session are made without prompting. Other files that depend on the file being retrieved (such as part files for an assembly) are automatically corrected and retrieved.

Note:

- The correction process increases the total retrieval time for the Pro/ENGINEER model and may decrease the amount of available temporary swap space on your system.
- You may not be able correct the error when retrieving a document.

Using the Command Line Utility

You can use the command line utility `dos_to_binary` to correct files individually or to correct all files in a specified directory. The syntax for this utility is `dos_to_binary [-inf] [-r] file.prt.1`, where `file.prt.1` is the name of the file to be corrected. If a file version is not specified, the latest version is used.

Multiple files can be corrected by specifying each file name and separating each one with a space. If you specify a directory name, using the `-r` option, all files in that directory will be corrected.

When the command line utility is used, the specified files are checked to determine if correction is necessary. If correction is necessary, the file is corrected and saved as an incremented revision. For example, if `sphere.prt.2` is corrected, the corrected file is named `sphere.prt.3`.

Note: The file is not marked as being modified for Pro/PDM or Pro/INTRALINK.

Retrieving Objects Using Wildcards

When you are retrieving objects, you can enter wildcard characters for object names, their extensions, and directory names. You cannot use wildcards in place of version numbers.

The match syntax is

- An asterisk (*) can replace any number of characters.
- A question mark (?) can replace any one character. When the question mark is entered alone, you get the complete listing of the appropriate objects in the directory.
- If you retrieve a file from a non-working directory, rename the file and then save it, the renamed file is saved to the directory from which it was originally retrieved. It is not saved in the current working directory.


Simplified Representations

Simplified representations improve regeneration, retrieval, and display time and let you tailor your work environment to include only the information of current interest to you.

For example, you can remove a subassembly that you are not working on from an assembly.

You can simplify the geometry of a part by excluding some of its features or by defining a specific work region. In drawings, you can create views of simplified representations.

To Open a File Containing a Family Table (Instances)

1. Click  or **File > Open**.
2. Select the file containing the Family Table. The **Select Instance** dialog box opens.

3. Do one of the following:
 - Click the **Name** tab and select the generic.
 - Click the **Name** tab and select an instance.
 - Click the **By Parameter** tab, and select the desired parameter and value. All instances containing the specified parameter and value appear in the **Name** list. Select the instance you want to open.


About Previewing Files

Use **Preview** in the **File Open (File > Open)** dialog box to preview parts, drawings, and assemblies before opening them or to search for a file if you do not know the file name. Previewing while browsing is convenient when you must open a Pro/ENGINEER object file to incorporate into a design, for example, or when you must specify a component to assemble.

Note:

- Previewing is available for files saved in a version of Pro/ENGINEER that supports previewing.
- Objects appear in the **Preview** window as they were saved. For example, if the layer for datum curves was blanked when the file was saved, that layer is blanked in the preview.

To Preview Files

1. Click  or **File > Open**. The **File Open** dialog box opens.
2. Select the file from the file list or type the file name in the **File name** box.
3. Click **Preview**. The object appears in the preview area.
4. To change the preview configuration, right-click, and choose your preferred settings from the shortcut menu:

Display style:

- **Shaded** (default)
- **No Hidden**
- **Hidden Line**
- **Wireframe**

Projection style:

- **Orthographic** (default)
- **Perspective**

Frame rate:

- **Frames per second**

View type:

- **Dynamic**
 - **Anchored**
 - **Delayed**
 - **Velocity**
 - **Fly Throug**
 - **Standard** (default)
5. To return to the default preview orientation, right-click, and choose **Refit** from the shortcut menu.

Previewing Assemblies

When you preview assemblies you can focus on individual assembly components. Click an assembly component in the preview area to get a preview of that component only.

For optimal assembly preview performance, set the `save_model_display` configuration option to `shading_lod`.

Previewing Drawings

To preview a drawing the drawing must be saved with the `save_drawing_picture_file` configuration option set to `embed`, `export`, or `both`. By default, the first sheet of the drawing appears in the preview area. You can pan and zoom the previewed drawing as you would in the Graphics window. To change sheets, right click and choose a sheet from the shortcut menu.

About Searching For and Retrieving Models

From Pro/ENGINEER you can open a model from another directory by specifying a directory path when you enter the model name. If you do not specify the directory, Pro/Engineer searches for the model in the following areas in the order shown:

1. Directory appearing in the **Look In** box in the **File Open (File > Open)** dialog box
2. Pro/ENGINEER session
3. Active workspace or commonspace
4. Current **Working Directory**
5. Search Paths

About the Search Path and search.pro

During model retrieval, Pro/ENGINEER searches for and then retrieves submodels from the following areas in the order shown:

1. Pro/ENGINEER session
2. Directory in which the top model is being retrieved
3. Active workspace or commonspace
4. Current **Working Directory**
5. Other search paths and directories such as:
 - **Specific Directories**—Set the configuration option `search_path` and Pro/ENGINEER searches in the directories as specified.
 - **search.pro file**—Set the configuration option `search_path_file` and Pro/ENGINEER searches in the `search.pro` file.
 - **Libraries**—Set the configuration option `pro_library_dir` to set the default directory for Pro/ENGINEER libraries, or for libraries you have built.
 - **Drawing Format Library**—Set the configuration option `pro_format_dir` to set the default directory for the drawing format library.

If the assembly components or drawing models are not in any of the above areas, you must retrieve them before retrieving the assembly or drawing.

Adding Search Paths to Pro/ENGINEER

If you want Pro/ENGINEER to search additional paths besides the working directory, such as a server, you can add paths using one of the following methods:

- Within a Pro/ENGINEER session, specify additional directory paths with the configuration option `search_path`.
- Type the paths directly into the `search.pro` file and set the configuration file option `search_path_file`. `search.pro` is an editable text configuration file that contains relative and absolute paths to search for models. You may specify either the full path name for the file or just the full path name of the directory containing the file. You can search for models on registered Windchill server cabinets and workspace, and other sites and servers.

Directory path names can be relative or absolute. A path name can contain up to 260 characters. When you specify a relative path name, you can use special characters, such as `..` in UNIX and Windows (NT, 95, 98, and 2000).

Relative pathnames are initially resolved relative to the startup directory. If you subsequently reload the configuration file, the system reevaluates the relative pathnames relative to the current (working) directory and appends the new directories (if any) to the search path (the previous path remains in place). It is better to always specify the full pathnames (from root) so as to avoid problems if you change working directories or use the same configuration file in another startup directory.

In Windows NT/95/98/2000 systems, specify the drive at the beginning of the path to avoid problems if you change your working directory to a directory in another drive.

It is possible for each configuration file read in at startup to call out its own search path directories or `search.pro` file. As Pro/ENGINEER processes the first configuration file, it loads any specified `search_path` directories and the contents of the specified `search.pro` file (if any) into its search path. As it processes subsequent configuration files, it likewise appends any specified directories to the search path. If a particular directory is specified more than once, it appears more than once in the search path. All search-path directories are listed and processed in the order in which they were read.

If an object can be found in more than one search path directory, the system retrieves the first one it finds, regardless of which object is the most recent.

Note: Search paths can be disabled by inserting a `!` character in front of the appropriate line or lines in the `search.pro` file.

The primary server is always included in the search path without being specified in the `config.pro` file.

To Load a `search.pro` File

Pro/ENGINEER does not automatically read the `search.pro` file at startup. You must load the `search.pro` file if you want Pro/ENGINEER to read it.

1. Click **Tools > Options**. The **Options** dialog box opens.
2. In the **Options** box, type the configuration option `search_path_file`. For a value, specify either the full path name for the file or the full path name of the directory containing the file.

You must enclose in quotation marks any search path with a separator character (space, comma, or semicolon) in a directory name, for example:

- o **Windows:** `search_path_file "C:\Program Files\proe2001\models"`
- o **UNIX:** `search_path_file "/home/user/smith/odd,dir;name/search.pro"`

Note: For Windows NT, omit the last backslash (`\`) from the path, or enclose the path in quotation marks, or add a trailing space after the backslash.

3. Click **Apply**. Pro/ENGINEER reads the `search.pro` file automatically.

When you load a `search.pro` file, Pro/ENGINEER appends the contents of the `search.pro` file to the search path. Pro/ENGINEER reads any paths that you added and ignores any paths that you commented out.

To Edit a `search.pro` File

1. Open the `search.pro` file from within a text editor such as Notepad or Microsoft Word.
2. Add or delete search paths, or designate lines that contain search paths as comment lines only, as desired.

3. Save the file. When you save it, you can either accept the default name `search.pro` or assign a new name to the file.

Note:

- Only one directory path (search path) is allowed per line.
- You can insert comments into the file. Simply precede the line with a `!` and Pro/ENGINEER interprets the line as a comment and ignores it. Comment lines must begin with `!` They may be interspersed throughout the file.
- You can specify that certain paths in the file not be read at startup, by designating the lines containing the search paths as comments only. Pro/ENGINEER ignores the comment lines.
- Lines may also be blank, except for the first line.
- You must enclose in quotation marks any search path with a separator character (space, comma, or semicolon) in a directory name, for example:
 - Windows: `"C:\Program Files\proe2001\models"`
 - UNIX: `"/home/user/smith/odd,dir;name/search.pro"`

Note: For Windows NT, omit the last backslash (`\`) from the path, or enclose the path in quotation marks, or add a trailing space after the backslash.

- When you are defining a search path in the Windows NT operating system, you must precede the delimiting backslash character with another formatting character.
- Directory path names can be relative or absolute and can include special characters such as `..` in UNIX, to specify a relative path. However, it is better to always include the absolute path names (that is, from root) to avoid problems if you change working directories or if you use the same configuration file in another startup directory.
- Search paths may also include previously defined environment variables. This is done by preceding the variable with `$` in the search path definition in UNIX, or enclosing it between `%` characters in Windows NT/95/98/2000. For example, the environment variable `OBJ_TYPE` can be used as follows:

(UNIX)/partlib/\$OBJ_TYPE/objects

(Windows NT/95/98/2000)D:\partlib\%OBJ_TYPE%\objects

- You can create multiple `search.pro` files and store them in different directories. However, you may instead want to use the same `search.pro` files for all your projects, and simply comment out the paths in the file that you do not want Pro/ENGINEER to read for certain sessions.

To Temporarily Disable a Search Path

In Pro/ENGINEER, you can temporarily prevent one or more paths from being read at startup.

1. Open the `search.pro` file from within a text editor such as Notepad or Microsoft Word.
2. Comment out the lines that contain the paths you want to suppress. To comment out a line, place a `!` character at the beginning of the line.

To re-enable the path, delete the `!` character from the line that contains the path you want to re-enable. Pro/ENGINEER reads the path at startup.

Example: Adding Search Paths

For maximum clarity, you should specify only one directory path name in the **Value** field in the **Options** dialog box. You can set the option any number of times in the configuration file. You can also set the configuration option `search_path_file` to a file that contains multiple search paths.

The two directories shown in the following example are added by setting the configuration option twice.

```
search_path /components/washers
```

```
search_path wtpub://<alias> where <alias> is the name you specify in the site registry dialog.box.
```

Separator Characters and Search Paths

You must enclose in quotation marks any search path with a separator character (space, comma, or semicolon) in a directory name, for example:

- Windows: `search_path "C:\Program Files\proe2001\models"`
- UNIX: `search_path "/home/user/smith/odd,dir;name"`

Note: For Windows NT, omit the last backslash (`\`) from the path, or enclose the path in quotation marks, or add a trailing space after the backslash.

Using More Than One Path

The system permits you to enter several path names on a single line, separated by commas, or semicolons. You must use the same delimiter within the same line to separate path names. For instance, you cannot use a space and a comma in the same line. If you wanted to include the directories `/components/washers` and `wtpub://windchill_server` but you wanted to set the `search_path` option only once, you would add the two directories at the same time in the **Value** field, as shown in the following example:

```
search_path /components/washers, wtpub://windchill_server
```



Note: Make sure you separate the directory paths with spaces, semicolons, or commas.

Selecting the Working Directory


To Select a Working Directory

Your working directory is an area you designate to store Pro/ENGINEER files. Typically, the default working directory is directory in which you start Pro/ENGINEER. To select a different working directory for your current Pro/ENGINEER session, use one of the following procedures.


To Select a Working Directory from the Folder Navigator

1. Click . (You may need to click  first to see the Folder Navigator.) The Folder Navigator appears.
2. Select the directory you want to set as your working directory and right-click. A shortcut menu appears.
3. Click **Set Working Directory**. A message confirms the working directory is changed.

To Select a Working Directory from the File Menu

1. Click **File > Set Working Directory**. The **Select Working Directory** dialog box opens.
2. Browse to the directory you want to set as the new working directory. A folder followed by a period () is displayed indicating the location of the working directory.
3. Click **OK** to set as the current working directory.

Note:

- The new working directory setting is not saved when you exit Pro/ENGINEER.
- If you retrieve a file from a directory other than your working directory and then save the file, the file saves to the directory from which it was retrieved. If you save a copy and rename the file, the copy saves to the current working directory.
- You can access the working directory from the **File Open**, **Save Object**, **Save a Copy**, and **Backup** dialog boxes by clicking .

Mirroring Parts

To Mirror a Part

When you mirror a part, the new part is a mirror image of the original part and can be independent of or dependent on the original part.

1. Open a part.
2. Click **File > Mirror Part**. The **Mirror Part** dialog box opens

3. Select one of the following combinations of options and then follow the appropriate steps.

Click **Mirror Geometry Only** and clear the **Geometry Dependent** check box.

- a. Accept the default **New Name** or type a new name.
- b. Click **OK**. A new window opens with the mirrored part. No features from the original part appear and the only feature in the Model Tree is **Mirrored Merge**. You can select the **Mirrored Merge** feature and right-click to select options such as **Open Base** (opens the original part from which this part was copied) or **Edit Definition**. If you change the geometry in the original part, it does not change in the mirrored part.

Click **Mirror Geometry Only** and click the **Geometry Dependent** check box.

- a. Accept the default **New Name** or type a new name.
- b. Click **OK**. A new window opens with the mirrored part. No features from the original part appear. The only feature in the model tree is a **Mirrored Merge** feature. You can select the **Mirrored Merge** feature and right-click to select options such as **Open Base** (opens the original part from which this part was copied) or **Edit Definition**. If you change feature dimensions, for example, in the original part, the changes are propagated to the mirrored part. If you make changes in the mirrored part, they are propagated to the original part.

Click Mirror **Include all Feature data**. The **Geometry Dependent** checkbox is not available.

- a. Accept the default **New Name** or type a new name.
- b. Click **OK**. A new window opens with the mirrored part. Features in the original part are propagated to the mirrored part and a **Mirrored Merge** feature is added as the last feature in the Model Tree. The original and mirrored part are independent of each other. Changes in one do not affect the other.

Deleting Files

About Deleting Files

Erasing Objects in Memory

An object is a file you create using a Pro/ENGINEER.

Every time you save an object, you create a new version of the object and write it to the disk. Pro/ENGINEER numbers each version of an object storage file consecutively (for example, `box.sec.1`, `box.sec.2`, `box.sec.3`).

When you close a window using the **Close Window** command, the object is no longer displayed but it remains in memory during the current session. Erasing an object removes it from memory but not from disk.

Note:

- You cannot erase objects while assemblies or drawings that are referencing them are still active.
- Erasing an object does not necessarily erase from memory those objects to which it refers (for example, erasing an assembly does not necessarily erase its components).

Deleting Objects

Every time you save an object, you create a new version of the object in memory, and write the previous version to disk. Pro/ENGINEER numbers each version of an object storage file consecutively (for example, `box.sec.1`, `box.sec.2`, `box.sec.3`).

You can use the **Delete** command to free up disk space and remove old, unnecessary versions of objects:

You can quickly purge Pro/ENGINEER files on disk (any files with version numbers) from an operating system window using the purge script, which is loaded by Pro/ENGINEER at installation.

To use the purge script, type `purge string`, where `string` is a filter to match for files.

If you do not include a string, the system purges all but the latest version of assembly files, part files, datum curve files, and so on. If you do include a string, the system purges only those files that match the `string`. The string can contain wildcards: (*) matches any number of characters, (?) matches a single character. To prevent wildcard interpretation in UNIX, use a backslash in front of (*) and (?).



To Erase the Current Object from Memory

To erase the top-level object and selected sub-components displayed in the active window:

1. Click **File > Erase > Current**.
2. Click **Yes**. If you are working in a part, the part is erased from the graphics window. If you are working in an assembly, manufacturing model, or drawing, the **Erase** dialog box opens.
3. Select the **Associated Objects** (objects reference by the current object) that you also want to erase from memory.
4. Click **OK**.

Note:

Click **Tools > Customize Screen** to add the following to the toolbar:



- —Erase Current
- —Erase Not Displayed

To Erase Undisplayed Objects from Memory

Use the **Erase Not Displayed** dialog box to erase all objects from the current session, except for objects that are currently displayed and any objects referenced by the displayed objects. For example, if an assembly instance is displayed, the generic of an instance and its components are not erased.

1. Click **File > Erase > Not Displayed**. The **Erase Not Displayed** dialog box opens.
2. Click **OK**. If you have set the configuration option `prompt_on_erase_not_disp` to `yes`, you are prompted to save each object before it is erased.



Notes:

- Click **Tools > Customize Screen** to add the following to the toolbar:
 - —Erase Current
 - —Erase Not Displayed
 - Set the configuration file option `prompt_on_erase_not_disp` to `yes` to display a prompt for each modified, unsaved object, allowing you to save the object before it is erased. When set to `no` (the default), Pro/ENGINEER immediately erases all undisplayed objects.

To Delete Old Versions of a File

Every time you save an object, you create a new version of the object in memory, and write the previous version to disk. Pro/ENGINEER numbers each version of an object storage file consecutively (for example, `box.sec.1`, `box.sec.2`, `box.sec.3`).

Follow the steps below to delete all versions of an object except the most current version (the version with the highest version number).

1. Click **File > Delete > Old Versions**. A confirmation prompt appears.
2. Click  to delete older versions of the current object, or type the name of a different object and click .

Note: You cannot delete a part or subassembly that has been used in an assembly or drawing during the current working session until the assembly or drawing is deleted.

To Delete All Versions of a File

To delete all versions of the current object:

1. Click **File > Delete > All Versions**. A `Delete All Confirm` message appears.
2. Click **Yes** to delete all versions of the current object.

Note:

- You cannot delete a part or subassembly that has been used in an assembly or drawing during the current working session until the assembly or drawing is deleted.
- If the object is an instance, neither the instance nor the generic is deleted. If an object is a generic and you select it for deletion, only the generic is deleted.

Saving, Exporting, Renaming, and Backing Up Files**About Saving Files**

You can save Pro/ENGINEER files using **Save** or **Save a Copy** commands. From the **Save a Copy** dialog box you can also export Pro/ENGINEER files to different formats, and to save files as images.

Pro/ENGINEER saves only changed objects, except in the following cases:


- An object selected for saving is not found in the destination directory
- The configuration file option `save_objects` is set to `all`
- The configuration file option `save_objects` is set to `changed_and_specified` and the current object is the top object in an assembly
- A change is made to a dependent object and the configuration file option `propagate_change_to_parents` is set to `yes`.

Saving an object on the disk creates a file with the format `object_name.object_type.version_number`. For example, if you create a part with the name `bracket`, it initially saves to the file name `bracket.prt.1`. Saving the same part again results in file name `bracket.prt.2`.

If you retrieve an object using a relative path specification (for example, `../partname`), the same path specification is used to save it in its original directory. So if you change your working directory between retrieving and storing, the object could be saved in a wrong directory. Be careful when you use relative path names.


Note: Path names can contain up to 260 characters.

To Save a File

1. Click  or **File > Save**. The **Save Object** dialog box opens.
 - If you previously saved there are no options available in the **Save Object** dialog to change directories. Click **OK** to complete the save.
 - If you have not previously saved the object, go to Step 2.
2. The directory in the **Look In** box defaults to one of the following:

- **My Documents** (Windows platforms only) if you have not set a working directory or previously saved an object to another directory in your current Pro/ENGINEER session.
 - The Working Directory you set for your current session.
 - The directory you last accessed to open, save, save a copy, or back up your file.
3. Accept the default directory or browse to a new directory.

Note: You can access the working directory by clicking .

4. In the **Model Name** box, the name of the active model appears. To select a different model click .

5. Click **OK** to save to the directory displayed in the **Look In** box or select a sub-directory and then click **OK**. The Pro/ENGINEER graphics area is displayed.


Note:


- Objects are stored in their original directories unless you set the configuration option `override_store_back` to `yes`.
- If you do not want your file to save to the last accessed directory, set the configuration option `file_open_default_folder`. Use this configuration option to specify the directory from which you want to open, save, save a copy, or backup files.
- If you do not have write permission to the original directory and have `override_store_back` set to `no`, set the configuration option `save_object_in_current` to `yes` to store the objects in the current directory.
- By default, Pro/ENGINEER saves the model on which a drawing is based only when changes have been made to it. You can use the configuration file option `save_modified_draw_models_only` to save the model every time the drawing is saved.
- To save disk space, compress file output by setting the configuration option `compress_output_files` to `yes`. Compressed files take longer to read and write, but are one-half to one-third the size of uncompressed files. They are also fully compatible across platforms.

To Save a Copy of a File

1. Click **File > Save a Copy**. The **Save a Copy** dialog box opens. The directory in the **Look In** box defaults to one of the following:
- My Documents (Windows platforms only) if you have not set a working directory or previously saved an object to another directory in your current Pro/ENGINEER session.
 - The Working Directory you set for your current session.

- The directory you last accessed to open, save, save a copy, or back up your file.
2. Accept the default directory or browse to a new directory.

Note: You can access the working directory by clicking .

3. In the **Model Name** box, the name of the active model appears. To select a different model click .
4. In the **New Name** box, type the new file name.


Note: An error message appears if you type the name of a model currently in session.

5. Click **OK** to save to the directory displayed in the **Look In** box or select a sub-directory and then click **OK**. The Pro/ENGINEER graphics area is displayed.

Note:

- To save a file to a different disk or directory using the same file name, use **File > Backup**.
- If you do not want your file to save to the last accessed directory, set the configuration option `file_open_default_folder`. Use this configuration option to specify the directory from which you want to open, save, save a copy, or backup files.

To Rename a File


1. Click **File > Rename**. The **Rename** dialog box opens and the current model name appears in the **Model** box.
2. To select a different model click .
3. In the **New Name box**, type the new file name.
4. Click either **Rename in session** or **Rename on disk and in session**.

Note: Problems result if you rename a file on disk and then retrieve a model (not in session) that depends on the previous file name. For example, a part cannot be found for an assembly.

5. Click **OK**.

Note:

- If you retrieve an object from a non-working directory, rename and save the object, it saves to the original directory from which it was retrieved and not in the current working directory.
- You cannot save or rename a file using the original file name, even if you save the file in a different directory.


- To add  (Rename) to your toolbar, click **Tools > Customize Screen**.

To Back Up a File

To make a backup copy of an object file that is in memory:

1. Click **File > Backup**. The Backup dialog box opens. The directory in the **Look In** box defaults to one of the following:
 - **My Documents** (Windows platforms only) if you have not set a working directory or previously saved an object to another directory in your current Pro/ENGINEER session.
 - The Working Directory you set for your current session.
 - The directory you last accessed to open, save, save a copy, or back up your file.
2. Accept the default directory or browse to a new directory for which you have write access.

Note: You can access the working directory by clicking .

3. In the **Model Name** box the name of the active model appears. To select a different model click .
4. Click **OK** to backup to the directory displayed in the **Look In** box or select a sub-directory and then click **OK**. The Pro/ENGINEER graphics window is displayed.

Note:

- Versions of the backed up object are reset in the backup directory.
- If you back up an assembly, drawing, or manufacturing object, Pro/ENGINEER saves all dependent files in the specified directory.
- If an assembly has related interchange groups, those groups are not saved in the backup directory when you back up the assembly.
- If you make changes to a model after backing it up and then save the model, the changes are always saved in the backup directory.

Printing and Plotting Files

About Printing and Plotting

You can print files in the following ways:

- From the **File** menu you can print with the following options: scaling, clipping, displaying the plot on the screen, or sending the plot directly to the printer. You can also print shaded images from this menu. You can create plot files of the current object (sketch, part, drawing, assembly, or layout) and send them to the

print queue of a plotter. The plotting interface to HPGL and PostScript formats is standard.

- Use the Pro/BATCH utility to create a command file of print or plot files without having to choose from the interactive menus. The command file contains a list of objects to plot. For example, you can submit the job at the end of a workday, and the plot files are created and plotted offline.

Notes for plotting:

- Hidden lines appear as gray for a screen plot, but as dashed lines on paper.
- You cannot plot when the option **Use Fast HLR** is checked in the **Environment** dialog box.
- When Pro/ENGINEER plots the system line fonts, it scales them to the size of a sheet. It does not scale the user-defined line fonts, which do not plot as defined.


You can use the configuration file option `use_software_linefonts` to make sure that the plotter plots a user-defined font exactly as it appears in Pro/ENGINEER.

- With the **CROSS SEC** menu active, you can plot a cross section from Part or Assembly mode.
- You can write plot files in Calcomp, Gerber, HPGL2, and Versatec format if you have the Pro/PLOT module. Pro/PLOT is part of the Foundation II license.
- For the HPGL2 driver to print a screen capture of OLE objects, the printer must support the HP RTL extension.

Configuring the Printer

You can configure your printer using the **Printer Configuration** dialog box, available from the **Print** dialog box. If you are printing a shaded image, the **Shaded Image Configuration** dialog box opens instead of the **Printer Configuration** dialog box.

To Print an Object: Basic Procedure

1. Click **File > Print**. The **Print** dialog box opens.
2. Click . A shortcut menu appears.
3. Select another printer or add a new printer. The selected printer type is saved for the current session only.
4. Click **Configure**. The **Printer Configuration** dialog box opens.

Note: If you are printing a shaded model, the **Shaded Image Configuration** dialog box opens instead. For more information about printing a shaded image, refer to the topic *To Print a Shaded Model*.

5. Use the **Printer Configuration** dialog box to configure the printer page, to format and scale the model for printing, and to set other printer options, as necessary.
6. Click **OK**.
7. In the **Print** dialog box, set other print options as necessary.
 - If you select **To file**, complete the dialog box to specify the contents and structure of the plot files.
 - If you select **To printer**, specify the sheets (if available), quantity, and command for the print.
8. Click **OK**.

Note:

- Shaded plots cannot be generated using the **MS Printer Manager**.
- In the **Plotter Command** box you can type an operating system command for plotting (obtain this from your system administrator or the operating system guide for your workstation), or use the default command.

Printer Configuration Dialog Box

This topic shows you how to configure the printer page, format and scale a model for printing, and set other printer options when printing all objects except a shaded 3D model.

The **Printer Configuration** dialog box opens when you click **Configure** in the **Print** dialog box (**File > Print**). It contains the following tabs:

- **Page** tab—Provides options for configuring the printer page, such as paper size, height, width, and units (inches or millimeters).
- **Model** tab—Provides options for formatting and scaling the model for printing.
- **Printer** tab—Provides other printing options that you can set, such as:
 - Setting pen speed
 - Specifying whether a paper cutter is installed
 - Selecting the plotter initialization type
 - Selecting the paper type

Configuring the Printer Page: The Page Tab

The **Page** tab allows you to configure the printer page and provides the following options:

Option	Action
Size list	Allows you to choose a paper size from the following available sizes: A (default size), B, C, D, E, F, A0, A1, A2, A3, A4, and Variable.
Height	Allows you to specify a paper height for variable size (default is 8.5 inches).
Width	Allows you to specify a paper width for variable width (default is 11 inches).
Offset X	Allows you to specify a horizontal printer offset.
Offset Y	Allows you to specify a vertical printer offset.
Label Include	Allows you to Indicate whether or not to print a label (the default setting is no label).
Height	Allows you to specify the height of the label (the default height is .125 inches).
Units Inches or Millimeters	Allows you to specify units as inches or millimeters (default is inches).

Formatting and Scaling a Model for Printing: The Model Tab

The **Model** tab provides the following options for formatting and scaling the model:

Command	Action
Plot list	Choose type of plotter (see list and descriptions in the following section entitled Plot Types).
Plot Scale	Specify scale of plotter (default is 1).
Segmented	If applicable, choose segmented plotting.
With Format	If applicable, choose plotting with formatting.
Layers All Visible	Print all visible layers.
By ID	Print specified layer.
Layers By ID field	Type ID of layer to be printed.

Command	Action
Quality list	Choose interface quality for checking of overlapping lines. The options are No line check, No overlap check, Simple overlap check, and Complex overlap check (default is Complex overlap check).

Note: Imported geometry cannot be scaled.

Plot Types

The options on the plot list are as follows:

- **Full Plot**—Creates a plot of an entire object (the object plots as if you chose **View > Pan/Zoom > Refit**).
- **Clipped**—Creates a clipped plot. Define the boundaries by drawing a box around the area to plot. This area appears on paper in its normal location relative to the lower left corner.
- **Based on Zoom** (default)—Creates a scaled, clipped plot. The scaling and clipping are based on paper size and the zoom setting in the graphics window.
- **Paper Outline** (available in 2D mode only)—Creates a plot of a specific area within a drawing on a specified paper size. For example, if you have a large drawing, such as size A0, and you want to plot an A4 area of the drawing, you can use this option.
- **Plot Area**—Creates a plot by shifting the area inside the clip box to the lower left corner of the paper and scaling the clipped area to match the user-specified scale. Zoom and pan screen factors and scale in the model size are also factored into the plot area. Thus, if the scale=1, the geometry is not necessarily printed to scale.
- **Model Size**—Scales the plot to a specified model scale. For example, if you enter .25, the system creates a plot of the model to ¼ scale. This choice is valid only for 3D objects.

Note: Clipped and Plot Area are affected when you change both the sheet size and the scale factor. Plots are not to scale if you change the sheet size on the **Page** tabbed page. When either of these have been specified, after you click **OK** to close the **Print** dialog box, the system asks you to specify the corners of the area to be plotted. Place the cursor at one corner and drag the cursor to the diagonally opposite corner of the area to be plotted.

Setting Other Printer Options: The Printer Tab


The **Printer** tab provides the following options:

Command or Option	Action
Pen Table File check box	Use a default pen mapping table.

Command or Option	Action
Pen Table File field	Type the name of the table.
Velocity	If applicable, specify speed of pen (centimeters per second).
Cutter Installed	Specify whether or not a paper cutter is installed.
Handshake Xon/Xoff or Hardware	Select the type of plotter initialization.
Sheet Type Sheet or Roll	If applicable, select the paper type.
Rotation	Specify rotation of image (0 or 90 degrees).

To Print a Shaded Model

Printing a model that is shaded is different from printing a model that is set to display in **Wireframe**, **Hidden Line**, or **No Hidden**. With shaded models, you specify model scaling and formatting using the **Shaded Image Configuration** dialog box.

1. Click **File > Print**. The **Print** dialog box opens.
2. Click  to select a printer or to add a new printer. The selected printer type is saved for the current session only.
3. Click **Configure** to specify model scaling and formatting for printing. The **Shaded Image Configuration** dialog box opens.
4. Set model scaling and formatting as desired, using the following options in the **Shaded Image Configuration** dialog box.

Command	Action
Size	Specify a standard size or a custom size by selecting from the list.
Height	Specify custom height dimensions (if you select Variable in the Size list as a paper size).
Width	Specify custom width dimensions (if you select Variable in the Size list as a paper size).
Top Margin	Specify custom top margin dimensions (if you select Variable in the Size list as a paper size).
Left Margin	Specify custom left margin dimensions (if you select

Command	Action
	Variable in the Size list as a paper size).
Units	Set the variable plot size to inches or millimeters.
DPI	Specify the resolution of the output from the list. Choose from 100 dpi, 200 dpi, 300 dpi, 400 dpi, 500 dpi, or 600 dpi.
Image Depth	Specify the image depth for the print. Choose from 8 bit index or 24 Bit RGB.

To Print an Object to a File

1. Click **File > Print**. The **Print** dialog box opens.
2. Under **Destination**, select a printer, and then select **To file**.
3. Click **OK**. The **Print to File** dialog box opens.
4. Select one of the following:
 - **Create Single File**—Creates a single `.plt` file that contains all print information
 - **Create Separate Files**—Creates separate `.plt` files. Use this option if the plot will be on more than one page, such as if you print a drawing with multiple sheets, or if the plot is too large to fit on the specified paper size. In the former case, each drawing sheet is printed on a separate sheet of paper and saved to a separate `.plt` file. In the latter case, different parts of the plot are printed on different sheets of paper and saved to separate `.plt` files. The system uses the specified file name and appends numbers to each file name (such as `drawing.plt`, `drawing1.plt`, and so on).
 - **Append to File**—Appends the new plot to an existing `.plt` file. Typically, the `.plt` file is erased and recreated with the new plot information.
5. Under **Name**, accept the default file name or type a new name for the output file. Use the **Browse** button to search the directory structure to choose the location for the output file. This overwrites any default location. If you do not set the directory, the system stores the plot file in the current working directory and adds the `.plt` extension to the file name. If you want the extension of the plot file to depend on the plotter type, set the configuration file option `plot_names` to `yes`.
 - If you choose **Append to File** in the preceding step, click **Browse** to open the **Save** dialog box and select an existing `.plt` file to contain the additional `.plt` information for the active file.
 - If you choose **Create Separate Files**, the system creates a separate file for each print page in cases where the plot information is too big to fit on one page. The system appends a number to each file in order to create

separate files. In cases where you want to print a multisheet drawing file, the system places each drawing sheet in a separate plot file and appends the sheet number to each plot file name.

6. Click **OK**. The system saves the `.plt` file and displays a message that the file has been created.

Note: OLE objects are printed only as screen captures.

To Print Part of a Drawing

1. Click **File > Print**. The **Print** dialog box opens.
2. Click **Configure**. The **Printer Configuration** dialog box opens.
3. Click the **Model** tab.
4. Under **Plot**, open the **Plot** drop-down list box and select **Paper Outline**.
5. Click **OK** in the **Printer Configuration** dialog box, and then click **OK** in the **Print** dialog box. The **Paper Outline Size Selection** dialog box opens, and a movable rectangle appears in the Graphics window. By default, the rectangle is white; its color depends on whether you have created a user-defined color scheme in Pro/ENGINEER.
6. To define the area you want to print, click anywhere inside the rectangle and drag it to the desired position on the drawing.
7. You can change the size of the rectangle by choosing a paper size under **Size**. (If you choose **Variable**, specify a height and width in the Height and Width boxes, then specify a unit of size.)
8. Click **OK**. The **Paper Outline Size Selection** dialog box closes, and the **Print** dialog box opens.
9. Click **OK** to print.

Segmented and PostScript Plots

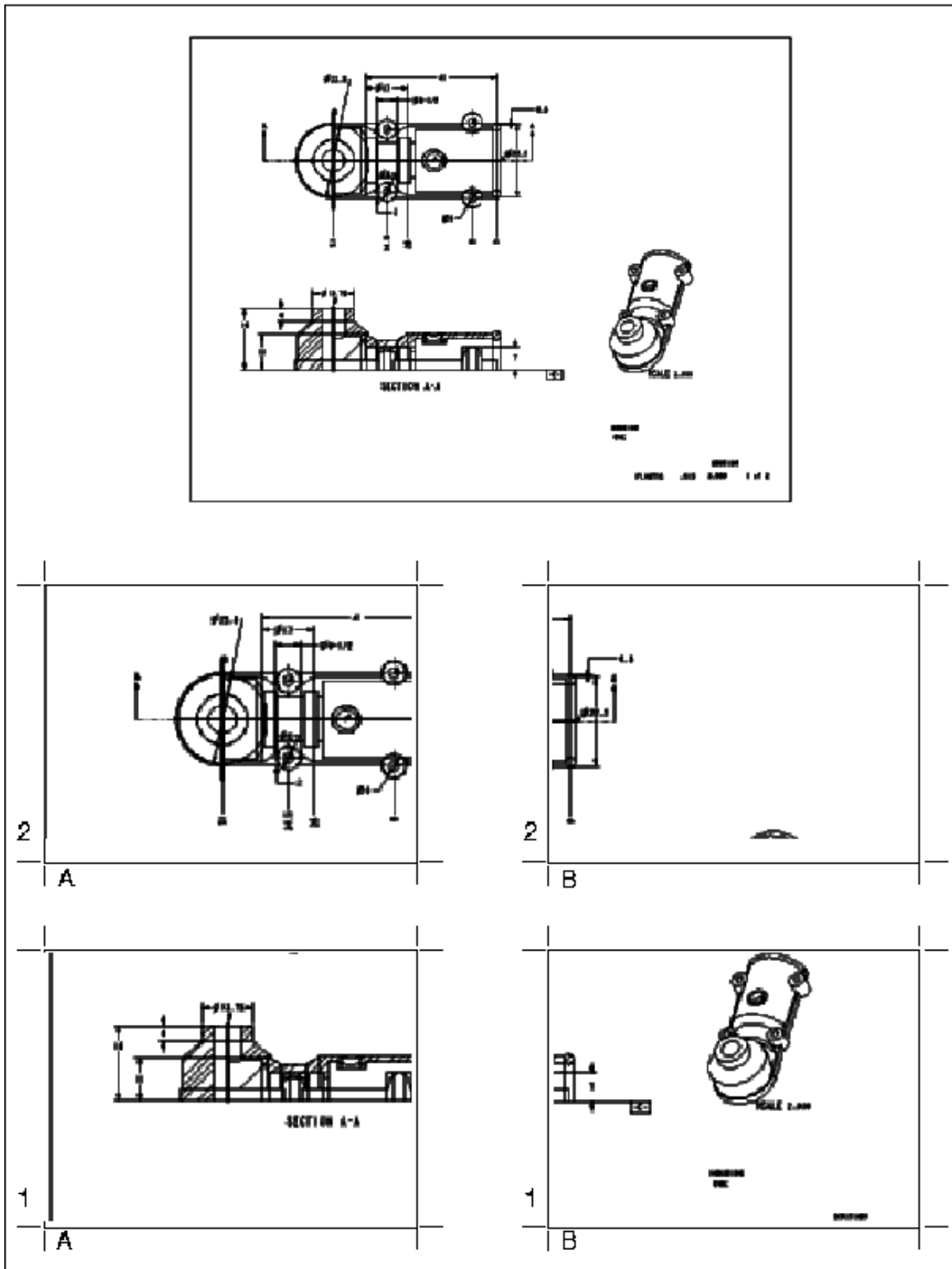
Example 1: Segmented Plots

When you want to plot a drawing full size but your plotter can only handle smaller paper sizes, select the **Segmented** check box in the Model page of the **Printer Configuration** dialog box. Pro/ENGINEER plots the drawing full size, but in segments that are compatible with the selected paper size. The following illustration shows how each segment is plotted with a border marking the cut-lines, tick marks indicating alignment, and row/column designators showing how to assemble the full-size plot. HP7475a and HP7440A plotters do not support segmented plots.

When specifying paper size, keep in mind that the segmented plot size is smaller than the actual paper size because of the border alignment marks.

The following diagram shows a C-size drawing.

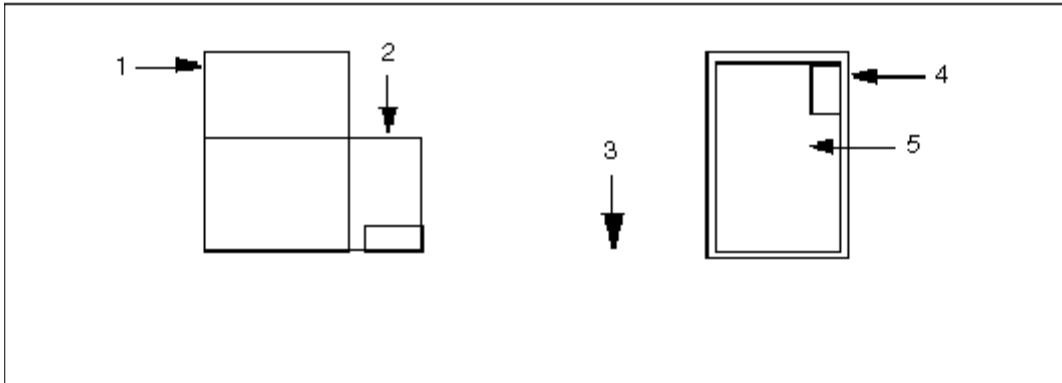
Note: Segmented plots are available only when plotting 2-D. It is not available in 3-D mode.



Example 2: PostScript Plots

You can generate PostScript plots for any plotter or laser printer that can handle PostScript data. You can also use them in documents created by electronic publishing

systems that can incorporate the PostScript data. The PostScript output format is 2.0 compliant (the bounding box directive is included). You might have to rotate PostScript plots to plot them correctly, as is shown in the following illustration.



The diagram on the left is a default PostScript plot. The diagram on the right is a PostScript plot after rotating.

1. Paper
2. Drawing
3. Paper feed
4. Paper
5. Drawing

To Change Plotting Pen Attributes

You can change the attributes and assign your own line font, thickness, or color to a pen using the `table.pnt` file. Any attribute that you specify overwrites the default instructions.

The format for an entry in the `table.pnt` file is as follows:

```
pen # pattern values units; thickness value units; color values;
<color_name>;
```

Note: For each pen definition, the pen number (`pen #`) must be the first entry and the color name (`color_name`) the last entry. Other attributes can be in any order.

- **pattern**—Specifies plotted line font definition drawn according to defined values in the given units. Values are created in the following order: first line segment length, first space length, second line segment length, second space length, and so on. For example: `pen 3 pattern .1 .05 .025.05.`
- **thickness**—Specifies plotted line thickness in the given units.
- **color**—Specifies color used for plotting; defines color using proportions of red, green, and blue on a scale 0 to 1. Works only on color plotters.
- **<color_name>**—Corresponds to the default Pro/ENGINEER color that the system assigns to a particular entity type (to access the default system colors, click View

> Display Settings > System Colors, and then, in the **System Colors** dialog box, click **Scheme > Default**.

Note: When changing attributes for pen plotting, consider the following:

- All units must be set in inches (IN) or centimeters (CM). Using millimeters (MM) results in a syntax error.
- It is possible to assign more than one color to the same pen in the `table.pnt` file.
- Separate multiple color names by spaces or commas.
- Separate attributes by semicolons.
- Include any or all attributes for each pen.
- Attributes that were not included in the `table.pnt` file appear unchanged, as in normal plotting.

Assigning Colors to Another Pen

The `table.pnt` file enables you to reassign Pro/ENGINEER colors to another pen to overwrite the default mapping. The following table shows the default correspondence between the name of the system color and how it actually appears on the screen.

Note: All colors can be toggled by using the **Use Pre-Wildfire Scheme** tab in the **System Colors** dialog box (**View > Display Settings > System Colors**).

Color Correspondence Table

COLOR_NAME	Visible Color
attention_color	dark green
letter_color	yellow
highlite_color	dark red
drawing_color	white
half_tone_color	gray
magenta_color	purple
edge_highlite_color	green
dimmed_color	dark gray
section_color	blue
presel_highlight_color	cyan
datum_color	brown

COLOR_NAME	Visible Color
quilt_color	magenta
selected_color	red
secondary_selected_color	orange
preview_geom_color	yellow
secondary_preview_color	pale yellow

If you change a default color using the **System Colors** dialog box, the Pro/ENGINEER maintains the `color_name` that corresponds to entities that had their color changed. However, if you reassign an entity to another line style, this changes the associated `color_name` and Pro/ENGINEER plots the entity as defined for that color name.

Note: Pro/ENGINEER plots entities by their assigned colors. User-defined colors always map to Pen 1. If Pen 1 has a default pattern, thickness, and so on, the system reflects this default in the plot.

Example: table.pnt File

Note: For each pen definition, the pen number (`pen #`) must be the first entry and the color name (`color_name`) the last entry. Other attributes can be in any order.

The following is a sample `table.pnt` file:

```
pen 1 color 0.0 0.0 0.7; highlight_color
pen 2 thickness .5 cm; letter_color
pen 5 pattern 1.0 0.1 0.5 .01 in; color 1.0 0.0 1.0; drawing_color
```

When creating a `table.pnt` file, you can:

- Use units in inches (in) or centimeters (cm), and mix them in font definitions; for example, you can specify a font pattern in inches and thickness in centimeters.
- Use the backslash (\) at the end of a line to continue the entry on the next line.
- Use an exclamation point (!) at the beginning of a line in the file to make it a comment.

The following is another example of a `table.pnt` file:

```
!Exclamation points denote comment lines in the file
!
!Change yellow entities to plot w/ pen 1
pen 1 thickness 0.1 cm; letter_color
!
Change hidden lines to plot w/ pen 2
```

```

pen 2 pattern 0.1, 0.1 in; thickness 0.1 cm; half_tone_color
!change geometry lines to pen 3
pen 3 drawing_color
!
Green sheetmetal lines to pen 5
pen 5 thickness 0.1 in; attention_color
    
```

Specifying a Default Pen Mapping Table

Pro/ENGINEER uses the default pen mapping table file unless another pen mapping file exists and Pro/ENGINEER is configured to use it. The default pen table file location is <loadpoint>/text/table.pnt. To use other pen tables, you must specify them as full path names in a plotter configuration file (.pcf) or in the config.pro file, or they must be in the current directory.

To specify the default pen mapping, remove any reference to pen table files from the config.pro and plotter configuration files (.pcf), and remove any file named table.pnt from all working directories.

When you specify a Pro/ENGINEER entity for plotting, the system assigns it a pen based on the default system color corresponding to this entity type. When it plots entities, it uses the line width and line font associated with a particular color. For example, the system uses pen #1 to plot all entities that appear in Pro/ENGINEER using the Geometry index (white by default) and have the same line width and thickness.

If your plotter supports eight plotter pens, and you want to use them for plotting, set the configuration file option use_8_plotter_pens to yes. The following table illustrates the default pen mapping that occurs. A plotter that supports four pens uses the first four pens listed in the table.

Pen Number	System Color	Mapping
1	Geometry (white) Curve (dark blue) Manufacturing Volume (purple) Quilt (magenta) Datum (brown) Preselection Highlight (cyan) Preview Geometry (yellow) Secondary Preview Geometry (pale	Visible geometry (plot as solid lines, except where noted): Cross-section cutting plane lines: plot as phantom lines Cross-section cutting plane arrows and text Drawing format and boundary Tag text Centerline line font with white color Brown portion of datum planes

Pen Number	System Color	Mapping
	yellow)	
2	Letter (yellow)	All items plot as solid lines (except where noted): Dimension lines Leaders Axes and centerlines: plot as centerlines Geometric tolerance lines All text (except cross-section text) Balloon notes Cross-hatching Centerline line font with yellow color
3	Hidden Line (gray)	Hidden lines: plot as dashed lines, phantom font
4	Highlight-Primary (dark red) Selected (red) Secondary Selected (orange)	All items plot as solid lines: Spline surface grid (does not plot in drawings)
5	Sheet Metal (dark green)	Sheet metal color entities
6	Sketched Curve (blue)	Sketcher section entities
7	Highlight-Secondary (dark gray)	Toggled sections, grayed dimensions and text, dimmed tangent edges

Pen Number	System Color	Mapping
		Dark gray portion of datum planes
8	Highlight-Edge (green)	Spline surface grid

Plotter Configuration Files

You can save or modify a `.pcf` file with the name of the text in the **Destination** box in the **Print** dialog box or create from a file.

When Pro/ENGINEER is installed, plotter configuration file options are available in addition to those that you define within the plotter configuration file directory.

You can also use a plotter configuration file (`filename.pcf`) in interactive and batch plotting to preset some or all of the plotting options. Pro/ENGINEER displays the name of your `.pcf` file without the extension in the list available from the button to the right of the **Destination** box in the **Print** dialog box. If your `.pcf` file has the option `button_name` defined, this string appears in the list.

When creating plotter configuration files, keep in mind the following requirements:

- You can maintain a plotter configuration file locally, or place it in another location. If you place it somewhere else, you must indicate its location by setting the `config.pro` configuration file option `pro_plot_config_dir`.
- You must enter the contents and the name of the plotter configuration file in lowercase characters.
- The plotter option is mandatory.
- All numeric arguments for the plotter configuration options are mandatory.

The following sample shows a plotter configuration file:

<code>plotter</code>	<code>postscript</code>
<code>plotter_command</code>	<code>plot</code>
<code>plot_names</code>	<code>yes</code>
<code>rotate_plotting</code>	<code>no</code>
<code>paper_size</code>	<code>e</code>
<code>plot_segmented</code>	<code>yes</code>
<code>plot_scale</code>	<code>model 2</code>

In the preceding example, the `.pcf` file instructs the system to print any size drawing to an E-size Postscript printer. Notice that the configuration file is set up to generate a segmented plot.

Plotter Configuration Options

The following is a list of plotter configuration options. The default values are in *italic*.

Note: These options are not the same as Pro/ENGINEER configuration file options.

`allow_file_naming`

yes, no

yes—Enters the name of the plot file.

no—Uses the name of the object as the default name for the plot file.

`button_name`

string

Adds the specified name to the selection list. Otherwise, the name of the plotter configuration file is used.

`button_help`

string

Provides one-line help for an item specified by the `button_name` option.

`create_separate_files`

yes, no

Defines the default in the **Print to File** dialog box.

yes—Creates separate files by default.

no—Creates a single file by default.

`delete_after_plotting`

yes, no

yes—Deletes the plot file from the directory after it plots successfully.

`interface_quality`

0, 1, 2, 3

Defines the amount of checking for overlapping lines in a plot or 2D export file, such as IGES, before making a file.

0—Collects lines of the same pen color but does not check for overlapping lines.

1—Collects lines of the same pen color for plotting but does not check for overlapping lines.

2—Collects lines of the same pen color for plotting and partially checks edges with two vertices.

3—Completely checks all edges against each other, regardless of the number of vertices, font, or color. Collects lines of the same pen color for plotting.

`paper_size`

paper size, <variable> X Y units

Specifies particular paper size to which to plot (for example, A1). You can specify a custom size by entering a variable, followed by the paper dimensions and units.

`paper_size_allowed`

List of possible paper sizes

Specifies all of the paper sizes (separated by spaces) you can use for your plotter model, for example, a b c. This limits sizes in the Size list to only those in your list.

`pen_slew`

`no_slew`, value

Sets the plotter pen speed along both the x-axis and the y-axis, where the value can be from 0.1 to 100.

`pen_table_file`

path and file name

Specifies a default pen mapping table that supersedes other pen mapping information. Paths can contain up to 260 characters.

`plot_area`

yes x1 y1 x2 y2, no

Sets the Plot Area button in the Plot list of the **Shaded Image Configuration** dialog box as the default.

`plot_access`

`create`, append, default

Creates a new plot file, or appends the new data to another plot file.

`plot_clip`

yes x1 y1 x2 y2, no, default

yes—Plots a portion of the drawing. The x- and y-coordinates are mandatory. They are normalized values from 0.0 to 1.0 (0,0 corresponds to the lower-left corner of the plot) that indicate which portion of the window to plot. For example, to clip half of the window in the center of the screen, set `plot_clip yes 0.25 0.25 0.75 0.75`.

`plot_destination`

file, printer, and file_and_printer

Directs plotting to write to a file, plot to a printer, or to write and plot. You can also specify this destination in the **Print** dialog box (**File > Print**) by clicking or clearing the **To file** and **To printer** check boxes.

`plot_drawing_format`

yes, no, default

no—Generates a plot without format.

`plot_file_dir`

directory name

Specifies the directory to which plot files are written. Use the full path name, for example, `/home/users/plotfiles`. Paths can contain up to 260 characters.

plot_label

yes, no, default

yes—Generates the plot with a label.

plot_label_height

<value> in, <value> cm

Sets the height of the plot labels. Enter a numeric value for this option, followed by the units in inches (in) or centimeters (cm).

plot_layer

current, by_layer #, default

by_layer—Specifies a layer of the drawing to plot where # is the layer ID number.

plot_linestyle_scale

any positive number, 1.0

Specifies the scaling factor for the hidden line font in your plot. Enter any positive number, such as 2.0, as a value for this option.

plot_names

yes, no

yes—Adds descriptive extensions to plot files when it creates them. The extensions are:

- hp—Hewlett-Packard plotters
- hp2—Hewlett-Packard hpgl2 plotters
- cal—Calcomp plotters
- ver—Versatec plotters
- ger—Gerber photoplotters
- ps—PostScript plotters (including color)

no—Adds the extension `.plt` to all plot files.

Note: Pro/BATCH does not support this option.

plot_roll_media

yes, no

yes—Uses roll paper for the media type.

no—Uses the default setting of the particular printer for media type.

plot_scale

fit_paper, model #, plot #, default

Specifies a scale factor for scaling a model or drawing for plotting, where # is the scale value from 0.01 to 100.0.

plot_segmented

yes, no, default

yes—Generates a segmented plot.

plot_sheets

current, range <low value> <high value>, default, all

Specifies sheets to plot. To indicate a range of sheets, set this option to range, followed by the low and high values for the range, such as `plot_sheets range 2 3`.

plot_translate

<value>, <x value> <y value>, <x value> <y value> <units>

Sets the offset of the plot from its origin. Enter only the x-offset value or both the x-offset and y-offset values. You can only specify units (such as mm) when you have given the offset in both directions. For example:

```
plot_translate .25
```

or

```
plot_translate .25 .35 for offsetting 0.25 in the x direction and 0.35 in the y direction.
```

Note: When a single value is given, it is used for both x and y.

plot_to_scale_full_window

yes, no

Scales the output plot file to the full size of the screen window.

plot_with_panzoom

yes, no

yes—Bases plot on zoom.

no—Plots full plot.

plotter_command

command, windows_print_manager <plotter or printer name>

Specifies the command for to use when sending a plot to the printer.

- In UNIX, specify a plotter command as:

```
plotter_command lp -d <plotter_name>
```

or

```
plotter_command lpr -P <plotter_name>
```

- In Windows specify the plotter command as:

```
plotter_command windows_print_manager <plotter or printer name>.
```

plotter_handshake

hardware, software

Sets the type of plotter handshake initialization sequence, generated in the plotter files. For each mode, you must set the data port of the host computer appropriately:

- Software handshake mode is Xon/Xoff.
- Hardware handshake mode requires hardware control.

Note: Consult your system administrator for more information on handshaking.

plotter

<plotter name>

Sets the default plotter when making plot files. This option is mandatory.

rotate_plotting

yes, no

yes—Rotates a PostScript plot 90 counterclockwise when plotting (portrait). The horizontal on the screen corresponds to the short edge of the paper.

no—Does not rotate a PostScript plot. Plotting of parts and assemblies assumes a landscape format. The horizontal on the screen corresponds to the long edge of the paper.

Note: This does not affect shaded PostScript output.

Plotting Views Directly

About Directly Plotting 3D Views

While using the **Standard** application (Part or Assembly), use the **Quick Print** command (**File > Quick Print**) to directly plot views for the active model, without manually creating a drawing.

By specifying one of the following layouts and a printer configuration file, you can create the plot:

- **Manual Layout**—Specifies the number of views per sheet, named views and a scale for each named view, and a display style for each view.
- **Projection Layout**—Selects the projection layout based on a named view.
- **Drawing Template**—Uses a template to define all aspects of the layout, except for the printer configuration.

The plot reflects the layer settings, simplified representation state, explode state, and zone clipping for the current model, and appears as if a drawing were created

and plotted. The file used to create the plot is automatically erased from your Pro/ENGINEER session.

Note:

- In Part, **Quick Print** is not available for simplified representations.
- In Assembly, **Quick Print** is available for geometry and graphics representations but not for symbolic representations.

Configuring for Direct Plotting

Use the following configuration options when you set up for direct plotting:

- **quick_print_plotter_config_file**—Specifies the path for the default plotter configuration file (.pcf) when using the **Quick Print** command.

If `plotter` is set to `ms_print_mgr`, no other options are necessary. If it isn't set to `ms_print_mgr`, adding to the plotter configuration file the values for `plot_file_dir` and `plotter_command` is recommended.

If you do not set `quick_print_plotter_config_file`, it defaults to `<Pro/E_loadpoint>/text/plot_config/postscript.pcf` on UNIX and `<Pro/E_loadpoint>/text/plot_config/ms_print_mgr.pcf` on Windows.

Note: The .pcf defined in the **Quick Print** dialog box overrides the configuration option setting.

- **quick_print_drawing_template**—Specifies the path for the default drawing template file when using the **Quick Print** command with **Drawing Template** selected as your **View Layout** definition.

Some settings in the **Quick Print** dialog box may conflict with settings in the plotter configuration file. For example, plotter configuration file settings for `plot_access` and `allow_file_naming` may be overwritten by settings in **Quick Print**.

Some settings in the plotter configuration file are ignored by **Quick Print** because they do not apply to **Quick Print**. Settings in the plotter configuration file that are ignored by **Quick Print** include `paper_size` and `paper_size_allowed`. Plotter configuration options that work with **Quick Print** include `plot_label`, `plot_sheets`, `plot_file_dir`, and `plot_segmented`.

To Directly Plot a 3D View

To use a default plotter configuration or drawing template (when **Drawing Template** is selected for a view layout), set the configuration options `quick_print_plotter_config_file` and `quick_print_drawing_template` before you use the **Quick Print** command. Setting this configuration options eliminates the need to type a path or browse to these files from the **Quick Print** dialog box.

1. Open a part or assembly and click **File > Quick Print**. The **Quick Print** dialog box opens.

Note: The **Quick Print** command is available while using the **Standard** (Part and Assembly) application only.

2. Click a **View Layout Definition**.
 - **Projection Layout**—Selects the projection layout based on a named main view.
 - **Manual Layout**—Specifies the number of views, view layout, view scale, and view display style.
 - **Drawing Template**—Specifies a drawing template so you do not need to specify view name, view scale, and display style for each view.
3. Depending on your **View Layout Definition** selection, click the appropriate procedure below:
 - To Plot Using a Projection
 - To Plot Using a Manual Layout
 - To Plot Using a Drawing

To Plot Using a Projection Layout View

After you click **File > Quick Print**, the **Quick Print** dialog box opens. Follow the procedure below when you select **Projection Layout** as your **View Layout Definition**.

1. In the **Plot Config File** box, type the path to your plotter configuration `.pcf` file or click **Browse** and select the file.

Note: If you have set the configuration option `quick_print_plotter_config_file`, the path to the plotter configuration file is automatically defined. You can override the configuration option by typing a path to or browsing to another `.pcf` file.
2. Click **Portrait** or **Landscape** to specify the orientation of the sheet, and click the **Standard Sizes** box to select the size of the sheet.
3. To use the same view scale and display style for all views, click the **Use View Scale for all views** and **Use Display Style for all views** check boxes. Note the **View Scale** and **Display Style** defaults for the **Main View** are listed in the **Views** area of the dialog box.

Note: To assign unique view scales and display styles, clear these check boxes.
4. Click the **Saved View Name** box for the **Main** view and select another name, or keep the default **Standard Orientation** name.
5. Click in the **View Scale** and **Display Style** boxes to select a new values or style, or accept the defaults for the **Main** view.
6. In the **View Layout** area click the projection views you want. Projection views determine view layout and positioning.

Note: Projection views are the unlabeled boxes in the **View Layout** area.

7. Click a general view if needed. Each general view is listed in the **Views** area of the dialog box.
 - If you selected **Use View Scale for all views** or **Use Display Style for all views**, the values for **View Scale** and **Display Style** initially default to the values for **Main**. If you change a value for **View Scale** or **Display Value** for the **Main View** or any general view, then that value changes for all views listed.
 - If you cleared **Use View Scale for all views** and **Use Display Style for all views** check boxes, then you can make changes for **Main** or any general views, without affecting other values.
8. Click **OK** to plot the part or assembly.

Note:

- If the plotter configuration file cannot be found, is missing required values, or is generating plotter errors, then an error dialog box opens after you click **OK**. Depending on the error, you can redirect to the correct plotter configuration file or edit the plotter configuration file from the dialog box.
- If you have defined a model grid, and you are plotting a **Project Layout View** or **Manual Layout View**, the model grid is automatically plotted for all views with the correct orientation. For information on defining a model grid, see Detailed Drawings help.

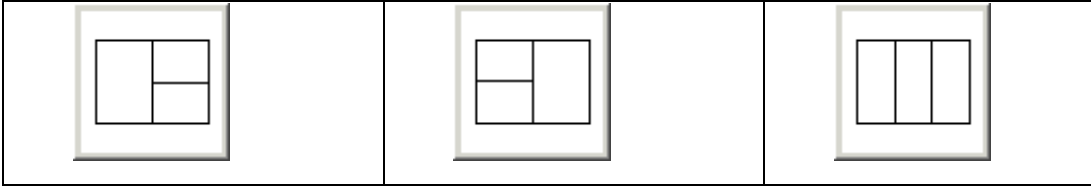
To Plot Using a Manual Layout View

After you click **File > Quick Print**, the **Quick Print** dialog box opens. Follow the procedure below when you select **Manual Layout** as your **View Layout Definition**.

1. In the **Plot Config File** box, type the path to your plotter configuration .pcf file or click **Browse** and select the file.

Note: If you have set the configuration option `quick_print_plotter_config_file`, the path to the plotter configuration file is automatically defined. You can override the configuration option by typing a path to or browsing to another .pcf file.

2. Click **Portrait** or **Landscape** to specify the orientation of the sheet and click the **Standard Sizes** box to select the size of the sheet.
3. In the **Number of Views** box, select the number of views (maximum 4).
 - Available configurations for the number of views you select appear in the **View Layout** area.
 - An entry for each view appears in the **Views** area.
 - If you select more than one view, the **Use View Scale for all views** and **Use Display Style for all views** check boxes become available.
4. Click the **View Layout** that you want. For example, you can select from the following layouts when you select 3 views.



5. If you select to have more than one view, and you want to use the same view scale and display style for all views, click the **Use View Scale for all views** and **Use Display Style for all views** check boxes.

Note: To assign unique view scales and display styles, clear these check boxes.

6. For each view listed in the **Views** area:

- Click the **Saved View Name** box and select another name, or keep the default **Standard Orientation** name.
- Click the **View Scale** and **Display Style** boxes to select a new values or style, or accept the defaults.

If you selected **Use View Scale** for all views or **Use Display Style for all views** and you change a value for **View Scale** or **Display Style** for view, then that value changes for all views listed.

If you cleared **Use View Scale for all views** and **Use Display Style for all views** check boxes, then you can make changes for a view, without affecting values for other listed views..

7. Click **OK** to plot the part or assembly.

Note:

- If the plotter configuration file cannot be found, is missing required values, or is generating plotter errors, then an error dialog box opens after you click **OK**. Depending on the error, you can redirect to the correct plotter configuration file or edit the plotter configuration file, from the dialog box.
- If you have defined a model grid, and you are plotting a **Project Layout View** or **Manual Layout View**, the model grid is automatically plotted for all views with the correct orientation. For information on defining a model grid, see Detailed Drawings help.

To Plot Using a Drawing Template View

After you click **File > Quick Print**, the **Quick Print** dialog box opens. Follow the procedure below when you select **Drawing Template** as your **View Layout Definition**.

1. In the **Drawing Template** box, type the path to your drawing template (.dwt) file or click **Browse** and select the file.

Note:

- If you have set the configuration option `quick_print_drawing_template` this is automatically defined for you. You can overwrite the configuration option by typing a path to or browsing to another `.drw` file.
 - For more information on Drawing Template, see About Drawing Templates in Detailed Drawings.
2. In the **Plot Config File** box, type the path to your plotter configuration (`.pcf`) file or click **Browse** and select the file.

Note: If you have set the configuration option `quick_print_plotter_config_file`, the path to the plotter configuration file is automatically defined for you. You can overwrite the configuration option by typing a path to or browsing to another `.pcf` file.

3. Click **OK** to plot the part or assembly.

Note:

- The **Quick Print** command plots only what is on the screen. Settings in the drawing template that are in conflict with the current model are ignored.
- If the plotter configuration file cannot be found, is missing required values, or is generating plotter errors, then an error dialog box opens after you click **OK**. Depending on the error, you can redirect to the correct plotter configuration file or edit the plotter configuration file, from the dialog box.

Exiting Pro/ENGINEER

To Exit Pro/ENGINEER

1. Click **File > Exit**. A **Confirmation** message appears.
2. Click **Yes**.

Note:

- Set the configuration option `allow_confirm_window` to `no` to prevent the **CONFIRMATION** dialog box from opening.
- Set the configuration option `prompt_on_exit` to `yes`. so you are prompted to save your files before you exit Pro/ENGINEER.

The Edit Menu

About the Edit Menu

Use the commands on the **Edit** menu to perform a wide variety of editing operations. Depending on the application in which you are working, different commands may become available. For example, some commands are not available in the **Standard** part and assembly modes but are available in the **Standard** drawing mode. You can

also access many of the same editing commands from shortcut menus that are available when selecting items in the model tree or in the graphics window.

Edit menu commands in the **Standard** part mode include the following:

- Pro/ENGINEER editing features, **Mirror**, **Fill**, **Intersect**, **Merge**, **Pattern**, **Project**, **Wrap**, **Trim**, **Extend**, **Offset**, **Thicken**, and **Solidify**
- **Regenerate**—Regenerates the model.
- **Undo**—Removes the last operation in the current session and restores the model to the existing state before that operation.
- **Redo**—Redoes the last operation you performed.
- **Copy**—Copies the selected object to the clipboard.
- **Paste**—Pastes the object from the clipboard
- **Paste Special**—Pastes the object from the clipboard dependently.
- **Flip Normal**—Flips the normal of a surface or quilt.
- **Suppress**—Suppresses the selected features.
- **Resume**—Resumes the selected suppressed features.
- **Delete**—Deletes the selected features,
- **Properties**—Opens the **Dimension Properties** dialog box after you select one or more dimensions.
- **Setup**—Activates the **PART SETUP** menu from which you can define items such as material, units, accuracy, and mass properties.
- **References**—Activates the **REROUTE REFS** menu from which you can Reroute the references of the selected item.
- **Definition**—Opens the dashboard so that you can redefine the selected feature.
- **Pattern Table**—Opens the **Tables** dialog box from which you can modify the table pattern
- **Scale Model**—Prompts you to enter a scale for the dimension of the active model.
- **Feature Operations**—Activates the **FEAT** menu, from which you can access commands to manipulate features.
- **Select > Preferences**—Opens the **Selection Preferences** dialog box from which you to set the selection behavior preferences for selectable items in the Graphics window. You can also turn preselection highlighting on or off.
- **Select > Deselect All**—Deselects all selected items.
- **Find**—Opens the **Search Tool** dialog box from which you can search for, filter and select model items.
- **Hyperlink**—Adds, removes, or edits a hyperlink to the selected text.

- **Read Only**—Activates the **R-ONLY FEAT** menu from which you can designate a feature as read only.

About Undoing and Redoing Actions

The **Undo** and **Redo** commands are available for many actions in Pro/ENGINEER. When working in a feature tool, you can undo and redo actions performed during the feature definition session. After you create a feature, save changes, and close the tool, you can undo the feature as one item. For example, when working in the Pro/ENGINEER graphics window, you can undo dimensions as many times as you change them. Specific **Undo** and **Redo** commands appear in the **Edit** menu and as tool tips, so you can confirm the desired action. The `general_undo_stack_limit` configuration option controls the number of times you can undo or redo actions. The default and maximum is 50.

Note: The undo and redo functionality requires additional memory allocation and may result in some change in performance.



What Actions Support Undo and Redo?

The undo and redo functionality is supported for actions that you perform while working in Standard part, Sheetmetal, Assembly, Welding, Sketcher, and Drawing modes.

Undo and redo actions are session- and window-based. The actions are sequentially stacked in memory. You can undo and redo most of the operations performed on a model. These include key feature operations such as:

- Creating and deleting
- Redefining (excluding redefinition of inheritance or features from a copy by reference)
- Suppressing and resuming
- Creating and deleting patterns
- Grouping and ungrouping
- Rerouting (excluding replace reference)
- Reordering
- Editing
- Creating and changing relations and parameters
- Family tables, visible or invisible instances of a family table in a session
- Data Sharing Features
- Interface features and Facet features
- Simplified Representations (You can undo operations performed in simplified representation but not the operations of creating or modifying simplified representations).

How to Use the Undo and Redo Commands

- Click  to undo or  to redo.
- Use the **Undo** and **Redo** commands on the **Edit** menu.
- Press CTRL+Z to undo or CTRL+Y to redo.

Undo and Redo Command Considerations

For actions that cannot be undone, one of the following occurs:

- You can undo prior actions for commands such as **Edit > Regenerate**. For example, you cannot undo **Edit > Regenerate**, but you can undo actions prior to **Edit > Regenerate** until the stack is not cleared.
- Some commands clear the Undo/Redo stack, in which case you cannot perform an undo operation. For example, clicking **File > Erase > Current > Yes** clears the stack.
- You cannot undo actions prior to commands such as **Insert > Independent Geometry**. However, you can undo the **Insert > Independent Geometry** command.
- When you activate a part in assembly mode and modify the part, you can undo only those changes that are made to the part. However, you cannot undo the action of activating the part.

Note:



- Within Sketcher, the `sketcher_undo_stack_limit` configuration option controls the number of times you can undo or redo operations. The default is 200 and the maximum is 500.
- In the Style tool, you can perform any number of undo or redo operations.
- When fixing a model, you can only use **Undo Changes** on the **Fix Model** menu. General undo and redo functionality is not available. The Undo/Redo stack is cleared when you fix the model using **Fix Model > Undo Changes**.

To Select Locations in the Graphics Window Using the Keyboard




If you do not want to or are not able to use the mouse, you can use the keyboard to select objects and specify locations in the graphics window.

1. Set the configuration option `kbd_selection` to `yes`.

Note: Click **Tools > Options** to set a configuration option.

2. Click **Edit > Select > By Keyboard**. The **Select Location** dialog box opens and  (crosshairs) is displayed in the Graphics window. Use  to select objects and specify locations in the graphics window. To navigate through the **Select Location** dialog box press TAB or SHIFT+TAB.

Note: The color of the crosshairs varies.

3. In the **Horizontal Position** box, type the relative horizontal location of where you want  in the Graphics window. The crosshairs move to the specified horizontal position.
4. **In the Vertical Position box**, type the relative vertical location of where you want  in the Graphics window. The crosshairs move to the specified vertical position.
5. Repeat Steps 2 and 3 until  is in the position you want or select one of the position boxes and use the UP ARROW or the DOWN ARROW to change the value. The crosshairs move, changing position.
6. To change the size of the box surrounding the crosshairs, type a new value between 1 and 500 in the **Selection Size** box. The default value is 7. The value corresponds to the distance between the center of the crosshairs and the sides of the box.

Note: To change the default **Selection Size** value, use the configuration option `pick_aperture_radius`.

7. Do one of the following:
 - Click **Select** for left-click actions. All objects within the box surrounding the crosshairs are selected.

Note: When working in a sketch or a drawing, only the entity closest to the crosshair intersection is selected.
 - Click **Middle** for middle-click actions.
 - Click **Menu** for right-click actions.

The View Menu

About the View Menu

The **View** menu provides options for controlling model and performance display. It includes options for setting model orientation, selecting the **View Manager**, model setup such as lighting and perspective, and setting system and entity colors. Using **View > Visibility > Unhide All**, you can show all objects in the main window.

About Orient Mode

Orient Mode (**View > Orientation > Orient Mode**) gives you more viewing capability than the standard spin, pan, zoom. With **Orient Mode** enabled, you can reorient the view with respect to specific geometry and change the style of view reorientation such as **Dynamic**, **Anchored**, **Delayed**, **Velocity**, or **Fly Through**.

Orientation Center

The Orientation Center is displayed by a graphical object that, when clicked with the middle-mouse button, reorients the model in various ways. As you spin, pan, or

zoom, the Orientation Center is visible. The Orientation Center is locked at the Spin Center but can be set anywhere in the graphics window when the Spin Center is disabled.

Spin and Pan Constraint






You can constrain spin and pan with respect to a derived vector when the **Spin Center** is disabled. The vector is derived by placing the Orientation Center on an edge or surface, combined with a drag action; or, if it is already on an edge or a surface, by initiating a drag action on the Orientation Center.

The constraining vector is derived from the object or geometry beneath the Orientation Center, and is either linear to an edge or curve, or normal to a solid face or surface.

Note: You cannot select items when Orient Mode is enabled.

View Type Menu

When Orient Mode is enabled, you can select the following viewing styles from the **View Type** menu (**View > Orientation > Orient Type**):

- **Dynamic** (default)—The Orientation Center is displayed as . Orientation is updated as the pointer moves. The model spins freely about the Orientation Center.
- **Anchored**—The Orientation Center is displayed as . The orientation is updated as the pointer moves. Model rotation is controlled by the direction and distance the pointer is moved from its initial position. The Orientation Center changes color at each 90-degree interval. When the pointer returns to the original mouse-down position, the view is reset to where it started.
- **Delayed**—The Orientation Center is displayed as . The orientation is not updated as the pointer moves. When the middle mouse button is released, the pointer model orientation updates.
- **Velocity**—Orientation Center is displayed as . The orientation is updated as the pointer moves. The velocity (speed and direction), is the rate of manipulation and is affected by the distance the pointer is moved from its initial position.
- **Fly Through**—The orientation and position of the model is controlled with flight simulator-like interactions. This option is only available if you are using Perspective display (**View > Model Setup > Perspective**). When the **Perspective** dialog box is opened, the viewer perspective is displayed by .

Note: You can set the viewing style to dynamic or anchored using the `orientation_style` configuration option. This is irrespective of whether the Orient mode is enabled or disabled, that is, you are outside Orient mode or in the Orient mode, respectively. When Orient mode is enabled, you may change the viewing style as required.

Orient Mode Shortcuts

You can right-click in the graphics area to open the shortcut menu:

- **Show/Hide Spin Center**
- **Dynamic**
- **Anchored**
- **Delayed**
- **Velocity**
- **Fly Through**
- **Exit Orient Mode**

About View Operations


Using the **View** menu, you can adjust the model view, orient the view, hide and show entities, create and use advanced views, and set various model display options. The following list includes some of the View operations you can perform.

- Orient the model view in the following ways, using the **Orientation (View > Orientation > Reorient)** dialog box:
 - Set the Orientation Center
 - Spin, pan, zoom, and turn models and drawings
 - Spin and pan about edges
 - Display the default orientation
 - Revert to the previously displayed orientation
 - Change the position or size of the model view
 - Change the orientation (including changing the view angle in a drawing)
 - Create new orientations
- Temporarily shade a model by using cosmetic shading
- Show, dim, or remove hidden lines
- Explode or Unexplode an assembly view
- Repaint the Pro/ENGINEER graphics window
- Refit the model to the Pro/ENGINEER window after zooming in or out on the model
- Update drawings of model geometry
- Hide and show entities, and hide or show items during spin or animation
- Use advanced views

- Add perspective to the model view

About Activating the Orient Mode

You can activate the Orient Mode by using one of the following methods:

- Click .
- Click **View > Orientation > Orient Mode**.
- Press **SHIFT+CTRL**+click the middle mouse button while the pointer is in the graphics window.

After the Orient Mode is enabled, **View Type** is enabled in the **Orientation** menu, and the Orientation Center and model center display in the graphics window.

About Disabling the Orientation Center

You can disable the orientation center by clearing the **Orientation center** check box in the **Model Display** dialog box that opens when you click **View > Display Settings > Model Display**. To disable the orientation center during spin, pan, or zoom operations, set the `spin_with_orientation_center` configuration option to `no` (default is `yes`).

You cannot disable the orientation center when you are in the Orient mode.

Adjusting the Model View

About Dynamic Viewing

Pro/ENGINEER dynamically displays the model as you reorient your view. For example, as you spin, pan, zoom or turn an item, you see it change as you move the mouse.

View Behavior

Spin, Pan, Zoom and Turn are all achieved by clicking the middle mouse button, and except for Spin, in combination with either the SHIFT or CTRL key.

Three-button Mouse

Spin	Press and hold the middle mouse button + move the mouse
Pan	Press and hold the middle mouse button + SHIFT + move the mouse
Zoom	Press and hold the middle mouse button + CTRL + move the mouse vertically
Turn	Press and hold the middle mouse button + CTRL +

	move mouse horizontally
Wheel Mouse	
Zoom	Place pointer over focused geometry area + spin mouse wheel.

About Handles

Handles are graphical objects that are used to manipulate geometry during creation or redefinition in realtime. Using your mouse, drag the handles to resize, reorient, or move feature geometry. Snap handles to existing geometric references or to user-defined grid increments. Your changes display dynamically in the graphics window.

Handle display changes at specified intervals to acknowledge your actions, and are used in conjunction with the left mouse button, **SHIFT** and **CTRL**.

To Cosmetically Shade a Model

Use this procedure to apply temporary shading to an image without affecting the saved view of the model, and without canceling the view choice for the current session. The shading is removed when you repaint the view.

Do one of the following:


- Click **View > Shade**
- Press ALT+V then S

Note: Do not use this procedure to shade the model view permanently (so that it remains shaded even when reoriented). Instead, click the **Shading** button.

To Display the Default Orientation

When you first create a model, it appears in a default view orientation, such as isometric or trimetric. When you open a model, it is retrieved in the orientation in which it was last saved. You can change the default orientation by using the **Orientation** dialog box (**View > Orientation > Reorient**) or the configuration option `orientation`.

To display the model in the standard orientation, do one of the following:

- Click **View > Orientation > Standard Orientation**
- Click . The Saved Views list opens.
- Click **Default**. The model reorients to the default view.

To Refit the Model to the Window


Use this procedure to refit the model to the screen so that you can view the entire model. A refitted model fills 80 percent of the screen.

Do one of the following:

- Click **View > Orientation > Refit**.
- Click the **Refit** button  on the Pro/ENGINEER toolbar.

To Revert to the Previously Displayed Orientation


Do one of the following at any time to resume the previously displayed view:

- Click **View > Orientation > Previous**.
- From the toolbar, click . (Use **Customize Screen** from the **Tools** menu to place buttons on the toolbar.)

To Repaint the Window

You can repaint a view to remove all temporarily displayed information. Repainting redraws the screen but does not regenerate the model.

Do one of the following:

- Click **View > Repaint**
- Click 
- Press ALT + V then R
- Press CTRL + R

To Disable Highlighting of Selected Model Tree Items

Pro/ENGINEER highlights all selected Model Tree items in the graphics window by default. If you do not want to highlight selected Model Tree items, you can disable this highlighting by clicking **Show > Highlight Model** from the Model Tree. The check mark clears to indicate that **Highlight Model** has been disabled. All items that you select in the Model Tree will not be highlighted in the graphics window.

To Update a Drawing of Model Geometry

To update a drawing after making changes to its source model, regenerate the view. Regenerating a drawing view is not object-action oriented; you must choose the command first, and then select the view.

Do one of the following:

- In Drawing mode, click **View > Update > Drawing View**.
- Select one or more drawing views.
- Set the configuration file option `auto_regen_views` to `yes`.

Orienting the Model View

About Orienting the Model View

A model appears in a default view orientation when it is first created and any time it is retrieved (provided it was saved in default orientation). You can change the default orientation of your model or create new orientations using the **Orientation** dialog box (**View > Orientation > Reorient**). Using this box, you can do the following:

- Orient the view dynamically by using the slider bars to spin, pan, zoom, and turn the model. You can also use the mouse in the graphics area to spin, pan, zoom, and turn the model.
- Orient the view by reference (default setting).
- Set a different spin center in the model. The default spin center is the model center. You can set the screen center, a specified point or vertex, a specified edge or axis, or a coordinate axis as the new spin center.

By default, the Spin Center symbol is present on the model at all times. This can be set using the configuration file option `spin_center_display`. The symbol is similar to that for a Coordinate System, except for the following:

- The three axes of the spin center are, by default, in three different colors—x-axis red, y-axis green and z-axis cyan, whereas all three axes of a coordinate system are yellow.
- A small solid ball is positioned at the tip of each axis of the spin center.

As you spin the model, the spin center rotates synchronously with the movement of the mouse. In addition, the balls at the ends of the axes grow larger as they move towards you (out from the screen) and smaller as they move away from you (into the screen). You can determine the relative orientation of the model at any time by inspecting the axes of the spin center.

- Choose a saved view orientation.
- Fit the model to the screen.
- Select a new screen center.
- Create a user-defined view orientation by specifying the reference axes and angles to rotate the model around each axis.

You can also use the **Orientation** dialog box to orient a draft view. In this case, the dialog box provides some different options for orienting a selected draft view.

About Saved Views

Saving a view saves the orientation of the view relative to the screen and any pan and zoom orientation associated with the view.

When you save views, the display is based on the default orientation of the model. If you change the default orientation of the model (for example, from isometric to trimetric), the orientation of saved views changes along with it.

If they are recalled with some of their view orientation references missing, saved views retain their orientation but do not retain the history of how that orientation was achieved.

Note: Views created within instances of a generic model cannot be saved; only views created for the generic can be saved.

Orientation Dialog Box

A model appears in a default view orientation when it is first created and any time it is retrieved. You can change the default orientation of your model or create new orientations using the **Orientation** dialog box.

In Drawing mode, you can also change the orientation of draft views by using the **Orientation** dialog box.

To open the **Orientation** dialog box, do one of the following:

- Click **View > Orientation > Reorient**. The **Orientation** dialog box opens with **Orient by Reference** selected under **Type**.
- In Drawing mode, right-click on the drawing view. Select **Reorient**. The **Orientation** dialog box opens. If the selected view contains child views, you will be prompted to continue. Click **Yes** to continue.

The following orientation types are available:

- Angles (available only for Drawings)—Lets you orient a drawing view by specifying an angle reference (Normal, Vertical, Horizontal, or Edge/Axis) and an angle value. All angles that you specify are listed in the Reference area.
- Dynamic Orient—Lets you orient the view dynamically by using the settings for panning, zooming, and spinning. Available for 3-D models only.
- Orient by Reference—In Part, Assembly, or Drawing mode, lets you select references by which to orient the model or drawing view. In Part or Assembly mode, you can select multiple surfaces as orientation references. The options are Front, Back, Top, Bottom, Left, Right, Vertical Axis, and Horizontal Axis.
- Preferences—This area contains different choices depending on whether you are in Part or Assembly mode, or in Drawing mode.
 - It lets you define the spin center for 3D models (Part and Assembly modes).
 - It lets you change the default orientation for parts, assemblies and drawings. The default view is **Trimetric**. You can set the view orientation to **Isometric** or **User Defined**. In a user-defined orientation, you specify angles to rotate the model around the X and Y axes.
 - It lets you set the view origin for a specified drawing view as either the default origin or a user-defined (custom) origin. To do so, you select a reference object within the specified drawing view.
- Saved Views—In Part, Assembly, or Drawing mode, lets you name or save a view, or retrieve a previously saved view.

- Undo—In Part, Assembly, or Drawing mode, lets you undo the changes you made in the **Orientation** dialog box and returns you to the most current view state in the model or drawing.

Tip: Orienting a View by Reference

You can orient your model by selecting multiple surfaces to use as references in orientation. These references can be Front or Back (parallel to the screen), Top or Bottom (normal to the screen facing up or down), and Left or Right (normal to the screen and facing left or right). The model automatically reorients itself after the second surface has been selected.

If two different parallel surfaces are selected before a normal surface is selected, the second selection overrides the first. You can also orient your model by selecting a single horizontal or vertical axis for orientation.

Tip: Shortcut for Display of Saved Views








To display a model or drawing in one of the saved view orientations, do one of the following:

- Click **View > View Manager**. This invokes the **View Manager** dialog box. Select a view from the list, then click **Set**. In the graphics window, the model orients to the selected saved view.
- Click the **Saved View List** toolbar button. A small selection box opens, listing all saved views. Select the name of the desired saved view. The model reorients to the selected view.

Note: Views created for instances of a generic model cannot be saved; only the views created for the generic can be saved.

Tip: Shortcuts for Changing the Model View

You can use the following toolbar buttons to adjust the model view:

- —Zooms in
- —Zooms out
- —Refits the object to the window
- —Spins the object around the spin center
- —Enters Viewing mode
- —Displays the saved view list
- —Reorients the view

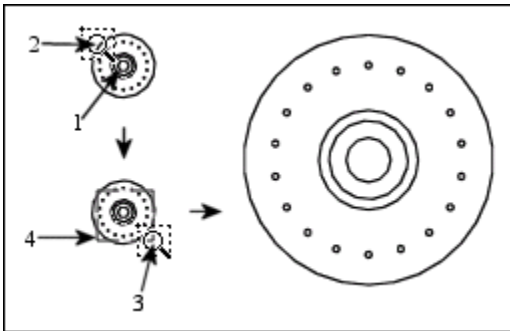
You can also spin, pan, zoom, and turn with the mouse, by doing the following:

- **Spin**—Hold down the middle mouse button and drag the mouse.
- **Pan**—Hold down **SHIFT**, the middle mouse button, and drag the mouse.
- **Zoom**—Hold down **CTRL**, the middle mouse button, and drag the mouse vertically.
- **Turn**—Hold down **CTRL**, the middle mouse button, and drag the mouse horizontally.

About Using Zoom on the Model

You can zoom in on a part or assembly to define the individual features that are necessary for the part or assembly that you are creating. You zoom in on target geometry by framing the target geometry in a zoom box. The following illustrates how to zoom in using the zoom box:

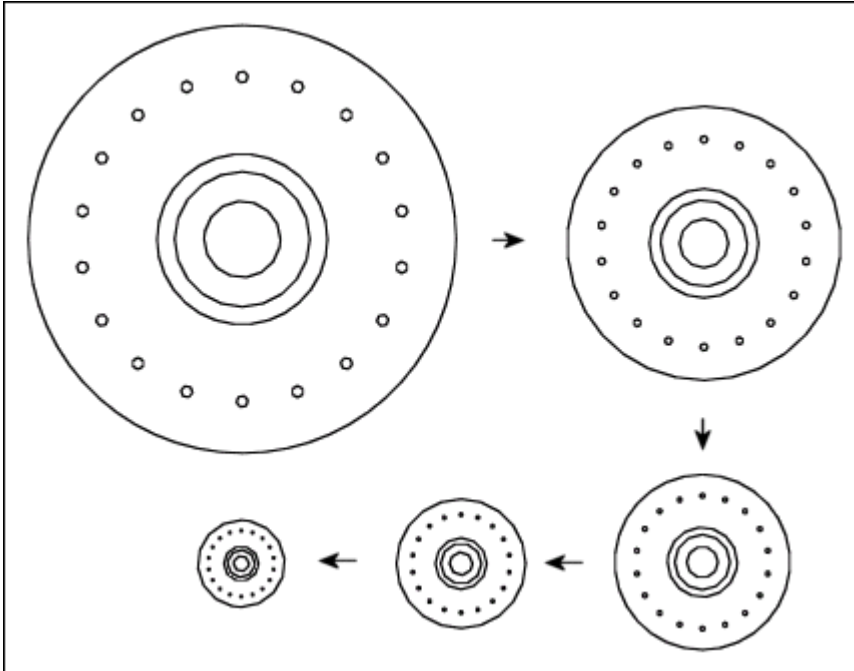
Zoom in





- 1 Target geometry
- 2 Zoom box start point
- 3 Zoom box end point
- 4 Zoom box

You can zoom out on a part of assembly to gain a wider perspective of the geometrical context. This perspective helps you better understand how the individual features that you created for the part fit into the overall model design. The following illustrates various zoom out states:


Zoom out





To Zoom on the Model View

1. If you want to zoom out on the model, skip to step 6.
2. To zoom in on the model, place the pointer above and to the left or right of the target geometry in the graphics window. This area is the zoom box start point.
3. Press CTRL and middle-click. The pointer changes to . Note that you hold the CTRL key but release the middle mouse button.
4. Drag the mouse diagonally over the target geometry and center the geometry in the zoom box. As you drag, the zoom box is created.
5. Middle-click to define the zoom box end point. The end point is diagonally opposite of the start point. Pro/ENGINEER immediately zooms in on the target geometry. Note that to cancel zoom in, simply release the CTRL key.
6. If you want to zoom out on the model, click  or use the scroll wheel on your mouse.

Tip:

- You can also click  to create the zoom box.
- You can manually zoom in or zoom out on target geometry by using the scroll wheel on the mouse. If you do not have a scroll wheel, place your pointer over the target geometry. Then press and hold the CTRL key and the middle mouse button, and drag up and down (left and right spins the model).









- If you want to manually zoom in or zoom out using the **Zoom** slider or by typing a zoom value, click . On the **Orientation** dialog box, select **Dynamic orient** from the **Type** box and use the **Zoom** slider. Note that negative values zoom out.
- If you need to refit the model to the screen, simply click .

To Change the Position or Size of the Model View

You can adjust the angle of spin, size, or centering of an object in the active window, using the **View** menu and various options in the **Orientation** dialog box.

1. Click **View > Orientation > Reorient**. The **Orientation** dialog box opens with the **Type** set to **Orient by Reference**.

You may do any of the following to change size or position of your model view:

- From the **Type** box, select **Dynamic Orient** to adjust Pan, Zoom and Spin settings.
 - Click **Refit** to redisplay the entire model in the window.
 - To select a new centerpoint, click **Center**, then select a new centerpoint in the graphics window. The model adjusts its position to the new centerpoint.
 - Under **Options > Spin**, do one of the following:
 - Click the **Spin Center** button () to spin the object around its spin center. Move the x (), y (), or z () sliders, enter adjustment values, or click the up or down arrows to adjust the spin around the X, Y, or Z axes.
 - Click the **Screen Center** button () to spin the object around the screen horizontal, screen vertical, or normal-to-screen axis. Move the **H** (), **V** (), or **C** () sliders, enter adjustment values, or click the up or down arrows to adjust the spin around the horizontal (X), vertical (Y), or center (Z) axes.
 - Click the **Dynamic update** check box to enable or disable viewing the view change as you make adjustments.
2. Click **OK** to accept the orientation changes and close the **Orientation** dialog box, **Undo** to undo your orientation changes and start over, or **Cancel** to close the dialog box without saving the changes.

To Change the View Angle in a Drawing

When you add a new general view in a drawing, the **Drawing View** dialog box opens automatically. To change the angles in a drawing view with respect to the normal, the horizontal, the vertical, or a specified edge or axis:

1. Select a general view to reorient.

Note: Reorienting a view reorients its children views accordingly.

2. Right-click on the selected view and click **Properties**. The **Drawing View** dialog box opens.
 - By default, the **View Type** category is selected.
 - The name of the selected view appears in the **View name** box.
 - The type of view appears as `General` in the **Type** box.
3. Click **Angles** under **View Orientation**. When asked for confirmation, click **Yes**.

Note: Modifying the view orientation also affects the orientation of the highlighted children views. The children views may be on different sheets.

4. Select the type of reference that you want to specify from the **Rotation reference** list.
 - **Normal**—Rotates the model around an axis through the view origin and normal to the drawing sheet.
 - **Vertical**—Rotates the model around an axis through the view origin and vertical to the drawing sheet.
 - **Horizontal**—Rotates the model around an axis through the view origin and horizontal to the drawing sheet.
 - **Edge/Axis**—Rotates the model around an axis through the view origin and according to the designated angle to the drawing sheet.

Note: If you have selected **Edge/Axis**, the **SELECT** dialog box opens. Select an appropriate edge or axis reference on the previewed drawing view. The selected reference is highlighted and is listed in the box adjacent to the **Rotation reference** box.

5. In the **Angle value** box, type the angular value by which you want to offset the orientation.
6. If required, specify multiple references by using the **Add** and **Remove** buttons.
7. Click **Apply** to display the model in its default view.

To Name, Retrieve, Save, or Delete a View

1. Click the **Reorient View** toolbar button. The **Orientation** dialog box opens.
2. Click the **Saved Views** bar. The **Saved Views** dialog box opens.
3. Select a view or type a name for the current view.

4. Select one of the following:

Save—Type a name for the current view in the **Name** box (the name can be up to 80 characters long), and click **Close** to save the current view.


Set—Select a name from the list and click **Set** (or double-click the name) to display the selected view.

Delete—Select a name from the list and click **Delete** (or double-click the name) to delete the selected view. The current view remains unchanged when a saved view is deleted.

Close—Click to close the **Saved Views** dialog box.

Note: To view the list of saved views, click the **Saved view list** button on the Pro/ENGINEER toolbar.

To Save a View Using the View Manager

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Orient**.
3. Click **New**. A default name appears for your view. Change the view name or press ENTER to accept the default name.
4. Right-click the new view name and click **Redefine** on the shortcut menu. The **Orientation** dialog box opens.
5. Orient the model by references or position the model to the desired view using the spin, pan, and zoom commands.

Note :

- The active orientation is marked with a red arrow.
 - A plus sign after the view name of the active orientation indicates that the orientation has changed.
6. Click **Edit > Save** in the **View Manager** or click **Save** on the shortcut menu that opens when you right-click the view name. The **Save Display Elements** dialog box opens.

Note: You cannot save the changes to the Standard orientation.

7. Select the name of the view from the list or type a new name.
8. Click **OK** in the **Save Display Elements** dialog box.

You can use the **Remove**, **Rename**, or **Copy** commands on the shortcut menu that appears when you right-click a view to remove, rename, or copy a view, respectively.

To Set the Spin Center for an Object

The default orientation for an object is set to trimetric but can be modified to be isometric or user-defined. The default spin center of a model is the model center. Use the following procedure to modify it to be any point on the screen.

Note: You cannot set the spin center for an object in the Drawing mode.

1. Click **View > Orientation > Reorient**.
2. From the **Type** list in the **Orientation** dialog box, choose **Preferences**.
3. From the **Spin Center** option buttons in the **Options** section, select one of the following:

Model center

Screen center

Point or vertex—Select a point or vertex on the object.

Edge or axis—Select an edge or axis on the object.

Coordinate system—Select a coordinate system on the object.

4. Under **Default Orientation**, select the type of default orientation you want to create (**Isometric**, **Trimetric**, or **User Defined**), and set the angular position of the user-defined default orientation in the X and Y directions.
5. To name, set, save, or delete the view, click the arrow triangle to the left of the **Saved Views** section, and use one of the following:
 - Save**—Type a name for the current view in the **Name** text box and click **Save** to save the current view.
 - Set**—Select a name from the list and click **Set** to display the selected view.
 - Delete**—Select a name from the list and click **Delete** to delete the selected view.
6. Click **OK** to accept the orientation changes and remove the dialog box, **Undo** to undo your orientation changes and start over, or **Cancel** to abort spin center changes and remove the dialog box.

You can also set orientation using the configuration file option `orientation`.

Model Display Options

About Setting Model Display Options

The flexibility in setting the model display makes it easy for you to visualize your model. For example, you can specify whether to show or hide items or set model appearance options (wireframe, hidden lines, shaded, displayed, or removed). Click **View > Display Settings > Model Display** to open the **Model Display** dialog box.

Use the **Display Style** area in the **General** tab to:

- Change the default display style **Shading** to Wireframe, or specify whether and how hidden lines are displayed.
- Toggle the model color display on or off.
- Enable Trace Sketch display.
- Change the display of dimension tolerances for the current Pro/ENGINEER session. The dimension tolerances are displayed on the bottom right side of the graphics window.
- Show or hide reference designators.
- Display cable portions that are inside cabling components.
- Show or hide datums, surface mesh, silhouette edges, or the Orientation Center while reorienting or animating using **Display while reorienting**.
- Enable animation by time and frames while reorienting using **Animation while reorienting**.

Use the **Edge/Line** tab to:





- Change the quality and detail of edges and lines.
- Set the display style of cables to thick or centerline using **Cable Display**.

Use the **Shade** tab to:

- Change the quality and detail of shaded areas.
- Set textures, transparency (stippled or blended), and realtime rendering using **Enable**.
- Set reflection and shadows using **Realtime Rendering**.

Setting Model Display Options

You can specify how you want your model to appear on the screen. The default display method is Shading, which allows for efficient visual comprehension of the model. Hidden lines are shown as light grey lines in Wireframe display. To set model display style, you can use the **Model Display** dialog box (**View > Display Settings > Model Display**), or you can use the following toolbar buttons.

	Wireframe —Hidden lines are visible as regular, white lines.
	Hidden Line —Hidden lines are displayed in grey.
	No Hidden Line —Hidden lines are removed.
	Shading —Default setting. Shades the model. Hidden lines are not visible. The shading adapts to changes in model orientation. You can use this display method to view the contours of a surface or to print a copy of the model.


To Show Cables as Thick or Centerline

1. Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.
2. Under **Display** on the **General** tabbed page, select or clear:
 - **Reference designators**—To change the display of reference designators of cabling connectors and ECAD components.
 - **Internal cable portions**—To change the display of cable portions that are inside cabling components.
3. Under **Cable Display** on the **Edge/Line** tabbed page, select or clear:
 - **Thick**—To display cables with 3-D thickness to show shading.
 - **Centerline**—To display the cable centerline with location points shown in green.
4. Click **OK** to save the changes and remove the dialog box, or **Apply** to save the changes and leave the dialog box displayed.

To Show or Hide Items During Spinning or Animation

To change the colors, spinning, or animation display of your model:

1. Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.
2. In the **General** tab, under **Display**, select **Colors** to display colors. This can also be set using the configuration file option `color`.



Note: You can also use the **Colors on/off** button () on the toolbar to switch the display of colors. Use **Customize Screen** from the **Tools** menu to place buttons on the toolbar.

3. Under **Display while reorienting**, select:
 - **Datums**—To display datum features during the dynamic spinning of the model. This checkbox is cleared by default. Note that you can also set the configuration option `spin_with_part_entities` to `yes` (default is `no`).
 - **Surface mesh**—To display surface mesh during the dynamic spinning of the model. This checkbox is checked by default.
 - **Silhouette edges**—To display silhouette edges during dynamic spinning of the model. This checkbox is cleared by default. Note that you can also set the configuration option `spin_with_silhouettes` to `yes` (default is `no`).
 - **Orientation center**—To display the Orientation Center during dynamic spinning of the model. This checkbox is checked by default. Note that you can also set the configuration option `spin_with_orientation_center` set to `yes` (default is `yes`).
4. Under **Animation while reorienting**, select:

- **Enable**—Enable functionality.
- **Maximum seconds**—Set the maximum number of seconds spent animating between views.
- **Minimum frames**—Set the minimum number of displays for animation between views.

To Switch Between Shaded and Wireframe Display

Click one of the following toolbar buttons:



-  **Wireframe** — Hidden lines are displayed as regular lines.
-  **Shading** — The model is shaded according to the view orientation. Hidden lines are not visible in shaded view display.

or

1. Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.
2. In the **General** tab, under **Display Style**, select **Wireframe** or **Shading**.
3. Click **Apply** to save the change and leave the dialog box open, or click **OK** to save the change and close the dialog box.

To Show or Remove Hidden Lines

Click one of the following toolbar buttons:

-  **Hidden Line** — Hidden lines are displayed in gray. The toolbar button shows dotted lines for clarity.
-  **No Hidden** — Hidden lines are removed.

or

1. Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.
2. Click the **General** tab. Under **Display Style**, select **Hidden Line** or **No Hidden Line**.
3. Click **Apply** to save the change and leave the dialog box open, or click **OK** to save the change and close the dialog box.

Tip: When Hidden Line Removal is Not Correct

Hidden line display may not always work as you expect. Keep the following in mind:

- Assembly components that physically intersect may not have correct display on one or both components. If this occurs it is an indication of incorrect component placement.
- Curves and cosmetic features that intersect a part in the middle of a surface, i.e., where half the curve is visible and half not, might have incorrect display for the whole line. The display is correct if, in the current view, the curve crosses an edge at the surface boundary.

To Change the Appearance of Edges and Lines

1. Click **View > Display Settings > Model Display**.
2. Click the **Edge/Line** tab of the **Model Display** dialog box.
3. Under **Edge Quality**, select the quality of display you want (**Medium**, **Low**, **High**, or **Very High**). This can also be set using the configuration file option `edge_display_quality`.
4. Under **Tangent Edges**, select the style for edges where two surfaces are tangent (**Solid**, **No Display**, **Phantom**, **Centerline**, or **Dimmed**). This can also be set using the configuration file option `tangent_edge_display`.

Note: You can also use the tangent edge style toolbar buttons





to change tangent edges display (use **Tools > Customize Screen** to place buttons on the toolbar).

5. Under **Options**, select the display options from the following:

- **Silhouette edges**—To display silhouette edges.

Note: You can select only the following types of silhouette edges: Silhouettes of cylinders and cones, and silhouettes of torii with axes that are parallel or perpendicular to the screen.

- **Smooth lines**—To display all nonvertical and nonhorizontal lines using anti-aliasing techniques.
 - **Depthcue always**—To display depthcue when the **Visibilities** dialog box is closed. This can also be set using the configuration file option `depthcue_options`.
 - **Clip always**—To display clipping when the **Visibilities** dialog box is closed.
6. Under **Cable Display**, select from the following:
 - **Thick**—To display clipping when the **Visibilities** dialog box is closed.
 - **Centerline**—To display the cable centerline with location points shown in green.

Note: You can also use the **Cable thick style**  and **Cable centerline style**  toolbar buttons to change the cable display. Use **Utilities > Customize Screen** to place buttons on the toolbar.


- Under **Cable HLR**, select **Full**, **None**, or **Partial** to set the level at which hidden lines are removed in a cabling assembly. This can also be set using the configuration file option `full_hlr_for_cables`.

To Change the Display of Tangent Edges

Use this procedure to change the line style for edges created by two tangent surfaces.

- Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.
- In the **Tangent Edges** list on the **Edge/Line** tabbed page, select one of the following styles:
 - **Solid**
 - **No Display**
 - **Phantom**
 - **Centerline**
 - **Dimmed**
- Click **Apply** to save the change and leave the dialog box open, or **OK** to save the change and close the dialog box.

Tip: Shortcuts for Setting the Display of Tangent Edges

- You can also use the tangent edge style toolbar buttons  to change tangent edges display (use **Tools > Customize Screen** to place buttons on the toolbar).
- You can also set the tangent edge display using the configuration file option `tangent_edge_display`.

To Control Shading Quality

The quality of a shaded model is determined by the size of triangles used for shading on revolved and spline surfaces. The higher the quality, the smoother the curved surfaces appear when shaded. Planar surfaces always appear the same, so they are not affected by the shade quality.

To change the quality and appearance of a shaded image:

- Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.

2. Click the **Shade** tab.
3. Under **Quality**, click the up or down arrows to adjust the value to the number you want. The range is 0 to 10. The default value is 3. A setting of 10 takes the longest to generate.
4. Under **Shade**, select any of the following shading features:
 - **Surface features**—To use shade surface features when shading the model. This can also be set using the configuration file option `shade_surface_feat`.
 - **With datum curves**—To display datum curves on the shaded model. This can also be set using the configuration file option `shade_with`.
 - **With edges**—To display edges darker than the adjacent surfaces. This can also be set using the configuration file option `show_shaded_edges`.
 - **Small surfaces**—To shade very small surfaces.
 - **Manufacturing reference model**—To shade manufacturing reference models when shading the model. This can also be set using the configuration file option `shade_mfg_ref_models`.
5. Under **Enable**, select any of the following:
 - **Textures**—To enable the display of textures on shaded models.
 - **Transparency**—To enable transparency for a model in the current window. There are two **Transparency** options:
 - Blended**—Default setting offers high-quality imaging with compromised performance in complex models.
 - Stippled**—Offers lower-quality imaging with faster performance in complex models.
 - **Capped Clipping**—To cap the plane that clips the shaded parts of a model.
6. Click **OK**.

To Hide or Unhide Datum Entities

To turn the display of datum features on or off, do one of the following:

- Click **View > Visibility > Hide** or **View > Visibility > Unhide**, or click its corresponding toolbar button. The **GET SELECT** menu appears. Use the Model Tree or the Graphics window to select the datum entities that you want to hide, and then click **Done Sel**. The selected datums are hidden.
- Select a datum entity or entities that you want to hide, and then click **View > Hide** or **View > Unhide**.

Note: In Pro/ENGINEER, the **Display Curve Names** button is not visible by default in any Pro/ENGINEER application except for Pro/SHIP. Use **Tools > Customize Screen** to place this button on the Pro/ENGINEER toolbar.

- Click **View > Display Settings > Datum Display**. The **Datum Display** dialog box opens.
 - Under **Display**, select the datum entities and entity names that you want to display. Some entities are selected by default, and you may want to clear the check boxes for the entities and entity names that you want to hide.
 - Use the **Select All** or **Unselect All** buttons to display or hide all datum entities at once.
 - Under **Point Symbol**, use the list box to select the type of symbol you want to use to designate a point. The options are **Cross**, **Dot**, **Circle**, **Triangle**, and **Square**.

Tip: Datum Display Configuration Options

You can change the datum and datum tag display by using any of the following the configuration file options:

- `display_planes`
- `display_plane_tags`
- `display_axes`
- `display_axis_tags`
- `display_coord_sys`
- `display_coord_sys_tags`
- `display_points`
- `display_point_tags`
- `spin_center_display`
- `curve_tag_display`

Datum Display Dialog Box

You use the **Datum Display** dialog box to specify the display status of individual datum entities independently from their names (tags). Click **View > Display Settings > Datum Display** to open the **Datum Display** dialog box.

The following datum display options are available on the dialog box:

- **Planes**—Displays datum planes and their names. This can also be set using the configuration file option `display_planes`.
- **Plane Tags**—Displays datum plane names. This can also be set using the configuration option `display_plane_tags`.
- **Axes**—Displays datum axes and their names. This can also be set using the configuration file option `display_axes`.

- **Axis Tags**—Displays datum axis names. This can also be set using the configuration option `display_axis_tags`.
- **Point Symbols**—Displays datum points and their names. This can also be set using the configuration file option `display_points`.
- **Point Tags**—Displays datum point names. This can also be set using the configuration file option `display_point_tags`.
- **Coordinate Systems**—Displays coordinate systems and their names. This can also be set using the configuration file option `display_coord_sys`.
- **Coordinate System Tags**—Displays coordinate system names. This can also be set using the configuration option `display_coord_sys_tags`.
- **Spin Center**—Displays the spin center symbol. This can also be set using the configuration file option `spin_center_display`.
- **Curve Tags**—Displays curve names. This can also be set using the configuration option `curve_tag_display`.
- **Section Tags**—Displays the section tags.

Use the **Select All** or **Unselect All** buttons to display or hide all datum entities at once.

Under **Point Symbol**, you can specify the type of symbol you want to use to designate a point. The options are **Cross**, **Dot**, **Circle**, **Triangle**, and **Square**.

To Improve Dynamic Display Quality

You can reduce the amount of calculation that the system performs when dynamically orienting a shaded model by controlling the level of detail at which surfaces are displayed. The result is that the movements of the model appear smoother and better synchronized with the driving force. The effectiveness of using levels of detail (LODs) in a model is dependent on the configuration of your system.

To control system performance:

1. Click **View > Display Settings > Performance**. The **View Performance** dialog box opens.
2. Under **Hidden Line Removal**, choose **Fast HLR** to enable hardware acceleration of dynamic spinning with hidden lines, datums, and axes.
3. Under **Spin Displays per Second**, choose:
 - **Enable**—To attempt to guarantee the time it takes to draw a frame during spinning.
 - **Frames Per Sec**—To set the minimum number of displays per second during spinning.
4. Under **Level of Detail**, choose:

- **Enable**—To use levels of detail in shaded models during dynamic orientation. This can also be set using the configuration file option `lods_enabled`.
- **Show detail %**—To set the maximum level of detail that the system calculates. The range is from 0% through 100%. A setting of 100% (the default) represents no loss of detail. A setting of 0% represents almost a total loss of detail.

To Display Process Components

When working on a process plan, use this procedure to set display characteristics for process displays.

1. Click **View > Model Setup > Process Display**. The **Process Component Display** dialog box opens.
2. Using the list boxes under **Settings**, choose the display states you want for:
 - **Previous Components** (previously assembled components)
 - **Current Components** (components active in the current step)
 - **Unused Components**
3. You can set the **Process Display** or **Selection Display** for previous, current, and unused process components to any of the following:
 - Current Environment
 - Phantom
 - Blank
 - Wireframe

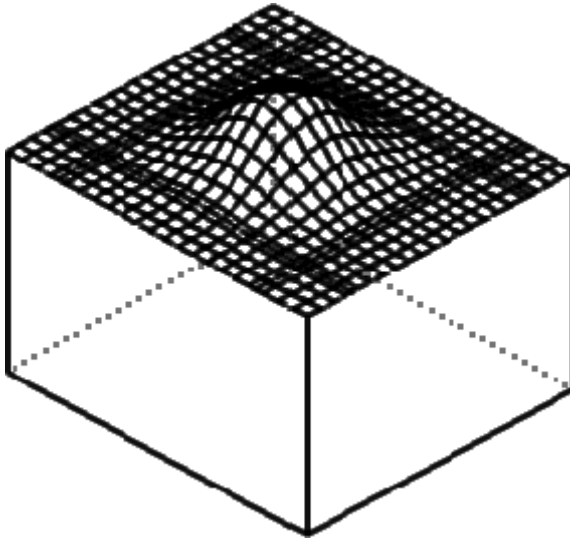
To Mesh a Part Surface

To create a meshed surface in order to assess the validity of part geometry:

1. Click **View > Model Setup > Mesh Surface** or from the toolbar, click the **Mesh Surfaces** button. (Use **Customize Screen** from the **Tools** menu to place the button on the toolbar.) The **Mesh** dialog box opens.
2. Under **Surface**, in the list box, select the type of surface you want to mesh. The options are surface and quilt.
3. Click the arrow button to display the **GET SELECT** menu to select the surface of quilt to mesh.
4. Under **Meshed Spacing**, use **1st Direction** to adjust the spacing between the mesh lines in the first direction and **2nd Direction** to adjust the spacing between the mesh lines in the second direction.

Surfaces remain meshed only until the view is repainted. Hidden lines are not removed when a part is meshed.

Example: Meshed Surface



About Hiding Entities

About Hiding Entities On the Fly

Pro/ENGINEER allows you to hide and unhide selected model entities on the fly any time during the current Pro/ENGINEER session. Using the **Hide** or **Unhide** command, you can save valuable design time.

- You do not have to assign the entities to a layer and then blank the entire layer.
- You can hide and redisplay individual datum features, such as datum planes and datum axes, without having to hide or redisplay all of the datum features at once.
- You can use the Model Tree search capability (**Edit > Find**) to select all items of a specified type (for example, all features of the same type in all components within an assembly) and then hide them using **View > Visibility > Hide**.

The following types of items can be hidden on the fly:

- Individual datum planes (as opposed to hiding or showing all datum planes at once)
- Datum Axes
- Features that contain axes, planes, and coordinate systems
- Analysis features (points and coordinate systems)
- Datum points (whole arrays)
- Coordinate systems
- Datum curves (whole curves, not individual curve segments)
- Quilts (whole quilts, not individual surfaces)

- Assembly components

You can also use the **Hide and Unhide** toolbar buttons, or choose the same commands from the shortcut menu when you right-click an item in the Model Tree or the Graphics area. Use **Tools > Customize Screen** to add these buttons to the toolbar.

Hide—Hides the selected items.

Unhide—Redisplays the selected items.

Unhide All—Redisplays all hidden items that you have hidden. **Unhide All** does not redisplay items that were automatically hidden by Pro/ENGINEER, such as datums created on the fly or sketches for sketched entities.

When you hide an item, Pro/ENGINEER removes the item from the graphics window. The hidden item remains in the Model Tree list, and its icon dims to reveal its hidden status. When you unhide an item, its icon returns to normal display (undimmed) and the item is redisplayed in the Graphics window.

You can save or reset a layer display status as follows:

Save Status—To save the current layer display status, click **View > Visibility > Save Status**.

Note: Saving the object does not save the current layer display status. To save the current layer display status when saving the object, you must use **View > Visibility > Save Status** and then save the object.

Reset Status—If you change the layer display status after you have saved a status, click **View > Visibility > Reset Status** to revert to the last saved layer display status.

To Hide an Entity On the Fly

Note: Hiding operations are object-action oriented only. You must select the item you want to hide or redisplay before you choose **Hide** or **Unhide**.

1. In the Graphics window or the Model Tree, select and then right-click the item you want to hide.
2. On the shortcut menu, click **Hide**. The selected item or items are blanked from the display. Their icons in the Model Tree are dimmed to reveal their hidden status.

To Redisplay Hidden Entities

Note: Hide and redisplay operations are object-action oriented only. You must select the item you want to hide or redisplay before you choose **Hide** or **Unhide**.

1. In the Model Tree, select the hidden items that you want to redisplay. On the shortcut menu, click **Unhide**, or click the **Unhide** toolbar button.

Note: The toolbar button is not present on the toolbar by default; you must add it using **Tools > Customize Screen**.

2. The selected entities are redisplayed in the Graphics window, and their icons are returned to normal display status (not dimmed) in the Model Tree.

To Redisplay All Hidden Entities at Once

Do one of the following:

- Click **View > Unhide All**.
- Click the **Unhide All** toolbar button.
- Right-click the selected item and choose **Unhide All** from the displayed shortcut menu.

All hidden (blanked) entities are redisplayed in the Graphics window.

Using Advanced Views

About Adding Perspective to a View

Perspective creates a single-vanishing-point perspective view of a shaded or wireframe model. These views allow you to observe an object as the view location follows a curve, axis, cable, or edge through or around an object.

To add perspective to a model view, you select a viewing path and then control the viewing position along the path in either direction. You can also rotate the perspective view in any direction, zoom the view in or out, and change the view angle at any point along the path.

About Fly-Through and Walk-Through

The fly-through and walk-through capabilities in Pro/ENGINEER enable you to view model geometry from different perspectives.

Using fly-through or walk-through, you can do the following:

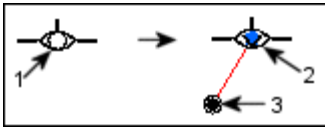
- Move forward or backward, toward or away from the model.
- Spin around so that model is in a different direction (up or down, or to the left or right).
- Combine the preceding these movements to rotate around the model and view it from any perspective.

Your view always moves (forward or backward) in the direction that you determine by using your mouse. Change the direction and the fly-through motion changes with respect to your direction.

Fly-Through vs Walk-Through

Fly-through provides a way to view model geometry in its entirety by using continuous motion. You navigate the model by using the eye target and your mouse. You can fly-through the geometry to view the design as a whole. Seeing the larger context enables you to see how each feature or part fits into the design.

The Eye Target



1. Eye target (not activated)
2. Eye target (activated)
3. Navigation handle

Walk-through provides all of the moving capabilities of fly-through, except that you direct each movement increment using the sliders in the **Perspective** dialog box (**View > Model Setup > Perspective**). This provides greater navigational control enabling you to accurately view targeted geometry.

You can also use both fly-through and walk-through to get a comprehensive view of your design. For example, you could use fly-through to view the model from a certain perspective, and then use walk-through to view smaller parts of the view or to turn corners while traveling around a model.

Perspective Dialog Box

The **Perspective** dialog box (**View > Model Setup > Perspective**), enables you to add perspective to a model view. This dialog box contains the following options:

Under Type:

- **Fly Through**—Activates fly-through. You use your mouse to navigate the model.
- **Walk Through**—Activates walk-through. You use the sliders located under **Options** to navigate the model.
- **From To**—Enables selection of two datum points or vertices that define the viewing path along the object. To use this perspective, select the features from the object using the **SELECT** dialog box.
- **Follow Path**—Enables selection of an axis, edge, curve, or silhouette as the viewing path. For this perspective, select the features from the object using the **SELECT** dialog box.

Note: All perspective view manipulation that is contained in the **Perspective** dialog box can also be done using the functionality available in View Mode and selecting the eye (viewer perspective) icon as the manipulation center.

Under Options:

- **Dynamic Update**—Controls whether or not the display dynamically changes as you adjust the settings in this dialog box.
- **Eye Dist**—Moves the viewpoint along the selected path through the model. If the model is currently shaded and the path lies completely inside the model, nothing is visible on screen unless there are voids in the solid.

- **Rotate Eye**—Sets the amount of display rotation around horizontal, vertical and center axes from -180 to +180 degrees.
- **Pan Eye**—Sets the horizontal and vertical positioning of the display.
- **Zoom**—Sets the percentage of magnification in the display from 0% to 100%.
- **View angle**—Sets the angle of view from 0 to 45 degrees.
- **OK**— Accepts all changes and closes the dialog box
- **Cancel**—Cancels your changes and closes the dialog box
- **Reset**—Restores the settings to their original (default) status and leaves the dialog box open so you can make additional changes

To Add Perspective to the Model View

1. Click **View > Model Setup > Perspective**. The **Perspective** dialog box opens.
2. Under **Type**, select one of the following:
 - **Fly Through**
 - **Walk Through**
 - **From To**
 - **Follow Path**
3. Under **Options**, select or clear the check box; or move sliders, click arrows, or enter values for the following:
 - **Dynamic Update**
 - **Eye Dist**
 - **Rotate Eye**
 - **Pan Eye**
 - **Zoom**
 - **View Angle**
4. For **From/To** or **Follow Path** perspectives, select the items from the object using the **SELECT** dialog box.
5. Click **OK** accept the settings or **Cancel** to remove the settings you made. Click **Reset** to restore the settings to their original (default) status.

Change Depth Cue, or Clip Percentages

To Change Depth Cue or Clip Percentages

Use the following procedure to change depth cue and clip percentages for the model view:

1. Click **View > Display Settings > Visibilities**. The **Visibilities** dialog box opens.
2. Adjust sliders or enter values for the following options:
 - **Clip %** — To change the placement of the clipping plane (a plane through a shaded model that lets you display only the portion of the model behind the plane). The range allowed is 0% to 100%, where 0% is the front of the model (nearest the screen). (To enable clipping plane display, choose **View > Display Settings > Model Display** and use the **Shade** tabbed page of the **Model Display** dialog box.)
 - **Depth Cue %** — To change the thickness of wireframe lines so that they appear darker as they extend into the screen (away from you) and lighter as they extend out from the screen (toward you). The range allowed is 0% to 100%. At 0%, all lines have maximum brightness; at 100%, all wireframe lines are blanked out. (To enable wireframe display, choose **View > Display Settings > Model Display** and use the **General** tabbed page of the **Model Display** dialog box.)

To Fly Through a Model

1. Click **View > Model Setup > Perspective**. The **Perspective** dialog box opens.
2. Under **Type**, select **Fly Through**.
3. In the graphics window, middle-click the eye target. Fly-through activates.
4. Do one of the following:
 - Click the middle mouse button to move towards the object. To increase the speed, middle-click multiple times or hold the button down. Notice that if you are moving away from the object, clicking only the middle mouse button slows the reverse motion.
 - Press CTRL and middle-click to do the following:

Decrease forward speed. Note that acceleration or deceleration must be re-initiated after reaching zero-velocity.

Move away from the object. To increase the reverse speed, press CTRL and middle-click multiple times or hold the button down. Notice that clicking only the middle mouse button slows the reverse motion.
 - Click to roll the object to the left.
 - Right-click to roll the object to the right.
 - Move the mouse and use the navigation handle to navigate the object . Notice that if the object is a zero velocity, this action repositions it.
5. Double-click the middle mouse button to stop the fly-through.

Note:

- The **Perspective** dialog box must remain open to use fly-through.

- If you use the sliders in the **Perspective** dialog box after you selected **Fly Through**, Pro/ENGINEER changes to walk-through and the eye target closes. Simply, select **Fly Through** from the **Types** box. Fly-through activates and the eye target appears.

To Walk Through a Model

1. Click **View > Model Setup > Perspective**. The **Perspective** dialog box opens. Notice that **Fly Through** is selected by default.
2. Under **Type**, select **Walk Through**.
3. Under **Options**, use any of the following slides or type a value in the box and press ENTER:
 - **Dynamic Update**—This check box controls whether or not the display dynamically changes as you adjust the settings in this dialog box.
 - **Eye Dist**—Moves the viewpoint along the selected path through the model. If the model is currently shaded and the path lies completely inside the model, nothing is visible on screen unless there are voids in the solid.
 - **Rotate Eye**—Sets the amount of display rotation around horizontal, vertical and center axes in degrees.
 - **Pan Eye**—Sets the horizontal and vertical positioning of the display.
 - **Zoom**—Sets the percentage of magnification in the display.
 - **View angle**—Sets the angle of view in degrees.
4. Click **OK**. The **Perspective** dialog box closes and walk-through session end.

Cross Sections

Creating Cross Sections

About Part and Assembly Cross Sections

You can use a cross section to see a cutout of a model along a particular direction. You can also use cross sections with the Info and Measure functions. If the model has changed since the cross section was created, the cross section is updated automatically.

Use the **View Manager** to create the following types of cross sections:

- Standard planar cross sections of parts or assemblies
- Offset cross sections of parts or assemblies
- Cross sections of a faceted model (.stl file)

About Cross Sections and Quilts

With Technical Surfacing, you can create a cross section of a selected quilt or a cross section that intersects all model geometry including quilts. Depending on the geometry you want to intersect, you can create the following types of cross sections:


- **Model** cross sections—You can create cross sections that intersect only the solid geometry in a model.
- **Model & Qlts** cross sections—You can create cross sections that intersect solid geometry and all quilts in a model. The intersection curves are displayed in any three-dimensional view of the model and in any area cross-sectional drawing view. They do not appear in total cross-sectional drawing views.
- **Surf/Quilt** cross sections—You can create cross sections that intersect a single quilt in a model to display its contour. Quilt cross sections can be created in Part and Assembly modes.
- **One Part** cross sections—You can create cross sections that intersect only the selected part in an assembly.

Restrictions for Using Cross Sections

- Planar cross sections can be crosshatched or filled. Offset cross sections can only be cross-hatched.
- Cross-hatching is not visible unless you have Detailed Drawings. If you have Detailed Drawings, the cross-hatching is visible even if you click the **No Hidden** toolbar button.
- Cross sections do not show intersections with cosmetic features that are present in a model.

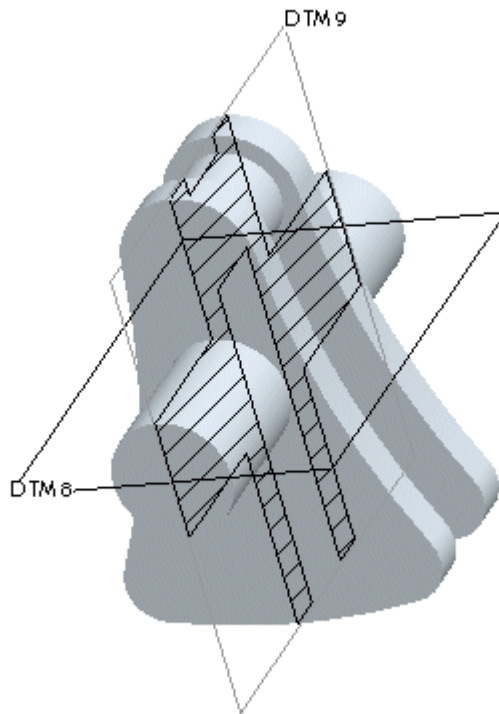
To Create a Planar Cross Section

You make a planar cross section through a model by aligning it with a datum plane or planar surface.

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Click **New**. A default name for the cross section appears.
4. Press **ENTER**. The **XSEC CREATE** menu opens.
5. To create a cross section through a single datum plane, click **Planar > Single > Done**. To create a cross section through all datum planes of a pattern, click **Planar > Pattern > Done**. The **SELECT** dialog box opens.
6. Do one of the following:
 - Select a planar surface or datum plane to intersect the model.
 - If you are creating a cross section through all datum planes of a pattern, select any datum plane that belongs to the pattern.
7. In the **View Manager**, click **Display > Visibility** to view the cross-section.

Example: Planar Cross Section

In this example, a planar cross section is created along the datum plane DTM9.



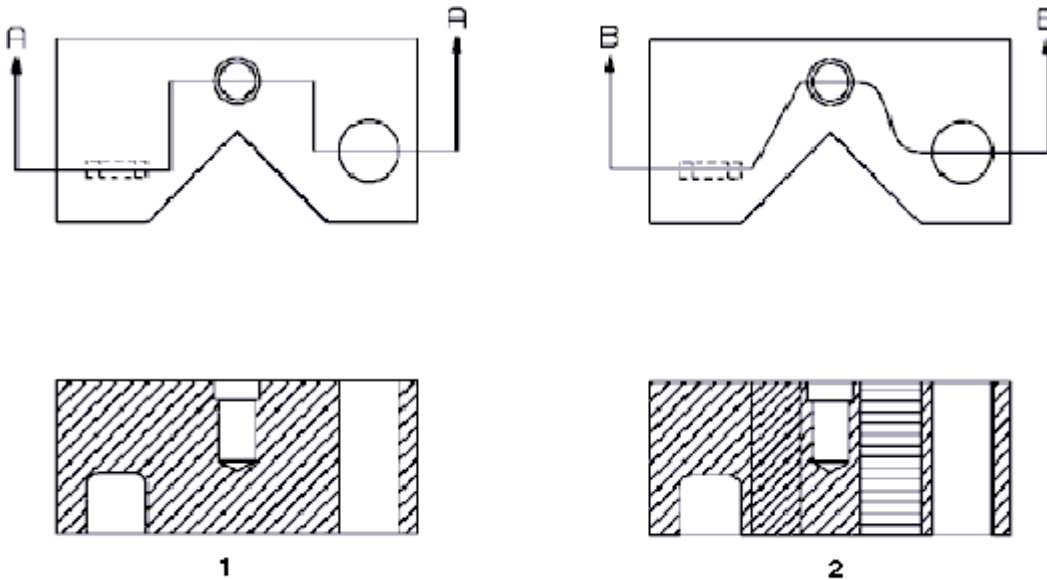
About an Offset Cross Section

You create an offset cross section by extruding a section outline perpendicular to the sketching plane. This type of cross section is valuable for opening up the part to display several features with a single cross section.

The following restrictions apply to an offset cross section:

- The sketched section must be an open section.
- Offset cross sections can be cross-hatched but not filled.
- The first and last segments of the open section must be straight lines. The cutting plane arrows displayed in a drawing are perpendicular to these end segments.
- In drawings, cross section edges always appear wherever the plane of the cross section is not parallel or perpendicular to the screen.
- Circular and spline cross section geometry create non-modifiable horizontal cross-hatching (see the following example).



Offset Cross Sections in a Drawing



1. Proper cross section
2. Improper cross section

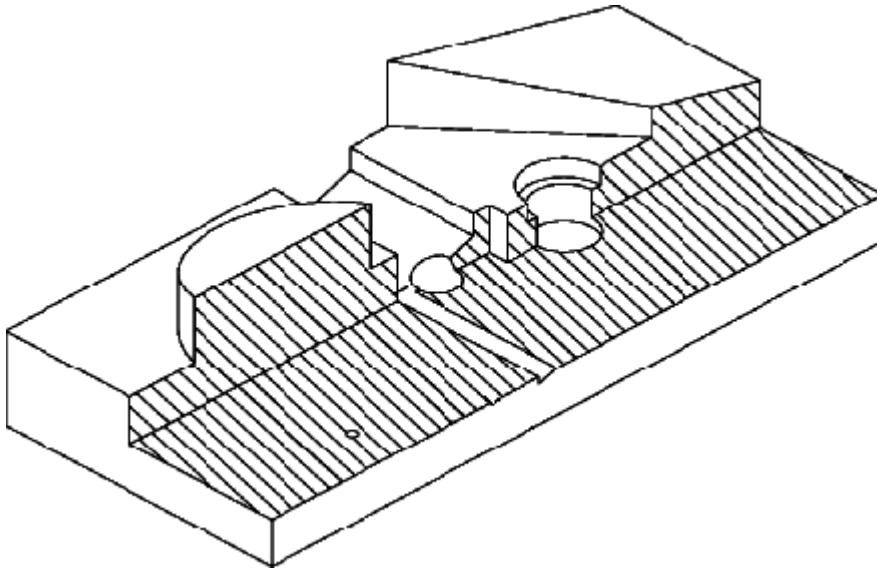
To Create an Offset Cross Section

To create an offset cross section, you must define an outline of the cut by sketching it in Sketcher. The outline is then projected perpendicular to the sketching plane.

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Click **New**. A default name for the cross section appears.
4. Press **ENTER**. The **XSEC CREATE** menu opens.
5. Click **Offset > One Side** or **Both Sides > Done**. The **SELECT** dialog box opens.
6. Select the sketching plane and define the direction of viewing and sketch orientation. You are placed in Sketcher.
7. Select reference entities to use for dimensioning.
8. Sketch the section outline and click  to exit Sketcher.
9. In the **View Manager**, click **Display > Visibility** to view the cross-section.


Example: Offset Cross Section Created to One Side

The following picture shows a cross section created to only one side of a sketching plane.



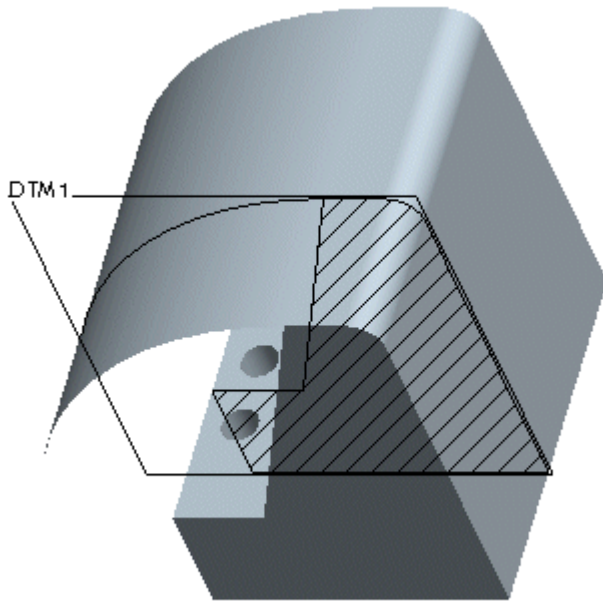
To Create Cross Sections Through Quilts

Use the **Model & Qlts** option to create a cross section that intersects both solid geometry and quilts. Use the **Surf/Quilt** option to intersect a selected quilt.

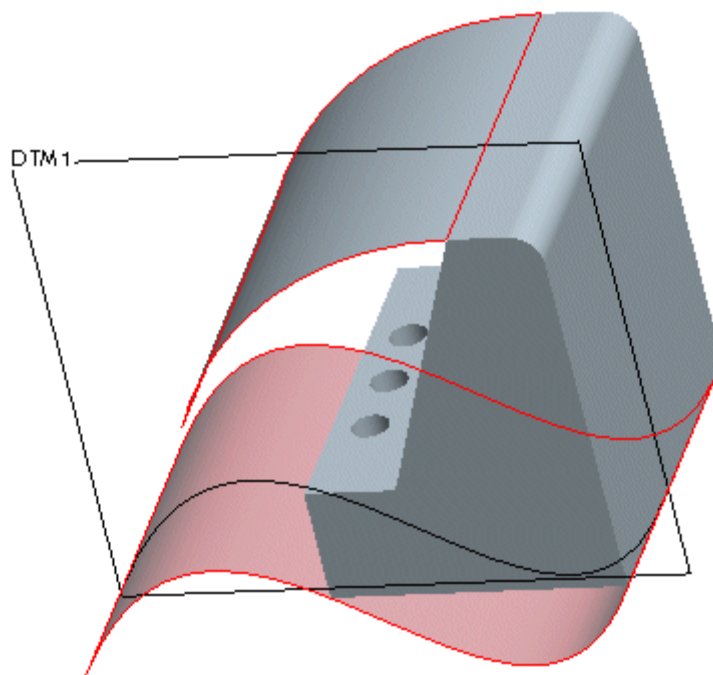
1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Click **New**. A default name for the cross section appears.
4. Press **ENTER**. The **XSEC OPTS** menu opens.
5. Select the type of a quilt cross section you wish to create, **Model & Qlts** or **Surf/Quilt**.
6. Click **Done**. The **SELECT** dialog box opens.
7. If you selected **Surf/Quilt**, select a quilt through which you want to create the cross section.
8. Select the datum plane along which to create a section.
9. In the **View Manager**, click **Display > Visibility** to view the cross section.

Example: Model and Quilts Cross Sections

This example shows a cross section created using the **Model & Qlts** option by intersecting both the solid and the surface geometry with the datum plane **DTM1**.



This example shows a cross section created using the **Surf/Quilt** option by intersecting the quilt with the datum plane *DTM1*. The bottom quilt was selected for sectioning.



About Zonal Cross Sections

Zonal cross sections are 3-dimensional cross sections created by defining zones. The zones contain the same functionality as normal zones. To make large models more

manageable, you can define specific regions called "zones" within a model. You can use zones to:

- Organize a model.
- Control view clipping.
- Select components in an assembly for a simplified representation.
- Create component display states.
- Define envelope parts.

Use the **Xsec** tab in the **View Manager** to create cross sections by defining zones. You can give each cross section a name and store it with the part or assembly. In addition, you can show cross-hatching and view clippings through zone boundaries which remains with the model.

You can define a zone using planar references, for example datum planes or surfaces. These references can come from any level of the part or assembly. You can define the datum planes while you create a zone, or use existing datum planes or surfaces. You can use planar references to define a zone to include everything on one side of the reference. This side is a "half-space" of the datum plane. You can combine any number of half-spaces. However, if you use more than 6 half spaces, view clipping is not available.

You can also define a zone by specifying offset distances from a coordinate system. Additionally, in Assembly, you can define a zone by using closed assembly feature surfaces, or by specifying a distance from an entity.


When you create a zonal cross section, Pro/ENGINEER places it in the Regeneration Footer of the Model Tree. However, if you create datum planes while creating a zonal cross section, then Pro/ENGINEER groups the zonal cross section along with the datum planes. This group is directly placed in the Model Tree as a feature and not in the Regeneration Footer. If you want to move the zonal cross section to the Regeneration Footer, you must first ungroup the zonal cross section and the datum planes that were grouped earlier, and then drag the zonal cross section to the regeneration footer.

Note: You can move a zonal cross section to the Regeneration footer but you cannot move it outside the Regeneration footer.


Pro/ENGINEER includes components in zones as follows:

- If a component lies in more than one zone, Pro/ENGINEER includes it in both zones.
- If a zone intersects a component's bounding box, Pro/ENGINEER includes it in that zone.


To Create a Zonal Cross Section Using Planar References

1. Click  or **View > View Manager** or **View > Zone**. The **View Manager** dialog box opens.


Note: If you have clicked **View > Zone** proceed to step 4.

2. Click the **Xsec** tab.
3. Click **New**. A default cross section name appears in the Names section of the **View Manager** dialog box.
4. Accept the default cross section name or type a new name and then press ENTER. The **XSEC OPTS** menu appears.
5. Click **Zone**. The **Zone** dialog box opens.
6. To select components on one side of a plane using planar references, select **Half-Space** from the list located below the Reference list area. **Half-Space** is the default option.
7. Select a planar reference, for example a datum plane or a surface, for defining the zone. These references can come from any level of the part or assembly.
 - The planar reference name appears in the text box next to the selection arrow.
 - Nine arrows appear in the graphics window, indicating which side of the datum is used to define the zone. You can flip the direction of the arrows using the flip button located below the reference list.
8. To select additional references, click  and select the required reference.

Note:

- You can add any number of planar references. However, for view clipping, define a zone using a maximum of six planar references.
 - In Assembly, if you change the reference type after specifying some references, those references are no longer displayed. If you switch back to the earlier reference type, the references appear again.
 - Click  to remove a reference at any time.
9. If you specify more than one reference, click **AND** or **OR**.

Note: When you select more than one reference, they have logical AND and OR and capabilities. Pro/ENGINEER uses parentheses to maintain an order of operations. You cannot change the position of the parentheses. The **OR** operations are always grouped within parentheses while the **AND** operation separates operations into new parentheses.

10. Click  to preview all the selected references at the same time.

Note: To see a highlighted reference in the Graphics area, select the reference in the References list area. You can select only one reference at a time.

11. Click  to close the **Zone** dialog box.

- The name of the new zonal cross section is displayed in the **View Manager**.
 - The zonal cross section is added to the Regeneration Footer in the Model Tree.
12. To preview defined zones, click **Display** in the **View Manager** and click one of the following:
- **Zone Refs**—To preview zone references
 - **Zone Comps**—To preview zone components
 - **Zone Only**—To preview zone only
13. To show the cross-hatching permanently, in the **View Manager**, click **Display > Visibility** or right-click on the cross section name and select **Visibility** from the shortcut menu.
14. To make the cross section hatching or zone geometry or both visible in the model, select **Xhatching** or **Show Region Boundary** or both and click **OK**.


Working with Cross Sections

About Cross Section Operations

You can perform the following operations on a cross section:

- Display or blank a cross section
- Show a clipped model using a cross section
- Rename a cross section
- Copy a cross section from another model
- Add a description to a cross section
- Remove a cross section
- Include or exclude assembly members from a cross section
- Modify section dimensions for offset cross sections
- Redefine the section attributes, geometry, and dimensioning scheme
- Change a cross section color and cross-hatching
- Switch between filled and hatched cross sections

To Display a Cross Section

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Under **Names**, click the cross section that you want to display.

3. Click **Display > Xhatching**. The cross section hatching is visible in the model. However, upon selection of another cross section the cross-hatching is not visible.
4. To display the cross-hatching permanently, click **Display > Visibility**.

Note: To view zone geometry in a zonal cross section, click **Display > Show Region Boundary**.

About View Clipping

You can perform view clipping using the **View Manager**. You can clip models through cross sections. You can create drawing views from clipped views.


The following are some guidelines for view clipping:

- You can define multiple cross sections. However, you can view clip a model using only one cross section at a time.
- You can use a maximum of six planar boundaries to define zones to view clip them.
- You cannot use cross-section view clipping and model view clipping simultaneously.
- You can save the model in the view clipped state by setting the `save_clipped_view` configuration option to `yes`. On subsequent retrieval of the model, Pro/ENGINEER sets the saved clipping state to active, and the model is displayed in the clipped state.

Note: You need the `save_clipped_view` configuration option only at the time the model is saved. Retrieval of the model depends on how the model was saved; retrieval is completely independent of the configuration option.


To Clip a Model Using a Cross Section

You can display a clipped model using a cross section.

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Right-click the name of the cross section to clip. The shortcut menu appears.
4. Click **Set Active**. The model is clipped.

Note: You can click **Display > Flip** to change the direction of clipping on the model.


To Rename a Cross Section

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.

3. Under **Names**, click the cross section that you want to rename.
4. Click **Edit > Rename**.
5. In the name cell, type a new name.
6. Press **ENTER**.


To Copy a Cross Section

Use the **Copy** command from the **Edit** menu in the **View Manager** to copy a cross section from another model. The model from which the cross section is copied must be of the same type as the model in which the new cross section is being created (part or assembly).

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Click **Edit > Copy**.
4. Under the Names column, type a name for the new cross section and press **ENTER**. The **Open** dialog box opens.
5. In the **Open** dialog box, select the reference model that contains the cross section you want to copy. The reference model can be the same as the current model.
6. The reference model appears in a secondary window. The **XSEC NAMES** menu lists the cross sections that are present in the reference model. Select the cross section that you want to copy. The references for the selected cross section are highlighted in green on the reference model.
7. For each highlighted reference, click a corresponding reference on the current model. The cross section is copied to the current model and displayed.


Note: You can also copy or group features that reference a cross section, for example, datum curves created with the **UseXsec** command.

To Remove a Cross Section

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under the Names column, click the cross section that you want to remove.
4. Click **Edit > Remove**.
5. Confirm by pressing **OK** at the system prompt.


To Modify Section Dimensions

You can change the dimensions of an offset cross sections.

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under the Names column, click the cross section you want to modify.
4. Click **Edit > Redefine**. The **XSEC MODIFY** menu appears.
5. Click **XSEC MODIFY > Dim Values**. The cross section dimensions appear.
6. Modify the dimensions as needed. Click **OK** from the **SELECT** menu. The part is automatically regenerated.
7. Click **Done/Return**.

To Redefine an Offset Cross Section

You can redefine an existing offset cross section to change the section attributes, section geometry, and dimensioning scheme.

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. In the Names column, select the existing offset cross section that you want to redefine.
4. Click **Edit > Redefine**.
5. Click **XSEC MODIFY > Redefine**. From the **REDEFINE** menu, select options to modify:
 - **Attributes**—Redefine the attributes of the feature.
 - **Direction**—Redefine the direction of feature creation.
 - **Section**—In Sketcher, modify the sketch of the cross section. You can add, delete, or modify sketcher entities and dimensions.
 - **Scheme**—Add or delete dimensions in Sketcher.
6. Click **Done**.

Modifying Cross Sections

About Modifying Cross-Hatching

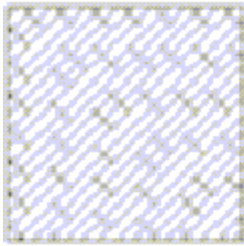
The default line style for cross sections is leader style. When you quit the **View Manager**, the cross section is displayed in yellow. You can modify the cross-hatching in planar and offset cross sections.

Cross-hatch cosmetics are independent of orientation. Your spacing specifications are stored in model coordinates, while angles are measured from the horizontal of the current view. The latter is designed to retain the same angle of cross hatching when the model is retrieved in a drawing cross-sectional view.

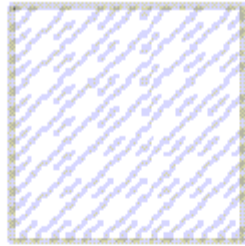
Note: Modifying the cross-hatching in **Part** or **Assembly** mode does not update any existing drawing views.

Standard Cross-Hatch Patterns

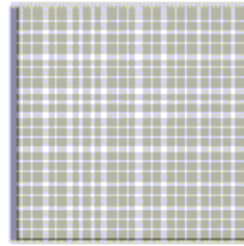
Note: Nine standard crosshatch patterns are included with Detailed Drawings (see the following figure).



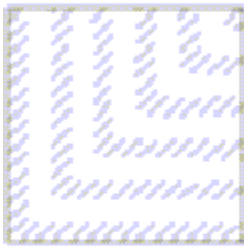
1



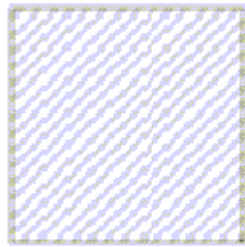
2



3



4



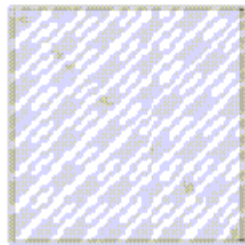
5



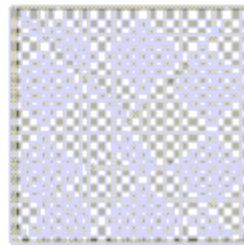
6



7



8




9

1. Aluminum
2. Copper
3. Electric
4. Glass
5. Iron
6. Plastic
7. Steel


8. Titanium
9. Zinc

To Display a Cross Section as Hatched or Filled


After you defined a planar cross section, you can display it as hatched or filled.

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under the Names column, click the cross section that you want to change.
4. Click **Edit > Redefine**.
5. Click **XSEC MODIFY > Hatching**.
6. From the **MOD XHATCH** menu, select one of these options:
 - **Hatch** (default)—Create a hatched cross section. To see the current color and line definitions, click **Line Style** from the **MOD XHATCH** menu.
 - **Fill**—Fill the cross section in a solid color.
7. When finished, click **MODIFY XHATCH > Done**.

To Change the Fill Color

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under the Names column, click the cross section that you want to change.
4. Click **Edit > Redefine**.
5. Click **XSEC MODIFY > Hatching**.
6. From the **MOD XHATCH** menu, click **Fill** and then **Line Style**. The **Color** dialog box opens.
7. Select a color or create one.
8. Click **OK**. Pro/ENGINEER updates the color in the cross section and closes the dialog box.

To Modify Hatch Spacing

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under there Names column, click the cross section that you want to change.
4. Click **Edit > Redefine**.


5. Click **XSEC MODIFY > Hatching > Spacing**. The **MODIFY MODE** menu appears.
6. Specify if you want to change an individual or all pattern lines:
 - **Individual**—Apply the change in hatch spacing to an individual pattern line.
 - **Overall**—If hatching has multiple line patterns, apply the change in hatch spacing to all line patterns.

Note: In most cases, there is only one pattern. Therefore, choosing **Individual** or **Overall** gives the same result.

7. Define the spacing using one of these options:
 - **Half**—Change spacing to one half of the current spacing by creating more hatch lines.
 - **Double**—Change spacing to double the current spacing by decreasing the number of hatch lines.
 - **Value**—Enter a value for the spacing. The system calculates the number of hatch lines based on the spacing value.
8. When finished, click **MODIFY XHATCH > Done**.

Note: You can set the default spacing and angle of cross-hatching in newly created planar and offset cross sections by adding parameters. Click the topic *To Set the Default Spacing and Angle* under **See Also**.

To Modify the Hatch Angle

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under the Names column, click the cross section that you want to change.
4. Click **Edit > Redefine**.
5. Click **XSEC MODIFY > Hatching > Angle**.
6. Specify if you want to change an individual or all pattern lines:
 - **Individual**—Apply the change in hatch angle to an individual pattern line.
 - **Overall**—If hatching has multiple line patterns, apply the change in hatch angle to all line patterns.

Note: In most cases, there is only one pattern. Therefore, choosing **Individual** or **Overall** gives the same result.

7. Select the angle from the **MODIFY MODE** menu.
8. When finished, click **MOD XHATCH > Done**.

Note: You can set the default spacing and angle of cross-hatching in newly created planar and offset cross sections by adding parameters. Click the topic *To Set the Default Spacing and Angle* under **See Also**.

To Set the Default Spacing and Angle


You can set the default spacing and angle of cross-hatching for new planar and offset cross sections. To do that, add the following model parameters using **Tools > Parameters:**

- `default_xhatch_angle` with a value for the angle between -360 and 360 degrees.
- `default_xhatch_spacing` with a positive value for the spacing.

Consider the following information for setting the defaults for spacing and angle:

- If you modify the spacing and angle manually after setting the hatching using these parameters, the change overrides the parameter values.
- Changing the parameter values for angle and spacing only affects new cross sections—not previously created ones that have the parameters associated with them.
- These parameters do not affect different material styles for cross-hatching. They affect only the default style.
- If you enter an invalid value for a parameter, the system uses the default.

To Modify Hatch Line Style

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under the Names column, click the cross section that you want to change.
4. Click **Edit > Redefine**.
5. Click **XSEC MODIFY > Hatching > Spacing**.
6. Click **MOD XHATCH > Line Style**. The **Line Style** dialog box opens.
7. Select a new line style and click **Apply**. The line style in the section is updated.
8. To change the line color, click the color box. The **Color** dialog box opens.
9. Select a new color and click **OK**. The **Color** dialog box closes, and the new color choice appears in the **Line Style** dialog box.
10. Click **Apply**, then **Close**.
11. To update the display, click **MOD XHATCH > Fill > Hatch**.

Line Style Dialog Box

Use the options in the **Line Style** dialog box to modify line styles:

Style—Use to set the line style name by selecting from a list of existing line style names.

Existing Line—Use **Select Line** to copy the line style of an existing line.


Line Font—Use to set a line font by selecting from a list of all available line fonts.

Color—Use to set the color of selected lines using the **Color** dialog box.

Click **Apply** to apply the line style or **Cancel** to close the dialog box without making changes. If you click **Reset**, the line style resets to the previous values.

To Modify the Offset of Parallel Cross-Hatching


Use the **Offset** command to change the offset of the second line for the parallel type of cross-hatching.

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under there Names column, click the cross section that you want to change.
4. Click **Edit > Redefine**.
5. Click **XSEC MODIFY > Hatching > Offset**.
6. Type a value for the offset in drawing units and press **ENTER**.
7. When finished, click **MOD XHATCH > Done**.

To Save a Cross-Hatch Pattern

You can save and later retrieve a cross-hatch pattern. The configuration file option `pro_crosshatch_dir` specifies the path to the default directory where the patterns are saved. When you choose the **Retrieve** command from the **MOD XHATCH** menu, Pro/ENGINEER searches the default directory.

To save a pattern:

1. Click  or **View > View Manager**. The **View Manager** dialog box opens.
2. Click **Xsec**.
3. Under there Names column, click the cross section that you want to save.
4. Click **Edit > Redefine**.
5. Click **XSEC MODIFY > Hatching**.
6. Click **MOD XHATCH > Save**.
7. Type the name of the crosshatch pattern and press **ENTER**. The crosshatch pattern is stored to file with the extension `.xch` in the current directory.

To retrieve a pattern:

1. Click **MOD XHATCH > Retrieve**. The **Open** dialog box opens, containing the names of cross-hatch patterns in the current directory, all crosshatch patterns in the default cross-hatch directory, and the standard cross-hatch patterns.
2. Select a cross-hatch pattern from the list and click **Open**. The selected crosshatch pattern places the current cross-hatching in the view.
3. To update the color, click **Fill**.

System Colors

About System Colors

Pro/ENGINEER provides default system colors enabling you to easily identify model geometry, datums, and other important display elements. These default system colors are displayed in the **System Colors** dialog box (**View > Display Settings > System Colors**) under the **Graphics** tab. The **User Interface**, **Datum** and **Geometry** tabs enable you to further assign system colors to these elements.

Customizing System Colors

You can customize the default system colors by making the following changes:

- Define, save, and open color schemes
- Store a color scheme so you can reuse it
- Open a previously used color scheme
- Customize colors used in the user interface
- Change your entire color scheme to a predefined color scheme (such as black on white)
- Change the top or bottom background colors
- Redefine basic colors used in models
- Assign colors to be used by a geometry or datum entity

You can turn on and off Blended Background (gray and dark gray) by clicking the **Blended Background** check box under the **Graphics** tab. You can set the blended background from the **Blended Color** dialog box (click the **Edit** button under the **Graphics** tab).

Modifying System Colors Using Configuration Options

In addition to using the **System Colors** dialog box (accessed by clicking **View > Display Settings > System Colors**), you can modify each of the following colors by using the `config.pro` file. Refer to the topics under *Configuring Pro/ENGINEER Fundamentals* for more information.

User Interface Tab

Set colors for text, editable, and selected area, and background. Click the color to open the 16-color standard Windows palette. Click **Other** to customize and **Cancel** to close the color palette.

Datum Tab

Set colors for datum planes, axes, points, and coordinate systems. To change the default datum colors, you click the color to display the 13 default color settings, then click on the desired color.

Geometry Tab

Set colors for References, Sheet Metal Surface, Spine Surface Mesh, Cable, Quilt Edge, Mold and Cast Surfaces, and the ECAD Area. To change the default geometry colors, you click the color to display the 13 default color settings, and select the desired new color.

File Menu

Using the **File** menu, you can open an existing color scheme or save your current scheme.

Open—Allows you to restore a previously used color scheme by reading in a system colors (.sc1) file.

Save—Allows you to save the current system colors in a .sc1 file for future use.

Scheme Menu

A color scheme affects system colors. The default color scheme is a light to dark grey gradient background. You change the color scheme by using the **Scheme** menu. Pro/ENGINEER provides the following color schemes:

- **Black on White**—Displays black entities shown on a white background.
- **White on Black**—Displays white entities shown on a black background.
- **White on Green**—Displays white entities shown on a dark green background.
- **Initial**—Resets the color scheme to the one defined by the configuration file settings.
- **Default**—Resets the color scheme to the default Pro/ENGINEER Wildfire color scheme (light to dark grey gradient background).
- **Use Pre-Wildfire Scheme**—Resets the color scheme to the 2001 release of Pro/ENGINEER (blue to black gradient background).

To Open an Existing Color Scheme

1. Click **View > Display Settings > System Colors**. The **System Colors** dialog box opens.

2. Click **File > Open**.
3. In the **Open** dialog box, browse to the color scheme .sc1 file that you want to use.
4. Click **OK**. Pro/ENGINEER sets the display using the color scheme.

To Redefine a System Color

1. Click **View > Display Settings > System Colors**. The **System Colors** dialog box opens.
2. Click the tab related to the colors that you want to change. Pro/ENGINEER provides the following tabs:
 - **Graphics**—Displays the default system colors for the graphical elements.
 - **User Interface**—Sets the colors for text, editable and selected areas, and the background.
 - **Datum**—Sets the colors for datum planes, axes, points, and coordinate systems.
 - **Geometry**—Sets the colors for references, sheet metal surfaces, spine surface mesh, cables, quilt edges, and the ECAD area.
3. Click the button to the left of the check box for the system color you want to redefine.
4. In the **Color Editor** dialog box, you can adjust the **RGB** or **HSV** controls to create the color that you want. For example, to set **Background** to white, move the R, G, and B slider handles all the way to the right. To set **Background** to black, move the slider handles all the way to the left.
5. When you redefine a color, the check box for that color turns black to indicate a color change. Click the check box to reset the entity color back to the original color. Click it again to reset the color to the new color.
6. When you finish editing the chosen color, click **OK** in the **Color Editor** dialog box.
7. Repeat step 2. through step 4. to change other system colors.
8. Click **OK** in the **System Colors** dialog box.

To Define a Custom Background Color

1. Click **View > Display Settings > System Colors**. The **System Colors** dialog box opens.
2. Click the **Graphics** tab and then click the **Blended Background** check box to turn on or off the Blended Background.
3. Click **Edit**. The **Blended Color** dialog box opens.
4. To change the top or bottom background color, select the appropriate check box.

5. Click the color sample button to the left of the check box. The **Color Editor** dialog box opens.
6. Specify a new color. The color sample button changes to the new color. The new background color gradations that you selected appear in the preview window.

Note: If you want to have the backgrounds of windows with 3D objects automatically update as you change background colors, make sure that **Dynamically update model window** is selected. This is the default option.

7. Click **OK**. The **Color Editor** dialog box closes.
8. Click **OK** in the **System Colors** dialog box.

To Save the Current Color Scheme

1. Click **View > Display Settings > System Colors**.
2. In the **System Colors** dialog box, click **File > Save**.
3. Use the **Save** dialog box to name and save your current color scheme as a file. The file names for color schemes contain the extension `.scl`.

Note: To have those colors in place for the next session, this file must be pointed to the setting in the configuration option `system_colors_file`.

Tip: Modifying System Colors Using the Configuration File

In addition to using the **System Colors** dialog box (accessed by clicking **View > Display Settings > System Colors**), you can modify each of the following colors by using the `config.pro` file. The options are:

- `system_letter_color` (corresponds to the **Letter** option in the **System Colors** dialog box)
- `system_highlight_color` (corresponds to the **Highlight - Primary** option in the **System Colors** dialog box)
- `system_colors_file` (Alternatively, you can create a separate file with all of these values specified and use this option to specify the file location.)
- `system_background_color` (corresponds to the **Background** option in the **System Colors** dialog box)
- `system_hidden_color` (corresponds to the **Hidden Line** option in the **System Colors** dialog box)
- `system_edge_high_color` (corresponds to the **Highlight - Edge** option in the **System Colors** dialog box)
- `system_dimmed_menu_color` (corresponds to the **Highlight - Secondary** option in the **System Colors** dialog box)
- `system_volume_color` (corresponds to the **Manufacturing Volume** option in the **System Colors** dialog box)









- `system_section_color` (corresponds to the **Sketched Curve** option in the **System Colors** dialog box)
- `system_sheetmetal_color` (corresponds to the **Sheet Metal** option in the **System Colors** dialog box)
- `system_curves_color` (corresponds to the **Curves** option in the **System Colors** dialog box)
- `system_geometry_color` (corresponds to the **Geometry** option in the **System Colors** dialog box)
- `system_colors_file` (specifies a location for a separate file containing color values)





A keyword must be followed by three decimal values in the range from 0 through 100, which specifies the percentages of red, green, and blue (in that order) in the resulting color. These values are identical to those you see in the R, G, and B fields in the **Color Editor** dialog box (accessed by clicking a color button in the **System Colors** dialog box).

System Colors






Pro/ENGINEER provides system colors enabling you to easily identify model geometry, datums, and other important display elements. The following tables list the default system colors and their corresponding RGB (Red Green Blue) values.

Note that these default system colors are listed in the **System Colors** dialog box (**View > Display Settings > System Colors**) on the **Graphics** tab.


Sample	Color Name	RGB Values (R,G,B)	Description
	White	255,255,255	Identifies geometry.
	Gray	150,148,138	Identifies hidden lines.
	Blue	52,160,239	Identifies sketches.
	Dark blue	60,56,255	Identifies curves.
	Magenta	237,64,199	Identifies quilts.
	Purple	129,0,174	Identifies mill volumes in manufacturing, and mold volumes in molding and casting.
	Green	19,140,46	Identifies the driving surface of sheet metal parts.
	Brown	170,118,56	Identifies datum planes, axes, points, and coordinate systems.



Sample	Color Name	RGB Values (R,G,B)	Description
	Yellow	255,255,0	Identifies text.
	Dark red	148,0,41	Identifies (primary) highlighted items.
	Dark gray	94,93,89	Identifies (secondary) highlighted items such as weak dimensions and non-current subsections.
	Green	55,255,38	Identifies highlighted references.

The following table lists the default system colors that Pro/ENGINEER reserves for feedback during selection, and during current tool usage:

Sample	Color Name	RGB Values (R,G,B)	Description
	Cyan	0,255,255	Identifies the preselected item.
	Red	255,0,51	Identifies selected items.
	Orange	255,138,0	Identifies secondary selected items.
	Yellow	255,255,0	Identifies previewed results of the current activity.
	Pale yellow	252,255,198	Identifies secondary previewed results of the current activity.

The following table lists the default system colors that Pro/ENGINEER uses for displaying the geometry area background:

Sample	Color Name	RGB Values (R,G,B)	Description
	Black	51,50,46	Identifies the background color used in windows that display 2D geometry, and the non-blended background color used in windows that display 3D geometry. Note: You must clear the Blended Background check box in the System Colors dialog box

Sample	Color Name	RGB Values (R,G,B)	Description
			(Graphics tab) to display this color in windows that display 3D geometry.
	Gray	158,155,145	Identifies the top portion of the graphics window blended background in windows that display 3D geometry. Note: You must click Edit on the System Colors dialog box (Graphics tab) to modify this color.
	Light gray	227,225,213	Identifies the bottom portion of the graphics window blended background in windows that display 3D geometry. Note: You must click Edit on the System Colors dialog box (Graphics tab) to modify this color.

Using Model Colors in Drawings

You can set the system to use the original model colors in all associated drawings. Keep in mind:

- 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 appear in the standard hidden line color in the model.
- Any assigned process assembly colors always supercede the setting for using model colors or assigned drawing colors in the drawing.

In **Drawing** mode, set the system to use model colors in drawings by using the **Model Colors** option in the **VIEW DISP** menu (accessed when you select a drawing, click **Edit > Properties > View Disp**).

The Insert Menu

About the Insert Menu

Options for creating datum feature types such as holes, rounds, datums, points, axes, and planes, and for creating other features, such as holes, shells, ribs, drafts, chamfers, cuts, cosmetic features, and more. You can also create advanced features such as pipes, toroidal bends, and patches. Also includes options for appending data from external files to the current model. Additional options include working with shared data, and advanced blends.

About Datum Feature Creation

You can create all datum feature types, including points, axes, planes, coordinate systems, curves, graphs, and analyses, by clicking **Insert > Model Datum**.

During feature creation, model investigation, component assembly, and other operations, you are required to select one or more datum points, axes, planes, and/or coordinate systems, but in many cases the desired datum does not exist. It is unnecessary to abort your current operation in order to navigate the **Feature** menus and create the desired datum feature.

You can create datum points, axes, planes (including offset planes), curves, ribbons and coordinate systems at any time using **Insert > Datum**. You can create these features even during the creation of another feature, and then use them as references of that feature. At any point in feature creation, measurement, or analysis, if a datum feature is needed, you can create it.

This applies even for nested datum features, such as a condition where a point, axis, and plane must be created simply to get a reference for an offset make datum.

Note: Datum features, like other features, may have names that are identical to individual entity names. For example, a datum axis feature may contain an axis entity. In the former case, an axis is a type of feature, and in the latter case, it is a type of entity. Likewise, a datum point feature may contain many datum point entities.

For more information about and procedures for creating datum features, see the topics under *Part Modeling* in the Pro/HELP Table of Contents.

About the Style Function

The **Insert** menu contains the **Style** option, which inserts the **Styling** menu in the menu bar. Styling is an intuitive and interactive design environment for the creation of freeform curves and surfaces embedded in the Pro/ENGINEER parametric environment. The super-features are created at part level and allow you to model curves and surfaces with as many or as few constraints as you like. The four-view layout allows you to work simultaneously in multiple views, and the unique softpoint technology results in lighter and more flexible curves. The editing controls allow for quick, intuitive and dynamic interaction with models.

The Analysis Menu

About the Analysis Menu

Use these commands to display information about the models and modify the options for analyzing model parameters. Options exist for

- Comparing the features or geometric differences between two parts
- Performing model, curve, surface, Mechanical, Excel, or user-defined analyses
- Performing sensitivity analyses
- Feasibility or optimization studies, or creating a multi-objective design study
- Comparing a sheet to an existing picture and display the results in the Drawing mode

You can perform the following types of analysis on a model:

- Model—Analyzes the model properties
- Measure—Measures model geometry, such as lengths, angles, areas, and so on
- Geometry—Analyzes surface properties and curves

These analysis tools are used in applications such as solid modeling, surface modeling, and behavioral modeling.

About a Saved Analysis

You can save an analysis with the model. The saved analysis is displayed in the Pro/ENGINEER graphics window until you explicitly hide or delete it using the **Saved Analysis** dialog box.

You can open a saved analysis by clicking **Analysis > Saved Analysis**. Use the **Saved Analysis** dialog box to perform the following operations:



- Hide or unhide a saved analysis.
- Redefine a selected analysis.
- Use the filter to select the type of analysis that you want to view.
- Delete a saved analysis.

You can also create an analysis feature from a saved analysis. However, you can redefine a saved analysis only for the **Geometry** type of analysis.

To Create an Analysis Feature From a Saved Analysis

1. Click **Analysis > Saved Analysis** to open a saved analysis. The **Saved Analysis** dialog box opens.

Note: Using the **Saved Analysis** dialog box, you can redefine a saved analysis only for the **Geometry** type of analysis.

2. Click  in the **Saved Analysis** dialog box.
3. Specify a name for the new feature you want to create, in the text area displayed on the dashboard.
4. Click . An analysis feature with the specified name is created on the Model tree.

Analyzing Model Properties

About Model Analysis

Use **Analysis > Model** and select the appropriate type of analysis to analyze the model properties.

You can perform the following types of model analysis:

- **Mass Properties**—Computes mass properties for a part, assembly, or drawing. The Assembly Mass Properties type of analysis is available in Assembly mode and assembly drawing while the Model Mass Properties type is available in Part and Drawing modes.
- **X-Section Mass Properties**—Computes the mass properties for a cross section.
- **One-Sided Volume**—Computes the volume of a model on one side of a designated plane. The One-Sided Volume type of analysis is available in Part and Drawing modes.
- **Pairs Clearance**—Computes the clearance distance or interference between two objects or entities (any combination of subassemblies, parts, surfaces, cables, or entities) in a model. The Pairs Clearance type of analysis is available in Part, Assembly, Piping, and Drawing modes.

Note: Exploded views in Assembly are cosmetic and have no effect on clearance computations.

- **Global Clearance**—Computes the clearance between each part or subassembly of a model. The Global Clearance type of analysis is available in Assembly, Piping, and Drawing modes.

Note: Exploded views in Assembly are cosmetic and have no effect on clearance computations.

- **Volume Interference**—Verifies that the selected closed quilt has nothing interfering with it. The Volume Interference type of analysis is available in Assembly and Drawing modes.

Volume interference checking is used in ECAD to ensure that keepin or keepout areas have not been violated.

- **Global Interference**—Displays information (in a drawing or assembly) about interference between each part or subassembly in a model. The Global Interference type of analysis is available in Assembly and Drawing modes.
- **Short Edge**—Computes the length of the shortest edge in a selected part or component and also determines how many edges in the model are shorter than the specified length. The Short Edge Type analysis is available in Part and Assembly modes.
- **Edge Type**—Determines the type of geometry used to create the selected edge. The Edge Type analysis is available in Part, Assembly, and Drawing modes.
- **Thickness**—Checks whether the thickness of a part is greater than a maximum value or less than a minimum value or both and calculates the area for thickness check. The Thickness type of analysis is available in Part and Assembly modes.

Note: For clearance and interference checks, the accuracy of the calculation is determined by part accuracy. The accuracy of a clearance measurement or interference volume is controlled by the `measure_sig_figures` configuration option.

If the selected objects do not interfere, the minimum clearance appears graphically as a red line. A small red circle with a crosshair appears at each end of the line to identify the location at which the clearance is being measured. The clearance value appears in the message window. If there is an interference between the selected objects, Pro/ENGINEER highlights the volume of interference and the curve or point of intersection in the graphics window, as appropriate for the selected items.

Exploded views are only cosmetic and have no effect on clearance computations. The displayed results in such cases correspond to an unexploded model.

Note: Adding a feature or creating a saved analysis is not possible in the Drawing mode.

About Computing Mass Properties

You can set the default mass units for all new parts and assemblies that you create using the `pro_unit_mass` configuration option.

Mass properties for a part or assembly is calculated using system parameters within a relation. However, mass properties are not automatically updated when the model changes. You must recalculate the mass properties to see the effect of model changes.

Note: In Welding assemblies, you can include or exclude welds from the mass properties calculation. Mass property calculations for light welds are approximate, unless you specify weld section references. Set the `add_weld_mp` configuration option to `yes` to include welds in the mass properties calculation. The default is `no`.

If dimension bounds have been set, mass property calculations are based on hypothetical dimension values.

If features are suppressed, the mass properties are calculated as if the features did not exist. If a part has been simplified for assembly purposes and contains many suppressed features, mass properties may be assigned to the part so that those values are used in calculating mass properties of an assembly.

If parts on layers have been blanked from the current view, they are still used to calculate the mass properties of an assembly. Blanking items only affects the display of the object, not its composition.

The computed mass properties for a part or assembly include:

- Volume—Total volume of the model
- Surface area—Total surface area of the model
- Density—For part mass properties, the specified part density is listed here.

For assembly mass properties, the average density of its components is listed. The average density is calculated as the total mass divided by the total volume:

$$\text{Average density} = (\text{total mass of assembly}) / (\text{total volume of assembly})$$

The density of each individual part is used in the calculation of the mass properties.

- Mass—Total mass of the model
- Center of gravity—Location of the center of gravity with respect to a specified coordinate system. The center of gravity and coordinate system are displayed graphically on the model.
- Inertia tensor at the specified coordinate system origin and axes. The values are computed as follows:

$$I_{xx} = \int_{\text{vol}} \rho(y^2 + z^2) dV$$

-and so forth.

$$I_{xy} = - \int_{\text{vol}} \rho(xy) dV$$

-and so forth

-where ρ = specified part density.

- Inertia tensor at the center of gravity and oriented along the specified coordinate system axes
- Principal moments of inertia—Moments of inertia at the center of gravity with respect to the principal axes

$$I_1 = \int_{\text{vol}} \rho(y^2 + z^2) dV$$

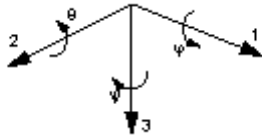
The principal coordinate axes, which define a new coordinate system, are labeled 1, 2, 3 instead of x, y, z. In the preceding equation, y is the coordinate along the principal axis 2 and z is the coordinate along the principal axis 3.

- Rotation matrix and rotation angles—Rotation from the coordinate axes to the principal axes in matrix and angular form. Rotation matrix is computed as follows:

$$\begin{bmatrix} \cos\theta\cos\psi & -\cos\theta\sin\psi & \sin\theta \\ \sin\phi\sin\theta\cos\psi + \cos\phi\sin\psi & -\sin\phi\sin\theta\sin\psi + \cos\phi\cos\psi & -\sin\phi\cos\theta \\ -\cos\phi\sin\theta\cos\psi + \sin\phi\sin\psi & -\cos\phi\sin\theta\sin\psi + \sin\phi\cos\psi & \cos\phi\cos\theta \end{bmatrix}$$

-where ϕ is the rotation angle about the 1 axis, θ is the rotation angle about the 2 axis, and ψ is the rotation angle about the 3 axis.

Rotation Angles




- Radii of gyration—The radii of gyration at the center of gravity with respect to the principal axes
- For an assembly, a summary of the mass properties of its components




To Analyze Mass Properties


1. Click **Analysis > Model > Mass Properties**. The **Mass Properties** dialog box opens. **Quick** is the default analysis type.
2. Click in the **CSYS** collector if it is not active by default and select a reference coordinate system. The name of the selected coordinate system is displayed in the collector. You can click the **Use Default** check box to select and use the default coordinate system.

The result of the analysis is displayed in the result area at the bottom of the **Mass Properties** dialog box. You can view the report of the analysis in the

Information Window by clicking .

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new coordinate system in the collector to create a new analysis.


3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Mass Properties** dialog box.

- c. Select **Saved** to save the analysis with the model, and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to create or change feature options of the current analysis, if required. You can access the feature options such as parameters or datum features only when you select a feature type of analysis.
5. Optionally, to customize or edit your analysis, perform the following steps:
 - a. Specify the material density in the **Density** box or select a density value from the list of most recently used values in the **Density** box. The density value is saved with the model. The default material density is 1.
Note: If a material file is already applied to the model, you cannot specify the material density.
 - b. Change the relative accuracy or accept the default accuracy value, 0.000010, in the **Accuracy** box. You can also select an accuracy value from the list of most recently used values in the **Accuracy** box. Part accuracy must not be less than 1/100th of the mass property accuracy.
6. Click  to compute the analysis. A coordinate system is displayed at the center of gravity in the Pro/ENGINEER graphics window.





To Compute the Mass Properties for a Cross Section

Note: Cross-section mass properties are available only for planar x-sections.

1. Click **Analysis > Model > X-Section Mass Properties**. The **Cross Section Properties** dialog box opens. **Quick** is the default analysis type.
2. Select a cross section name for which you want to compute the mass properties in the **Name** list.
Note: The **Name** list is available only if you have a cross section in the model.
3. Click in the **Plane** collector if it is not active by default and select a reference datum plane. The name of the selected plane is displayed in the collector.
4. Click in the **CSYS** collector and select a coordinate system. The name of the selected coordinate system is displayed in the collector. You can click the **Use Default** check box to select and use the default coordinate system.

The result of the analysis is displayed in the result area at the bottom of the **Cross Section Properties** dialog box. You can view the report of the analysis in the Information Window by clicking .

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new plane and coordinate system in the collector to create a new analysis.

5. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
6. Optionally, to save or create an analysis feature, perform the following steps:
- Click the **Analysis** tab if it is not selected by default.
 - Select the desired analysis types from the list at the bottom of the **Cross Section Properties** dialog box.
 - Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - If required, rename the analysis in the box adjacent to the list.
 - Click the **Feature** tab to create or change feature options of the current analysis, if required. You can access the feature options such as parameters or datum features only when you select a feature type of analysis.
7. Optionally, to customize or edit your analysis, change the relative accuracy or accept the default accuracy value, 0.000010, in the **Accuracy** box. You can also select an accuracy value from the list of most recently used values in the **Accuracy** box.
8. Click  to compute the analysis. A coordinate system is displayed at the center of gravity of x-section in the Pro/ENGINEER graphics window.

List of Computed Mass Properties for a Cross Section

The computed mass properties for a cross section include the following:

- Volume—Volume of the cross section.
- Surface Area—Area of the cross section.
- Density—Density of the mass of a cross section.
- Mass—Total Mass of the cross section.
- Center of gravity—Location of the center of gravity with respect to a specified coordinate system. The center of gravity and coordinate system are displayed graphically on the model.
- Inertia tensor at the specified coordinate system origin and axes.
- Inertia tensor at the center of gravity and oriented along the specified coordinate system axes.
- Principal moments of inertia—Moments of inertia at the center of gravity with respect to the principal axes.

$$I1 = \int_{\text{area}} y^2 dS$$

$$I2 = \int_{\text{area}} x^2 dS$$

- The principal coordinate axes are labeled 1 and 2 instead of x and y. In the preceding equations, x is the coordinate along principal axis 1 and y is the coordinate along principal axis 2.
- Polar moment of inertia—Cross-sectional polar moment of inertia at the center of gravity.

$$Iz = \int_{\text{area}} (x^2 + y^2) dS$$

- Rotation matrix and rotation angle—Rotation around the z-axis from the coordinate axes to the principal axes, in matrix and angular form.
- Radii of gyration—The radii of gyration at the center of gravity with respect to the principal axes.
- Section moduli and the corresponding points—Calculated for each principal axis (for example, 1-axis, 2-axis) by dividing the area moment of inertia with respect to the chosen axis by the distance to the point or points of the cross section that are farthest from the chosen axis. These points have the lowest (most negative) and highest (most positive) coordinate values along the orthogonal principal axis in the cross section plane.

Note: Cross section mass properties are not available for offset cross sections.





To Calculate the Volume of One Side of a Model

1. Click **Analysis > Model > One-Sided Volume**. The **One Sided Volume** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Plane** collector if it is not active by default and select a reference datum plane. The name of the selected plane is displayed in the collector.




The result of the analysis is displayed in the result area at the bottom of the **One Sided Volume** dialog box. An arrow in the graphic window indicates the side for which the volume is calculated. You can flip the arrow to change the direction by clicking on the arrow in the graphic window. You can view the report of the

analysis in the Information Window by clicking .

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new plane in the **Plane** collector to create a new analysis.

3. Click  to complete the analysis or  to cancel the analysis.
Alternatively, click  to start a new analysis.
4. Optionally, to save or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **One Sided Volume** dialog box.
 - c. Select **Saved** to save the analysis with the model, and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to create or change feature options of the current analysis, if required. You can access the feature options such as parameters only when you select a feature type of analysis.
5. Optionally, to customize or edit your analysis, change the relative accuracy or accept the default accuracy value, 0.000010, in the **Accuracy** box. You can also select an accuracy value from the list of most recently used values in the **Accuracy** box.
6. Click  to compute the analysis.


To Calculate Clearance Between Two Entities

1. Select any two curves, edges, datum points, surfaces, pipe segments in Assembly mode, cable segments in Assembly mode, cables in Assembly mode, parts in Assembly mode, or components in Assembly mode.
2. Click **Analysis > Model > Pairs Clearance**. The **Pairs Clearance** dialog box opens. **Quick** is the default analysis type.
3. Click  to complete the analysis or  to cancel the analysis.
Alternatively, click  to start a new analysis.
4. Optionally, to save or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Pairs Clearance** dialog box.
 - c. Select **Saved** to save the analysis with the model, and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.

- e. Click the **Feature** tab to create or change feature options of the current analysis, if required. You can access the feature options such as parameters or datum features only when you select a feature type of analysis.
5. Optionally, to customize or edit your analysis, perform the following steps:
 - a. Click in the **From** collector and select the required entity. The name of the selected entity is displayed in the **From** collector. If the **From** entity is a surface, **Whole surface** and **Near pick** options are activated to select the **Geometry** from which you want to calculate the clearance. **Whole surface** is selected by default. If you click **Near pick**, you can select only a surface in the **To** collector.
- Note:** In the Assembly and Drawing mode, you can also select quilts or facets of the selected part, subassembly, surface, or cable in the **From** and **To** collectors to compute the clearance.
- b. Click in the **To** collector and select the required entity. The name of the selected entity is displayed in the **To** collector.
6. Select a reference plane, coordinate system, surface, axis, edge, curve, or datum plane in the **Projection Reference** collector. The projected clearance and actual clearance are displayed in the result area at the bottom of the **Pairs Clearance** dialog box.





7. If required, click  to view the projected clearance and actual clearance report in the Information Window.


Note: If you select a coordinate system to project the direction in the **Pairs Clearance** dialog box, you can select the required coordinate system in the Pro/ENGINEER graphics window. Projected clearance is displayed for the x-, y-, and z- directions.

8. Click  to compute the analysis. The clearance between the two entities is displayed in the result area at the bottom of the **Pairs Clearance** dialog box.





To Display Global Clearance Information


1. Click **Analysis > Model > Global Clearance**. The **Global Clearance** dialog box opens. **Quick** is the default analysis type.
2. Select **Parts only** or **Sub-assembly only** as a **Setup** value, to calculate the global clearance between parts or assemblies.
3. Click the **Include Harness** check box if harnesses are present in the assembly.
4. Specify the clearance value in the **Clearance** box. You can also select the clearance value from the list of most recently used values in the **Clearance** box.

5. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
6. Optionally, select **Saved** in the list at the bottom of the **Global Clearance** dialog box to save the analysis with the model, and dynamically update the analysis while modeling.
7. If required, rename the analysis in the box adjacent to the list.
8. Click  to compute the analysis. The result of the analysis is highlighted in the Pro/ENGINEER graphics window.




Pairs of components within the specified clearance value are displayed in the result area at the bottom of the **Global Clearance** dialog box. You can view the report of the analysis in the Information Window by clicking .
9. Click **Show All** to highlight the result of the analysis in the Pro/ENGINEER graphics window or **Clear** to clear the clearance items and the result of analysis.

To Display Closed Quilt Volume Interference



1. Click **Analysis > Model > Volume Interference**. The **Volume Interference** dialog box opens. **Quick** is the default analysis type.
2. click in the **Quilt** collector if it is not active by default and select the required quilt. The name of the selected quilt is displayed in the **Quilt** collector.
3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, select **Saved** in the list at the bottom of the **Volume Interference** dialog box to save the analysis with the model, and dynamically update the analysis while modeling.
5. If required, rename the analysis in the box adjacent to the list.
6. Click  to compute the analysis. The parts that interfere with the selected quilt and the area of interference are highlighted in the Pro/ENGINEER graphics window.

A list of parts that interfere with the quilt is displayed in the result area at the bottom of the **Volume Interference** dialog box. You can view the report of the analysis in the Information Window by clicking .
7. Click **Show All** to display the result of the analysis in the Pro/ENGINEER graphics window or **Clear** to clear the collector and the result of analysis.

To Display Global Interference Information

1. Click **Analysis > Model > Global Interference**. The **Global Interference** dialog box opens. **Quick** is the default analysis type.
2. Select **Parts only** or **Sub-assembly only** as a **Setup** value to compute the interference for parts or subassemblies.
3. Click **Include Quilts** to include surface quilts in the computation.
4. Click **Include Facets** to include facets in the computation.
5. Select a value for **Compute**. Click **Exact** to obtain a complete and detailed computation or **Quick** to perform a quick check. **Quick** lists any pair of parts or subassemblies that interfere. **Exact** also highlights the volume of interference.
6. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
7. Optionally, to save or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Global Interference** dialog box.
 - c. Select **Saved** to save the analysis with the model, and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to create or change feature options of the current analysis, if required. You can access the feature options such as parameters only when you select a feature type of analysis.
8. Click **Show All** to display the result of the analysis in the Pro/ENGINEER graphics window.

Alternatively, click **Show All** and then **Clear** to graphically view the result one by one. If you select **Quick** to perform a quick check, click **Verify** to verify the result in exact conditions and display the interference volume.


9. Click  to compute the analysis. Pairs of components that interfere are highlighted in the Pro/ENGINEER graphics window and displayed in the result area at the bottom of the **Global Interference** dialog box. You can view the report of the analysis in the Information Window by clicking .





To Compare Edge Lengths

1. Click **Analysis > Model > Short Edge**. The **Short Edge** dialog box opens. **Quick** is the default analysis type.




- Click in the **Part** collector and select a part for analysis.

Note: The **Part** collector is available only in the Assembly mode.

- Select a **Quick** or **Saved** type of analysis. **Quick** is the default. Click **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling.
- For a saved type of analysis, accept the default name for the current analysis or rename the analysis if required.
- The length of the shortest edge in a selected part or component is displayed in the **Length** box. In the **Length** box, type a new length or use the default length. The number of edges that are shorter than the specified length is displayed in the result area at the bottom of the **Short Edge** dialog box. You can view the report of the analysis in the Information Window by clicking .




- Click  to compute the analysis. The result of the analysis is highlighted in the Pro/ENGINEER graphics window.
- Click  to start a new analysis, if required.
- Click  to complete the analysis or  to cancel the analysis.


To Display Edge Type Information

- Click **Analysis > Model > Edge Type**. The **Edge Type** dialog box opens. **Quick** is the default analysis type.
- Click in the **Edge** collector and select an edge for analysis. The name of the selected edge is displayed in the **Edge** collector. The result of the analysis is displayed in the result area at the bottom of the **Edge Type** dialog box.
- Optionally, Select **Saved** type of analysis to save the analysis with the model, and dynamically update the analysis while modeling.
- If required, rename the analysis.
- Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.

To Check the Thickness of a Part

- Select a plane in the part to check the thickness of the part.
- Click **Analysis > Model > Thickness**. The **Thickness** dialog box opens. **Quick** is the default analysis type.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis, perform the following steps:
 - a. Select **Saved** from the list at the bottom of the **Thickness** dialog box to save the analysis with the model, and to displays and dynamically updates the analysis while modeling.
 - b. If required, rename the analysis in the box adjacent to the list.
5. Optionally, to customize or edit your analysis, perform the following steps:
 - a. Click in the **Planes** collector and select a plane. The name of the selected plane is displayed in the **Planes** collector.

Note: If you select a plane, **From Slices**, **To Slices**, **Direction**, **Use number of slices**, and **Offset** become unavailable.
 - b. If you do not select a plane, select the start and end points to insert the number of slices between the points in the model. Click in the **From Slices** collector and select a vertex as the start point. Similarly, click in the **To Slices** collector and select a vertex as the end point. The names of the selected start and end points appear in the **From** and **To** collector, respectively.
 - c. Click in the **Direction** collector and select a surface, curve, edge, axis, or coordinate system. An arrow in the graphic window indicates the surface, curve, edge, axis, or coordinate system for which the thickness is calculated.
 - d. Click the **Use number of slices** check box to specify the number of slices or use the default value.
 - e. Click the **Offset** check box and change the offset value if necessary.
 - f. Specify the maximum and minimum thickness values in the **Max** and **Min** boxes, respectively. You can also select maximum and minimum values from the list of most recently used values in the **Max** and **Min** boxes.
6. Click  to compute the analysis. The result of the analysis is highlighted in the Pro/ENGINEER graphics window.

The **Results** box displays the names of parts that have a thickness greater than the specified maximum value or less than the minimum value or both and also displays the area selected for the thickness check. You can view the report of the analysis in an Information Window.

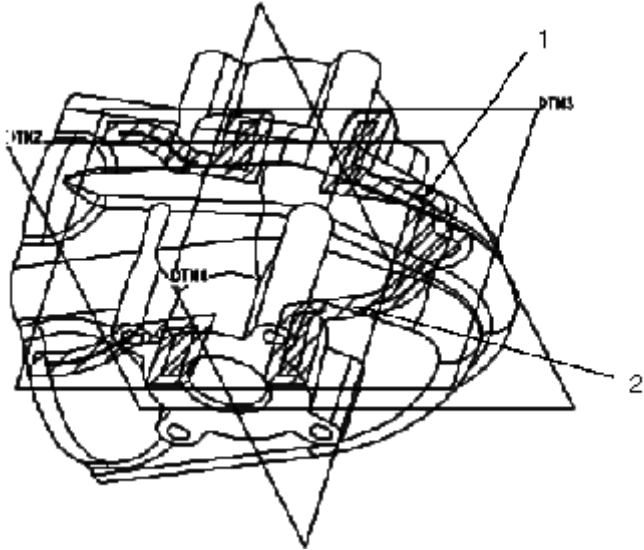
When the thickness check is complete, the cross-section is highlighted as follows:

- **Yellow**—Thickness is between the specified maximum and minimum values.

- **Red border**—Thickness exceeds the specified maximum value.
 - **Blue border**—Thickness is below the specified minimum value.
7. Click **Show All** to highlight the result in the Pro/ENGINEER graphics window or click **Clear** to clear the results.

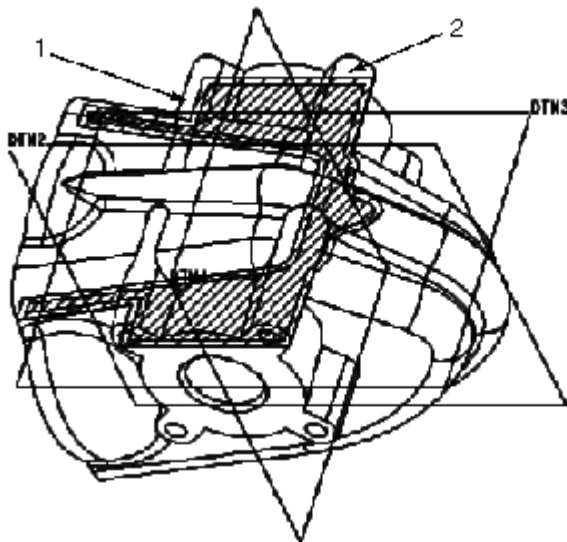
Example: Thickness Check

Thickness Check Using a Plane



1. Regions within the cross section that violate the maximum thickness are highlighted in red.
2. The cross section of the model displays cross-hatching in yellow.

Thickness Check Using Slices



1. The cross section of the model displays cross-hatching in yellow.
2. Regions within the cross section that violate the maximum thickness are highlighted in red.

Performing Measure Operations

About Measuring a Model Geometry

Using the measure analysis, you can measure model geometry with one of the following commands:

- **Distance**—Measures the distance between two entities.
- **Length**—Measures the length of the curve or edge.
- **Angle**—Measures the angle between two entities.
- **Area**—Measures the area of the selected surface, quilt, facets, or an entire model.
- **Diameter**—Measures the diameter of the surface.
- **Transform**—Displays a note pointing to the second coordinate system.

Note: In Assembly mode, all measurements are based on the unexploded assembly distances. Exploding an assembly affects only the view of the assembly components.

Types of Geometry




Geometry Type	Description
Length	Measures the length of the selected edge or curve, or chain of curves or edges.
Distance	Measures distances with respect to a basic entity (From). After you have selected the basic entity, you can measure as many distances from it as you like by selecting various other entities (To). Calculates all the distances with respect to the first entity until you restart the measuring process by selecting a new basic entity.
Angle	Measures the angle between two entities. The entities can be axes, planar curves, planar nonlinear edges, planar faces, or datum planes.
Area	Measures the area of any surface on the part or the area of a quilt or the area of a selected surface.

Geometry Type	Description
Diameter	Measures the diameter of any part surface that you have created by revolving a sketched entity or extruding an arc or a circle.
Transform	Generates a transformation file containing the transformation matrix values between two coordinate systems. This information is in a format that is required to create other coordinate systems using the From File option available in the Offset type list of the COORDINATE SYSTEM dialog box.

To Measure a Distance in an Object

1. Click **Analysis > Measure > Distance**. The **Distance** dialog box opens. **Quick** is the default analysis type.
2. Click in the **From** collector if it is not active by default and select an entity. This entity can be a vertex, datum point, scan point, coordinate system, edge or curve, scan curve, scan curve location, axis, curve feature, surface, quilt, facet vertex, facet edge, face location, curve location, edge location, pipe segment (in assembly mode), cable segment (in assembly mode), part (in assembly mode), or a component (in assembly mode).
3. Click in the **To** collector and select a second entity. This entity can be any entity listed for the first entity.

The distance between the two selected entities is calculated and displayed in the result area at the bottom of the **Distance** dialog box and in the Pro/ENGINEER graphics window. A leader points to the entity displaying the computed distance.


4. Click  to complete the analysis or  to cancel the analysis.
Alternatively, click  to start a new analysis.
5. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Distance** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.

- e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.
6. Optionally, to customize or edit your analysis, select the required entities in the **From** and the **To** collector.
 - o If you select a plane as either the From or To entity, then the **Use As Plane** check box becomes available and is selected by default. **Use As Plane** allows you to measure the distance between the two entities normal to the plane.
 - o If you select a curve or edge that appears like an arc or circle as the From or To entity, the **Use Center** check box becomes available and is selected by default. **Use Center** allows you to measure the distance between the center of the selected curves or edges that appear like an arc or circle. Clear this check box if you want to use the edge or circle and not the center of the circle or arc of the From or To entity.
 - o If you select a surface that appears like a cylindrical surface or a true cylindrical surface as the From or To entity, the **Use Center Axis** check box becomes available and is selected by default. **Use Center Axis** uses the center axis of the surface to measure the distance between the two entities.

Note: You can switch between the From and To entities or collectors by right-clicking in the Pro/ENGINEER graphics window and using the shortcut menu.

7. Click the **View Plane** check box to view the projected distance and the actual distance between the two entities in the graphics window. If you clear the **View Plane** check box, then the **Projection Direction** collector is activated.
8. Select a reference plane, surface, axis, edge, curve, coordinate system, pipe segment (in assembly mode), or cable segment (in assembly mode) to project the distance in the **Projection Direction** collector. The projected distance and actual distance are displayed in the result area at the bottom of the **Distance** box.

Note: If you select a coordinate system in the **Projection Direction** collector, then **CSYS** becomes available and you can select the required coordinate system, that is, one of **Cartesian**, **Cylindrical**, or **Spherical**.

9. **Update** is selected by default. You can see the effects of your selections as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click  to view the result of the analysis.

Tip: Measuring the Distance From an IBL Curve to Another Feature

If a feature such as a curve consists of multiple datum curves and you measure the distance between the curve and another entity such as an axis, then if you select a specific curve segment to measure the distance from, Pro/ENGINEER calculates the shortest distance between that segment and the feature.




When you create multiple curve segments with an IBL file and measure the length of these segments using **Analysis > Measure > Length**, Pro/ENGINEER calculates the shortest distance between the entire IBL curve and the axis, and not from the individual segment.

If you do not want Pro/ENGINEER to include the entire IBL curve in the measurement of the distance between the IBL curve and the axis, then you can create a composite curve on one of the segments and measure the distance from the composite curve. For information on creating a composite curve, refer to the Pro/ENGINEER help topics on creating datum curves.

To Measure an Angle in an Object

1. Click **Analysis > Measure > Angle**. The **Angle** dialog box opens. **Quick** is the default analysis type.
2. Click in the **From** collector if it is not active by default and select a curve, edge, axis, surface, facet edge, or pipe segment (in assembly mode).
3. Click in the **To** collector and select an entity. This entity can be any entity listed for the first entity.

The angle between the two selected entities is calculated and displayed in the result area at the bottom of the **Angle** dialog box. The direction of the angle is determined by the right hand rule and is displayed as an arrow in the Pro/ENGINEER graphics window. A leader points to the angle displaying the computed angle in degrees. Clicking the arrow flips the direction of the entities and the angle is recomputed.

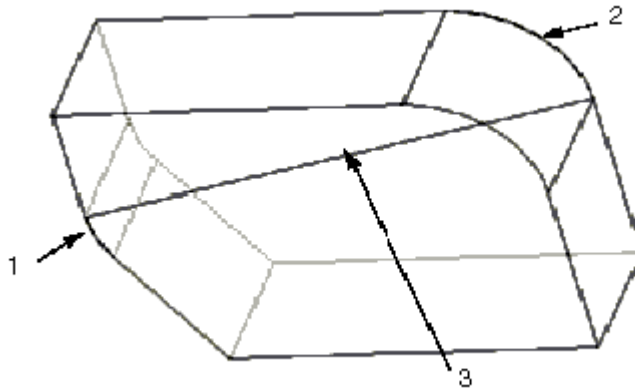
4. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
5. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Angle** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis
6. Optionally, to customize or edit your analysis, click in the **Direction** collector and select a reference plane or surface, curve, edge, axis, facet edge, pipe segment

(in assembly mode), or cable segment (in assembly mode) to project the angle in the **Direction** collector.

7. Select the range for plotting that is, one of **0-360** and **+/-180** and adjust the scale of the plot using the thumbwheel or specify the required scale in the **Scale** box.

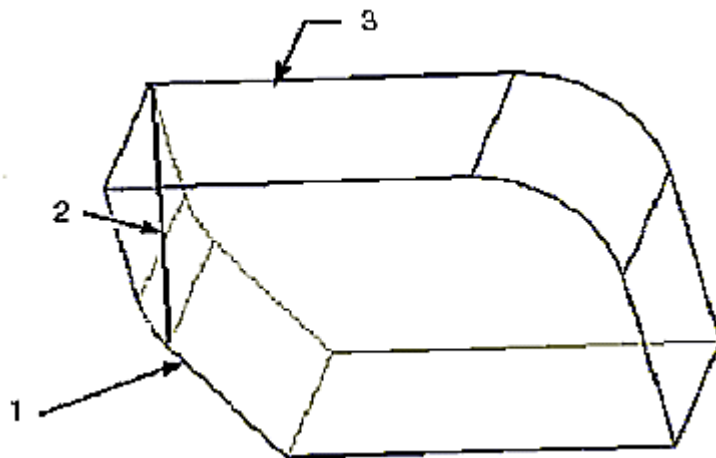
Example: Measuring Distances and Angles

Measuring the Minimum Distance between Two Curves



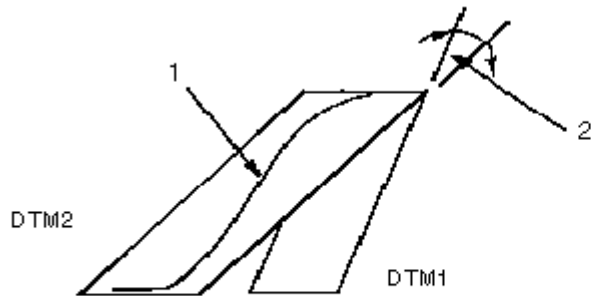
1. First entity (curve)
2. Second entity (curve)
3. Minimum distance

Measuring the Minimum Distance between Two Linear Entities



1. First entity
2. Minimum distance
3. Second entity

Measuring the Angle between a Plane and a Line






1. Planar curve
2. The smaller angle between DTM1 and DTM2 is the angle between DTM1 and the curve.

To Measure a Curve Length

1. Click **Analysis** > **Measure** > **Length**. The **Length** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Curve** collector if it is not active by default and select a curve, edge, chain, facet edge, feature, or pipe segment (in assembly mode). The name of the selected entity is displayed in the **Curve** collector.

The length of the selected curve or edge is displayed in the result area at the bottom of the **Length** dialog box and in the Pro/ENGINEER graphics window. A leader points to the curve or edge displaying the computed curve length.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Length** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.




5. Optionally, to customize or edit your analysis, click **Detail** under the **Curve** collector and select the required chain type in the **Chain** dialog box that opens. You can select standard or rule-based curves in the **References** collector in the **Chain** dialog box.

To Measure the Area of an Object

1. Click **Analysis > Measure > Area**. The **Area** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector if it is not active by default and select an entity such as a surface, quilt, facets, or a component. The name of the selected entity is displayed in the **Geometry** collector.

The area for the selected entity is displayed in the result area at the bottom of the **Area** dialog box and in the Pro/ENGINEER graphics window. A leader points to the surface displaying the computed area.




Note: The calculated area has a marginal error of about 0.15%.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Area** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.
5. Optionally, to customize or edit your analysis, click in the **Direction** collector and select a reference plane or surface, axis, edge, curve, pipe segment (in assembly mode), or cable segment (in assembly mode) to project the area.

To Measure a Diameter in an Object

1. Click **Analysis > Measure > Diameter**. The **Diameter** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Surface** collector if it is not active by default and select a surface for diameter measurement. The name of the selected surface is displayed in the **Surface** collector.




The diameter for the selected surface is displayed in the result area at the bottom of the **Diameter** dialog box and in the Pro/ENGINEER graphics window. A leader points to the diameter displaying the computed diameter.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Diameter** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.
5. Optionally, to customize or edit your analysis, click in the **Point** collector and select a point such as datum point, face location, curve location, cable or edge location at which the diameter must be measured.

To Generate a Transformation File

1. Click **Analysis > Measure > Transform**. The **Transform** dialog box opens. **Quick** is the default analysis type.
2. Click in the **From** collector and select the first coordinate system.
3. Click in the **To** collector and select the second coordinate system.

A transformation matrix between two coordinate systems in an object is displayed in the result area at the bottom of the **Transform** dialog box and as a note pointing to the second coordinate system in the Pro/ENGINEER graphics window.

4. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
5. Optionally, to save the analysis, perform the following steps:
 - a. Select **Saved** from the list at the bottom of the **Transform** dialog box to save the analysis with the model, and to display and dynamically update the analysis while modeling.
 - b. If required, rename the analysis in the box adjacent to the list.

To Measure Draft Entities in Drawings

1. Open the required drawing.
2. Click **Analysis > Measure Draft Entities**. The **Draft Measure** dialog box and the **SELECT** dialog box open.
3. Under **Type**, select the type of draft to measure.
4. Under **From** and **To**, select the first and second entity, respectively.

Note: To display the **SELECT** dialog box again, click .

5. If you select an angle, you can select **Use horizontal** to calculate the angle, instead of selecting a second entity.
6. Click **Compute**. The result appears in the **Results** box. The displayed result includes the angle, distance, intersection point, and tangent point.
7. Click **Compute** to perform another analysis and **Info** to display the results in an Information Window.
8. To create a new analysis feature that uses the current analysis definition, click **Add Feature**. You are prompted to specify a name for the analysis feature or use the default name.
9. Press ENTER to save the analysis feature.
10. Click **Close** to close the **Draft Measure** dialog box.

For more information on obtaining drawing information, refer to the topics on drawings in the Detailed Drawings module of the Pro/ENGINEER Help Center.

Analyzing Curves

About Analysis Types

When performing analysis, you can select one of the following analysis types:

- **Quick**—Displays the results of your selections at real time as you make them. **Quick** is the default.
- **Saved**—Saves the analysis with the model. **Saved** dynamically updates the analysis results as you change the geometry. The saved analysis is displayed in the Pro/ENGINEER graphics window until you explicitly hide or delete it using the **Saved Analysis** dialog box.

Note: Use **Analysis > Saved Analysis** to access a saved analysis.

- **Feature**—You can create a new feature from the current analysis of the selected point, radius, curvature, dihedral angle, offset, and deviation or their modified measurement. The new feature name is displayed in the model tree. Parameters and datums can be created for analysis features. Analysis features are used in BMX and other applications.

Note: Only some analysis types support feature creation.

Using Any Analysis Dialog Box


A separate dialog box opens for each type of analysis. These dialog boxes allow you to select the type of analysis, references, and set the options. The analysis controls are organized into the following tabbed pages:


- **Analysis**—You can select entities as references for the analysis. While all analyses have at least one selection set for references, some have additional selection sets such as a coordinate system, datum plane, datum axis, and so on depending on the type of analysis you select. You can define an analysis with options such as plot type, quality, and scale.

You can also select a **Quick**, **Saved**, or **Feature** type of analysis. **Quick** allows you to create a temporary analysis. **Saved** allows you to create a permanent analysis which is saved with the model while **Feature** allows to create a new feature for an analysis.

Note: Feature is available only for those analysis types that supports feature creation.

The result of the analysis is displayed in the result area at the bottom of the dialog box that opens depending on the type of analysis you select. Wherever


applicable, you can also click  to display the result of the analysis graphically, in the **Graphtool** window. The values of the parameters are also included in the analysis for the selected curve or edge.

Click  to view the result in an INFORMATION WINDOW.

- **Feature**—To regenerate the analysis feature, set the regeneration state to one of the following:
 - **Always**—Always regenerates the analysis feature during the model regeneration.
 - **Read Only**—Excludes the analysis feature from the model regeneration.
 - **Only Design Study**—Regenerates the analysis feature only when it is used by a design study.

You can create datums and parameters as a part of the feature. You can rename the parameter or datums of the selected entities, or accept the default name.

Feature allows you to display datums for the current analysis.

Note: If you click  to start a new analysis, the result of the previous analysis in the result area at the bottom of the dialog box is cleared. On selecting a new entity, the result of the new analysis is displayed in the result area.

Setting Plotting Resolution

You can improve the resolution of plotting using one of the following sampling types. These sampling types are available in the dialog box that opens for analyzing an offset curve, offset surface, cross sections of a surface, shaded curvature of a surface, shadow, curvature of a curve or edge, curvature of surface, slope of a surface, dihedral angle for an edge or curve, and draft check analysis.

- **Quality**—The quality of the plot is based on the number of sample points. Pro/ENGINEER performs the analysis for the number of samples that you specify as a value for **Quality**. Specify a higher value for **Quality** when you want the quality of the plot to be high. The percentage quality of the plot ranges from 0 through 100. **Quality** is the default.

Note: Specifying a higher value for **Quality** may increase the time taken for the analysis.

- **Number**—Indicates the number of samples on the curve or surface for which you are performing the analysis. The number of samples must always be greater than 1. A higher number of samples uses more sampling points, which in turn improves the accuracy of the analysis.
- **Step**—The distance between two sample points that you have specified for performing the analysis. This value increments the distance between the sample points by one step. This value must always be greater than 0.001. A lesser step value increases the accuracy of the plot.

You can change these values using the thumbwheel or the slider.

About Analyzing Curves

Curve analysis followed by surface analysis are used for surface modeling. Curve analysis checks the quality of curves or the continuity between curves before the surfaces are built from them. This analysis helps to ensure that the resultant surfaces are of good quality.

Use **Analysis > Geometry** and select one of the following analysis types to perform a curve analysis:




- **Point**—Evaluates a normal curvature vector at a datum point or a specified point on a curve or edge. Displays the curvature, normal, tangent, and radius at the selected point on the curve or edge. You can also specify a coordinate system.
- **Radius**—Evaluates and displays the minimum radius for a curve or edge at the selected point. The radius is equal to $1/\text{curvature}$.
- **Curvature**—Evaluates and displays the curvature of curves or edges. Mathematically, the curvature is equal to $1/\text{radius}$.
- **Offset**—Evaluates and displays an offset for a selected set of curves or edges. You must also select a reference plane to display the offset.
- **Deviation**—Evaluates and displays the deviation from a curve or edge to a datum point, curve, or datum point array.

To Display Information About a Point on a Curve or Edge

1. Click **Analysis > Geometry > Point**. The **Point** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Point** collector and select a datum point or point on a curve or edge of the object for analysis. The name of the selected datum point or point on curve or edge is displayed in the **Point** collector and the corresponding reference coordinate system is selected in the **CSYS** collector.

The normal vector for each newly selected entity is computed and is displayed in the result area at the bottom of the **Point** dialog box. The direction of the vector is determined by the right hand rule and is displayed as an arrow for the tangent vector, normal vector, and the radius in the Pro/ENGINEER graphics window. A point analysis reports the point, curvature, normal, tangent, dihedral edge point, and radius at the selected point.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select a point or coordinate system in the collector.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Point** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create analysis feature.
 - d. If required, rename the analysis in the box adjacent to the list.

Note: A point analysis reports the point, normal, minimum curvature, and maximum curvature at the selected point in the result area at the bottom of the **Point** dialog box.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.
5. Optionally, to customize or edit your analysis, adjust the scale in the **Scale** box using the thumbwheel or type the value in the **Scale** box. You can also adjust the scale graphically using the handle on the point.

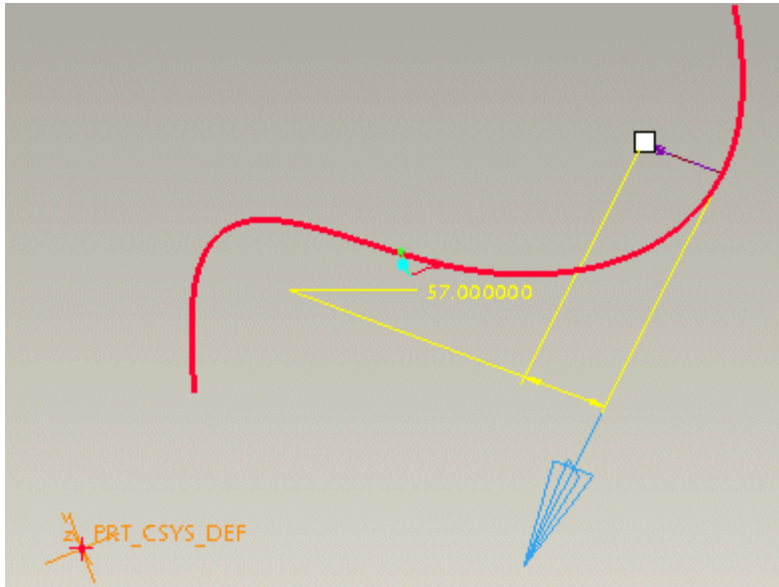
Point Information on the Edge or Curve

A normal to the surface at a selected point is a normal curvature vector. Principal curvature vectors are determined as follows: for every plane that contains the

normal vector, there is a curve of intersection of this plane with the surface. Pro/ENGINEER evaluates all the intersection curves to determine the maximum and minimum curvature value.

There can be only one curvature value if it is the same for all planes. The curvature values are called the principal curvatures of the surface at the specified point.

Curvatures can be positive or negative. If the curve turns in the direction away from the normal, it is indicated by a negative sign. The intersection of planes that contain the principal curvatures with the tangent plane defines the principal directions.






To Display Minimum Radius of a Curve or Edge

1. Click **Analysis > Geometry > Radius**. The **Radius** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector and select one or more curves or edges on the object for the analysis. The name of the selected curve or edge is displayed in the **Geometry** collector.

The minimum radius value for the selected curve or edge is computed and displayed in the result area at the bottom of the **Radius** dialog box. A vector is displayed with the minimum radius value for the selected curve or edge in the Pro/ENGINEER graphics window.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new entities in the collector to create a new analysis.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis or create an analysis feature, perform the following steps:



- a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Radius** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create analysis feature.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.
5. Optionally, to customize or edit your analysis, perform the following steps:
- a. Select one of the following types of plot in the **Plot** list:
 - **Inside & Outside**—Displays both the inside and outside minimum radius of the selected curve or edge.
 - **Inside**—Displays the inside minimum radius of the selected curve or edge.
 - **Outside**—Displays the outside minimum radius of the selected curve or edge.
 - b. Adjust the scale of plot using the thumbwheel or type a value in the **Scale** box. You can also adjust the scale graphically using the handle on the entity.

To Display the Curvature of a Curve or Edge

1. Click **Analysis > Geometry > Curvature**. The **Curvature** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector and select one or more curves or edges for the analysis. The name of the selected entity is displayed in the **Geometry** collector.


The minimum and maximum curvature of the selected curve or edge is computed. The result is displayed in the result area at the bottom of the **Curvature** dialog box. A curvature plot for the selected curve or edge is displayed in the Pro/ENGINEER graphics window.

Note:To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new entities in the collector to create a new analysis.


3. Click  to complete the analysis or  to cancel the analysis.

Alternatively, click  to start a new analysis.

4. Optionally, to save the analysis or create an analysis feature, perform the following steps.
 - a. Click the **Analysis** tab if it is not selected by default.

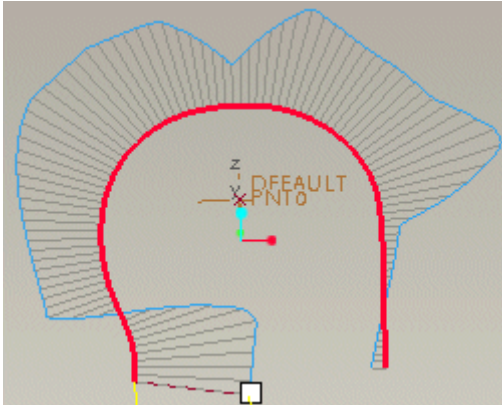
- b. Select the desired analysis types from the list at the bottom of the **Curvature** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create analysis feature.
 - d. If required, rename the analysis in the box adjacent to the list.
 - d. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.
5. Optionally, click  to display the analysis result in a separate window. The **Graphtool** window opens and the result of the analysis with the parameter values for the selected curve is graphically represented in the **Graphtool** window.
6. Optionally, to customize or edit your analysis, perform the following steps:
 - a. Select **Curvature**, **Radius**, or **Tangent** type of plot in the **Plot** box. **Curvature** is the default.
 - **Curvature**—Evaluates and plots the curvature along the selected curves or edges. For a single surface, evaluates and plots the curvature of surface in both the directions as isolines.
 - **Radius**—Evaluates and plots the radius along the selected curves and edges.
 - **Tangent**—Evaluates and plots the tangents along the selected curves and edges.

Note: You cannot create a new feature for analysis with a **Tangent** plot.
 - b. Select the **Quality**, **Number**, or **Step** type of sampling in the **Sample** box. **Quality** is the default. Adjust the sample values using the thumbwheel or slider.

Note: The number of samples must be greater than 1 and the incremental value for the step must be greater than 0.001 in model units.
 - c. Adjust the scale of plot using the thumbwheel or type a value in the **Scale** box. You can also adjust the scale graphically using the handle on the curve or edge.
 - d. Select the required plotting style.
 - e. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click  to view the result of the analysis.

Example: Curvature of a Curve or Edge

The following example shows the curvature (1/radius), including the minimum and maximum curvature values for the selected edge or curve on an object.






To Display an Offset Curve

1. Click **Analysis > Geometry > Offset**. The **Offset** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector and select a curve or a set of curves that you want to offset. The name of the selected curve or curves are displayed in the **Geometry** collector.
3. Click in the **Plane** collector and select a reference plane for the offset. The name of the selected reference plane is displayed in the **Plane** collector.

The selected curves are offset using the default offset distance and displayed in the Pro/ENGINEER graphics window.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new curve or plane in the collector to create a new analysis.

4. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
5. Optionally, to save the analysis, perform the following steps:
 - a. Select **Saved** from the list at the bottom of the **Offset** dialog box to save the analysis with the model, and to display and dynamically update the analysis while modeling.
 - b. If required, rename the analysis in the box adjacent to the list.
6. Optionally, to customize or edit your analysis, perform the following steps:
 - a. Specify the offset distance in the **Offset** box. The default offset distance is 0.0. You can graphically drag the handle to adjust the offset distance or change the value directly in the Pro/ENGINEER graphics window.

b. Select the **Quality**, **Number**, or **Step** type of sampling in the **Sample** box. **Quality** is the default. Adjust the sample values using the slider or thumbwheel.

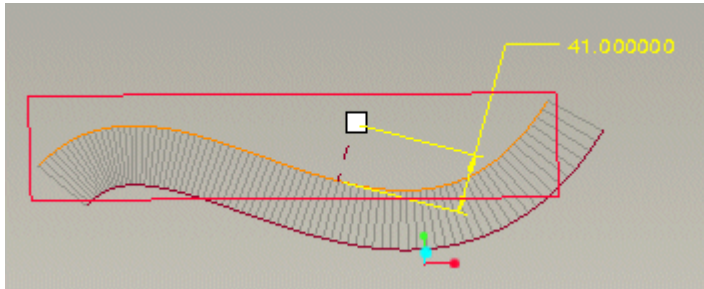
c. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click



to view the result of the analysis.

Example: Offset Curve

The following example shows an offset for the selected edge or curve on an object.



To Display Deviation of a Curve or Edge

1. Click **Analysis > Geometry > Deviation**. The **Deviation** dialog box opens. **Quick** is the default analysis type.
2. Click in the **From** collector and select a curve or edge. The name of the selected curve or edge is displayed in the **From** collector.
3. Click in the **To** collector and select a datum point, scan curve, or datum point array for which you want to measure the deviation in the **To** collector.


The minimum and maximum deviation is computed and displayed in the result area at the bottom of the **Deviation** dialog box, and the Pro/ENGINEER graphics window. The deviation plot is displayed with green and red lines where green denotes deviation within tolerance and red denotes deviation beyond tolerance.

Note:To clear the contents of the **From** and **To** collectors, right-click in the collectors and click **Remove** on the shortcut menu. You can then continue to select the **From** and **To** entities in the collectors.

4. Click to complete the analysis or to cancel the analysis. Alternatively, click to start a new analysis.

5. Optionally, to save the analysis or create an analysis feature, perform the following steps:

- a. Click the **Analysis** tab if it is not selected by default.

- b. Select the desired analysis types from the list at the bottom of the **Deviation** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create advanced features.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.
6. Optionally, to customize or edit your analysis, perform the following steps:
- a. Adjust the scale of plot using the thumbwheel or type the required scale value in the **Scale** box. You can also adjust the scale graphically using the handle on the entity.
 - b. Adjust the threshold value using the thumbwheel or type the required threshold value in the **Threshold** box. You can change the threshold value graphically by changing the value directly in the Pro/ENGINEER graphics window.
 - c. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click  to view the result of the analysis.

Analyzing Surfaces

About Analyzing Surfaces

Curve and surface analysis is used for surface modeling. A surface analysis is performed after curve analysis to check the quality of the surface. Along with the connections that it shares with its adjacent surfaces, a surface needs to be of high quality. This analysis also checks if the surface can be offset by the specified thickness value.

Surface analysis is an iterative process. After modifying or finalizing a shape, you can determine the suitability of the surface model for thickening and production. You can analyze surface properties in both Part and Assembly mode.

Use **Analysis > Geometry** and select one of the following options to perform a surface analysis:

- **Dihedral Angle**—Displays the angle between the normals of two surfaces that share an edge. This is an useful check for continuity during the evaluation of neighboring surfaces.
- **Point**—Evaluates a normal curvature vector at a datum point or a specified point on the surface. Analyzes and reports the curvature, normal, tangent, dihedral edge point, and radius at the selected point on the curve or edge. You can also specify a coordinate system.




- **Radius**—Displays the minimum radii for a surface. The radius is equal to $1/\text{curvature}$.
- **Curvature**—Evaluates and displays the curvature of surface. Mathematically, the curvature is equal to $1/\text{radius}$.
- **Offset**—Displays an offset for a selected set of surfaces.
- **Deviation**—Displays the deviation from a surface or datum plane to a datum point, curve, or datum point array to which you want to measure the deviation.
- **Sections**—Evaluates surface continuity, especially across shared boundaries.
- **Shaded Curvature**—Evaluates and displays the smallest and largest normal curvature for every point on a surface. Color values are assigned in ranges to show curvature. Values towards the red and blue ends of the spectrum indicate maximum and minimum curvatures, respectively.
- **Draft**—Analyzes a part design to determine if a draft is necessary for the part to be used in a mold. Displays a color plot of the draft.
- **Slope**—Displays, in color, the slope of a surface relative to a reference plane, coordinate system, curve, edge, or datum axis on a part.
- **Reflection**—Displays curves that represent the reflection due to linear sources of light on a surface when viewed from the specified direction. The reflection analysis is a shaded analysis. To view changes in the reflection, spin the model and observe the dynamic changes in the display.
- **Shadow**—Displays a color plot of the shadow area cast by a surface or model on another surface with reference to a datum plane, coordinate system, curve, edge, or axis.

To Display Information About a Point on a Surface

1. Click **Analysis > Geometry > Point**. The **Point** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Point** collector and select a datum point or point on a surface of the object for analysis. The name of the selected entity is displayed in the **Point** collector and the corresponding reference coordinate system is selected in the **CSYS** collector.

The normal vector for each newly selected entity is computed and is displayed in the result area at the bottom of the **Point** dialog box. The direction of the vector is determined by the right hand rule and is displayed as an arrow for the tangent vector, normal vector, and the radius in the Pro/ENGINEER graphics window.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select a point or coordinate system in the collector.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Point** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create analysis feature.
 - d. If required, rename the analysis in the box adjacent to the list.




Note: A point analysis reports the point, normal, minimum curvature, and maximum curvature at the selected point in the result area at the bottom of the **Point** dialog box.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a **Feature** type of analysis.
5. Optionally, to customize or edit your analysis, adjust the scale using the thumbwheel or type the value in the **Scale** box. You can also adjust the scale graphically using the handle on the point.

To Display Surface Radii

1. Click **Analysis > Geometry > Radius**. The **Radius** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector and select one or more surfaces or quilts on the object for analysis. The name of the selected entity is displayed in the **Geometry** collector.

The minimum inside radius and minimum outside radius for the selected surface is computed and their values are displayed in the result area at the bottom of the **Radius** dialog box. Vectors with the minimum inside radius and minimum outside radius value for the selected surface are also displayed in the Pro/ENGINEER graphics window.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select one or more surfaces or quilts in the collector.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.

4. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Radius** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create analysis feature.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a **Feature** type of analysis.
5. Optionally, to customize or edit your analysis, perform the following steps:
 - a. Select one of the following types of plot in the **Plot** list.
 - **Inside & Outside**—Displays both the inside and outside minimum radius of the selected surface.
 - **Inside**—Displays the inside minimum radius of the selected surface.
 - **Outside**—Displays the outside minimum radius of the selected surface
 - b. Adjust the scale of plot using the thumbwheel or type a value in the **Scale** box. You can also adjust the scale graphically using the handle on the plot.



To Display the Curvature of a Surface

1. Click **Analysis > Geometry > Curvature**. The **Curvature** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector and select one or more surfaces or quilts on the object for analysis. The name of the selected entity is displayed in the **Geometry** collector.

The minimum and maximum curvature of the selected surface is computed and displayed in the result area at the bottom of the **Curvature** dialog box. A curvature plot for the surface is displayed in the Pro/ENGINEER graphics window.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select the reference geometry or coordinate system in the collector.

3. Select **Spacing** in the **Sample** box and specify spacing between the mesh lines in the first and second direction. Alternately, click **Number** in the **Sample** box and specify the number of mesh lines in the first and second direction.

4. Click  to complete the analysis or  to cancel the analysis.

Alternatively, click  to start a new analysis.

5. Optionally, to save the analysis or create an analysis feature, perform the following steps:

- a. Select the desired analysis types from the list at the bottom of the **Curvature** dialog box.
- b. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling. You can select **Feature** to create analysis feature.
- c. If required, rename the analysis in the box adjacent to the list.

6. Optionally, to customize or edit your analysis, perform the following steps:

- a. Click in the **CSYS** collector and select a reference coordinate system.
- b. Select a **Curvature** or **Normal** type of plot in the **Plot** box. **Curvature** is the default.
- o **Curvature**—Evaluates and plots the curvature along the selected curves or edges. For a single surface, evaluates and plots the curvature of surface in both the directions as isolines.
- o **Normal**—Evaluates a single surface and plots the normals to surface in both the directions as isolines.
- c. Select the **Quality**, **Number**, or **Step** type of sampling in the **Sample** box to plot the curvature in the first direction. **Quality** is the default. Adjust the sample values using the slider.

Note: The number of samples must be greater than 1 and the incremental value for the step must be greater than 0.001 in model units.

- d. Adjust the sample values for the **2nd Quality**, **2nd Number**, or **2nd Step** type of sample plot and the scale in the second direction. These options are available based on the values selected in the **Sample** box.
- e. Adjust the scale of plot using the thumbwheel or type a value in the **Scale** box. You can also adjust the scale graphically using the handle on the curve or edge.
- f. Select the required plotting style.

Note: You cannot specify a plotting style for the **Normal** type of plot.

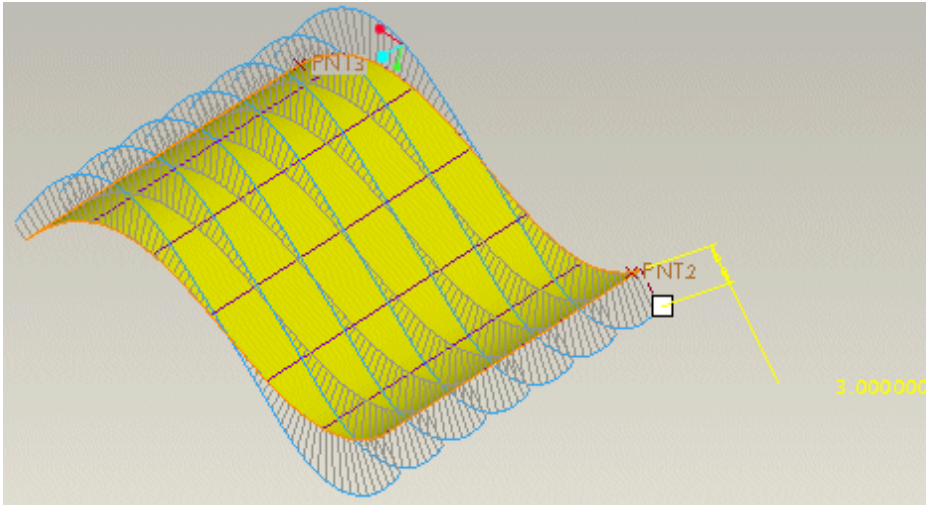
- g. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click



to view the result of the analysis.

Example: Curvature of a Surface

The following example shows curvature plot for the selected surface.






To Display Cross Sections of a Surface

1. Click **Analysis > Geometry > Sections**. The **Sections** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector and select a one or more surfaces, quilts, parts, or all model surfaces on which you want to perform a surface analysis. If you select a part, all solid surfaces in the part are analyzed. The name of the selected entities are displayed in the **Geometry** collector.
3. Click in the **Direction** collector and select a reference plane, coordinate system, a straight curve, edge, axis, or multiple planes to indicate the direction of sections. You can flip the reference to reverse the direction indicated by the arrow on the model.


The sections are computed for the **Cross** type of sections by default and displayed in the Pro/ENGINEER graphics window. The minimum and maximum curvature values for cross sections are displayed in the result area at the bottom of the **Sections** dialog box.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select the surfaces or direction references.

4. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
5. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Select **Saved** from the list at the bottom of the **Sections** dialog box to save the analysis with the model, and to display and dynamically update

the analysis while modeling. You can select **Feature** to create analysis feature.

b. If required, rename the analysis in the box adjacent to the list.

6. Optionally, click  to display the analysis result in a separate window. The **Graphtool** window opens and the result of the analysis with the parameter values for the selected surface is graphically represented in the **Graphtool** window.

Note: You can view parameter values only for the **Curvature** and **Radius** type of plots.

7. Optionally, to customize or edit your analysis, perform the following steps:
- Select **Cross** as the section type in the **Sections** box if it is not selected by default.
 - Select **Number** or **Span** as the sampling type in the **Sample** box.
 - If you select **Number**, then specify the number of sections in the **Number** box. The default value is 3.

Specify the distance between the cross sections in the **Spacing** box. The spacing value is displayed in the Pro/ENGINEER graphics window.

In the **Start** box, specify the start position of the sections with respect to the reference plane. The start and end locations of the sections are displayed as two straight lines at right angles, spanning the selection set. The start position of the sections with respect to the reference plane is displayed in the Pro/ENGINEER graphics window.

Note: The start position of the section and the spacing values can be negative.

You can graphically drag the start and end handles to adjust the spacing between sections. Drag any handle to move all sections. Drag the start or end handles and hold down the ALT key to increase or decrease the number of sections.

- If you select **Span**, you can cut sections across all selected geometry at a distance specified in the **Spacing** box. The number of sections is computed and the sections are equally spaced by the value specified in the **Spacing** box.
- Select the plot type for cross sections in the **Plot** box. You can select a **Curvature**, **Radius**, **Tangent**, or **Position** type of plot. The cross sections on the selected surface or quilt are displayed in the Pro/ENGINEER graphics window based on the plot type you select.

Curvature—Evaluates and plots the curvature of the surface in both the directions as isolines for surfaces or quilts.

Radius—Evaluates and plots the radius along the selected surfaces.

Tangent—Evaluates and plots the tangents along the selected surfaces.

Position—Evaluates surface continuity across shared boundaries and displays section curves for the selected surface.


Note: Feature is not available for **Tangent** and **Position** types of plot.

f. Select the **Quality**, **Number**, or **Step** type of sampling in the **Sample** box. **Quality** is selected by default. Adjust the sample values using the slider.

f. Adjust the scale of the plot by using the thumbwheel or type a value in the **Scale** box. You can also adjust the scale graphically by using the handle on the plot.

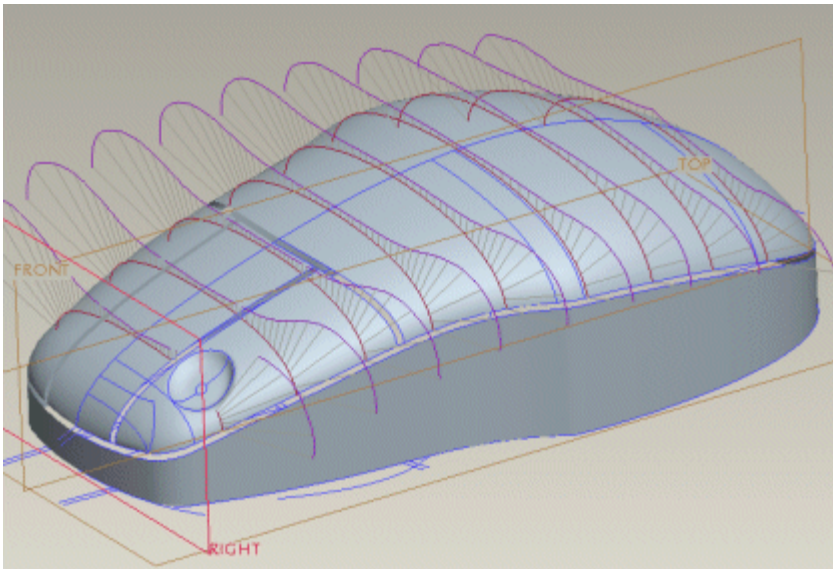
g. Select the required plotting style.

h. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and then

click  to view the result of the analysis.

Example: Section Curvature for a Surface

The following example shows the cross sections on the selected surface for Curvature type of plot.







To Display Highlight Sections of a Surface

1. Click **Analysis > Geometry > Sections**. The **Sections** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector and select a surfaces, quilts, parts, or all model surfaces on which you want to perform a surface analysis. If you select a part, all solid surfaces in the part are analyzed. The name of the selected entities are displayed in the **Geometry** collector.

3. Click in the **Direction** collector and select a reference plane, coordinate system, a straight curve, edge, axis, or multiple planes to indicate the direction of sections. The reference direction is indicated by an arrow on the model. You can flip the reference to reverse the direction indicated by the arrow on the model.
4. Select **Highlight** as the section type in the **Sections** box to provide information about surface connections.

Note: A highlight section is created where the surface normal is at a specific angle with respect to a horizontal line, that is, at the zero coordinate.

5. Specify the distance between the highlight curves in the **Spacing** box. The curves are highlighted on the surface and the spacing value is displayed in the Pro/ENGINEER graphics window. By default, the spacing is 10. You can change the spacing graphically by changing the value directly in the Pro/ENGINEER graphics window.
6. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and then click  to view the result of the analysis.

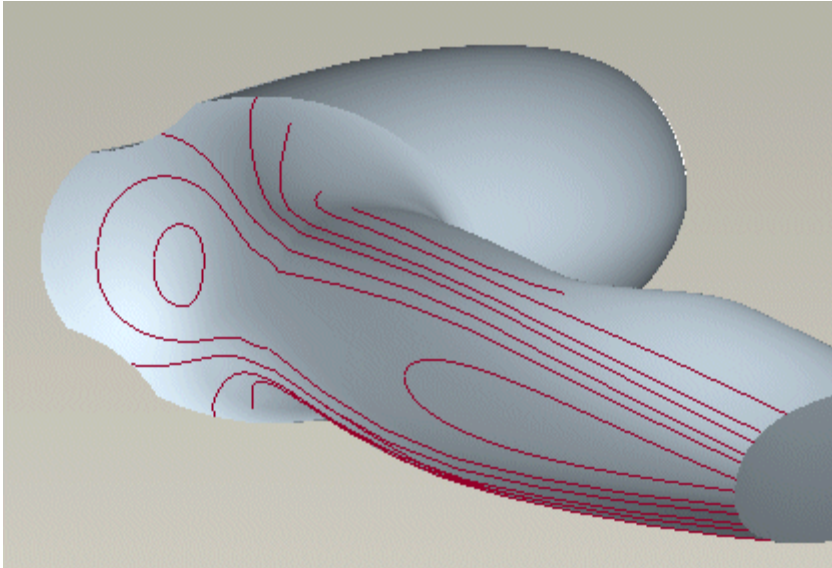
7. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.

For surfaces with tangent continuity, the highlight sections between surfaces are displayed with sharp corners. However, for surfaces that have curvature continuity, the highlight sections between surfaces are displayed as smooth curves.

8. Optionally, to save an analysis, perform the following steps:
 - a. Select **Saved** from the list at the bottom of the **Sections** dialog box to save the analysis with the model, and to display and dynamically update the analysis while modeling.
 - b. If required, rename the analysis in the box adjacent to the list.

Example: Highlight Sections of a Surface

The following example shows highlight sections on the selected surface.






To Display the Dihedral Angle for an Edge or Curve



Note: A dihedral angle is an angle between the normals of the two surfaces that share an edge.

1. Click **Analysis > Geometry > Dihedral Angle**. The **Dihedral Angle** dialog box opens. **Quick** is the default analysis type. The **Analysis** tab is selected by default.
2. Click in the **Edge** collector and select an edge, point, or datum point on an edge for dihedral angle analysis. The name of the selected entities are displayed in the **Edge** collector.

The minimum and maximum dihedral angles are displayed in the result area at the bottom of the **Dihedral Angle** dialog box and an angle plot is displayed in the Pro/ENGINEER graphics window.

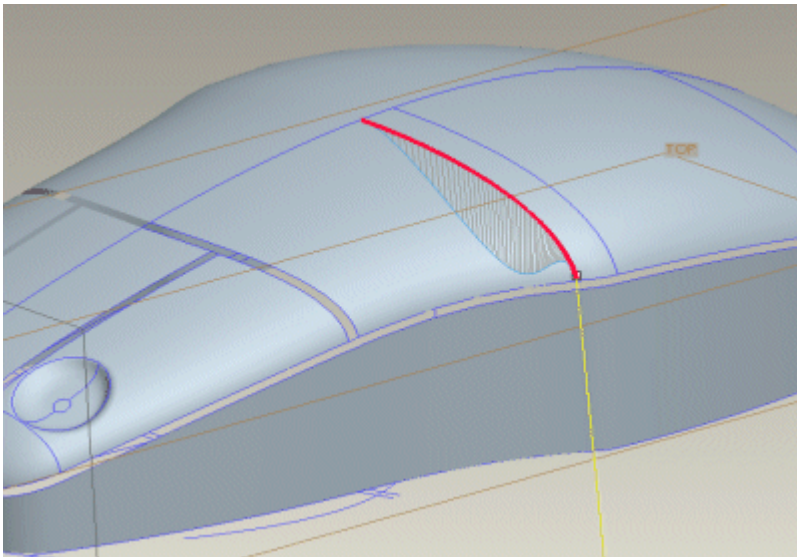
Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select a new edge to create a new analysis.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Dihedral Angle** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create analysis feature.

- d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a feature type of analysis.
5. Click  to display the analysis result in a separate window. The **Graphtool** window opens and the result of analysis with the parameter values for the selected curve or edge is represented graphically in the **Graphtool** window.
6. Optionally, to customize or edit your analysis, perform the following steps:
- a. Select the **Quality**, **Number**, or **Step** type of sampling in the **Sample** box. **Quality** is selected by default. Adjust the sample values using the slider or thumbwheel.
 - b. Adjust the scale of plot using the thumbwheel. The selected value is displayed in the **Scale** box. You can also adjust the scale graphically using the handle on the curve or edge of a surface.
 - c. Select the required plotting style.
 - d. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click  to view the result of the analysis.

Example: Dihedral Angle for an Edge or Curve

The following example displays a plot of angles between the normals of two adjoining surfaces that share a common edge:






To Display an Offset Surface

1. Click **Analysis > Geometry > Offset**. The **Offset** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Geometry** collector and select one or more surfaces or quilts that you want to offset. The name of the selected entity is displayed in the **Geometry** collector. The name of the corresponding plane is automatically displayed in the **Plane** collector.

The selected surfaces or quilts are offset using the default offset distance, mesh, plot type, and displayed in the Pro/ENGINEER graphics window. You can continue to select other surfaces or quilts.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new surface or surfaces to create a new analysis.

3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.

4. Optionally, to save the analysis, perform the following steps:


- a. Select **Saved** from the list at the bottom of the **Offset** dialog box to save the analysis with the model, and to display and dynamically update the analysis while modeling.
- b. If required, rename the analysis in the box adjacent to the list.

5. Optionally, to customize or edit your analysis, perform the following steps:

- a. Specify the offset distance in the **Offset** box. You can graphically drag the handle to adjust the offset distance or change the value directly in the Pro/ENGINEER graphics window.
- b. If you have selected one or more surfaces in the **Geometry** collector, select **Spacing** in the **Mesh** box and specify spacing between the mesh lines in the first and second direction. Alternatively, click **Number** in the **Sample** box and specify the number of mesh lines in the first and second direction.

Note: Mesh is available only if you select one or more surfaces in the **Geometry** collector. It is not available for edges or quilts.

- c. Select the **Quality**, **Number**, or **Step** type of sampling in the **Sample** box and adjust the sample values using the slider or thumbwheel.
- d. Adjust the sample values for the **2nd Quality**, **2nd Number**, or **2nd Step** type of plot sample using the slider. These options are available based on the values selected in the **Sample** box.
- e. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to view the analysis for selected options, the compute option is available.




- f. Select or adjust multiple analysis options, and click  to view the result of the analysis.

To Display the Deviation of a Surface to Points

1. Click **Analysis > Geometry > Deviation**. The **Deviation** dialog box opens. **Quick** is the default analysis type. The **Analysis** tab is selected by default.
2. Click in the **From** collector and select a surface or a datum plane. The name of the selected surface or datum plane is displayed in the **From** collector.
3. Click in the **To** collector and select a datum point, datum plane, datum curve, scan curve, or datum point array to which you want to measure the deviation. The name of the selected reference is displayed in the **To** collector.


The minimum and maximum deviation is computed and displayed in the result area at the bottom of the **Deviation** dialog box, and the Pro/ENGINEER graphics window. The deviation plot is displayed with green and red sticks where green denotes deviation within tolerance and red denotes deviation beyond tolerance.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select the **From** and **To** entities in the collectors.

4. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
5. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Deviation** dialog box.
 - c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create analysis feature.
 - d. If required, rename the analysis in the box adjacent to the list.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a **Feature** type of analysis.
6. Optionally, to customize or edit your analysis, perform the following steps:
 - a. Adjust the scale of plot using the thumbwheel or type the required scale value in the **Scale** box. You can also adjust the scale graphically using the handle on the entity.
 - b. Adjust the threshold value using the thumbwheel or type the required threshold value in the **Threshold** box. You can change the threshold value

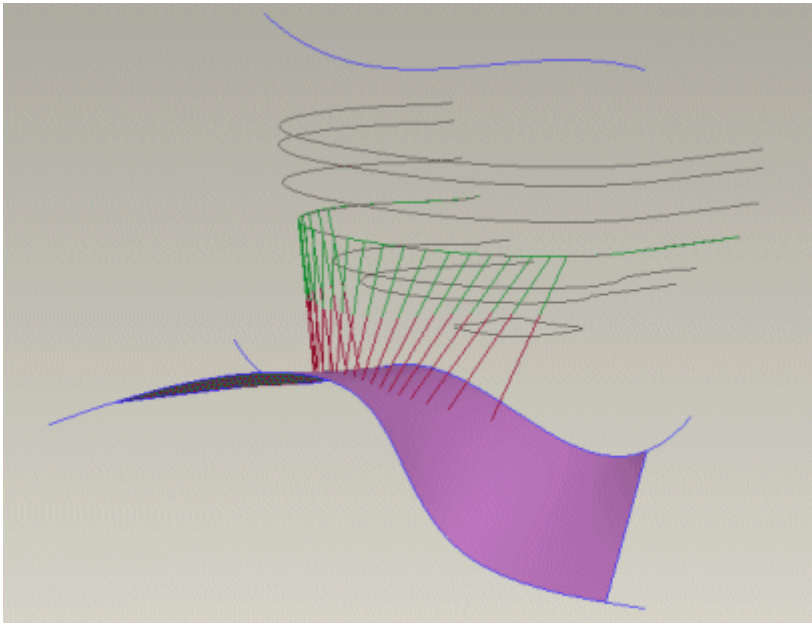
graphically by typing the value directly in the Pro/ENGINEER graphics window.

c. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and then

click  to view the result of analysis.

Example: Deviation of a Surface to Points

The following example displays minimum and maximum deviation from the surface to the selected curve.






To Display Shaded Curvature of a Surface

1. Click **Analysis > Geometry > Shaded Curvature**. The **Shaded Curvature** dialog box opens. **Quick** is the default analysis type. The **Analysis** tab is selected by default.
2. Click in the **Surface** collector and select one or more surfaces, quilts, solid geometry, or facets on which to perform the surface analysis. If you select a solid geometry, all solid surfaces in the part are analyzed. The selected entity is displayed in the **Surface** collector and the **Color Scale** dialog box opens.
3. Click in the **Plane** collector and select a reference plane. You can select a reference plane only for a **Section** type of plot.

The minimum and maximum shaded curvature values for each newly selected entity are displayed in the result area at the bottom of the **Shaded Curvature** dialog box. A shaded plot for the selected surface is displayed in the Pro/ENGINEER graphics window.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new surfaces in the **Surface** collector to create a new analysis.

4. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
5. Optionally, to save the analysis or create an analysis feature, perform the following steps:

- a. Click the **Analysis** tab if it is not selected by default.
- b. Select the desired analysis types from the list at the bottom of the **Radius** dialog box.
- c. Select **Saved** to save the analysis with the model, and to display and dynamically update the analysis while modeling or select **Feature** to create analysis feature.

Note: You cannot create an analysis feature for the **Maximum** type of plot.

- c. If required, rename the analysis in the box adjacent to the list.
 - d. Click the **Feature** tab in the **Shaded Curvature** dialog box to change the feature options of the current analysis, if required. You can access the feature options only when you select a **Feature** type of analysis.
6. Optionally, to customize your color plot, click one of the following types of color plots in the **Color Scale** dialog box:



—A rainbow type of color plot is displayed for the selected surface or surfaces.



—A gradient type of color plot is displayed for the selected surface or surfaces.



—A two-color type of color plot is displayed for the selected surface or surfaces.

7. Select one of the following types of styles for the selected surface or surfaces:



—Displays the color plot with a linear scale.



—Displays the color plot with a logarithmic scale.

Note: These styles are not available for the two-color type of color plot and when the Gaussian Curvature plot show positive and negative results.

The **Continuous** check box is selected by default.

8. Modify the number of colors for a linear or logarithmic type of plotting style. For a gradient type of color plot, you can also modify the minimum and maximum values for color gradation.

If you clear the **Continuous** check box, you can specify the number of colors for the color scale in the **Number** box and change the base value of the logarithm for a logarithmic scale used for the analysis.

Note: **Continuous** is not available for the two-color type of color plot.

9. If required, click **Default** in the **Color Scale** dialog box to restore the default values of the plot.
10. Optionally, to customize or edit your analysis, perform the following steps:

- a. Select one of the following types of plots in the **Plot** box:

Gaussian—Evaluates the curvature of a surface. Shaded curvature is the product of the smallest and largest normal curvatures for every point on a surface.

Maximum—Shows the largest normal curvature at every point on the surface.

Mean—Evaluates the continuity between surfaces.

Section—Displays the curvature of cross-sectional cuts parallel to a reference plane.

The following types of plots are available only when you are in the Reverse Engineering feature environment:

Compare Surfaces—Displays the minimum and maximum deviation between the surface and facet vertices.

Third derivative—Displays the percentage change in the curvature by using the shaded plot.

- b. Select the **Quality**, **Number**, or **Step** as the sampling type in the **Sample** box. **Quality** is selected by default. Adjust the sample values using the slider or thumbwheel.

- c. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click

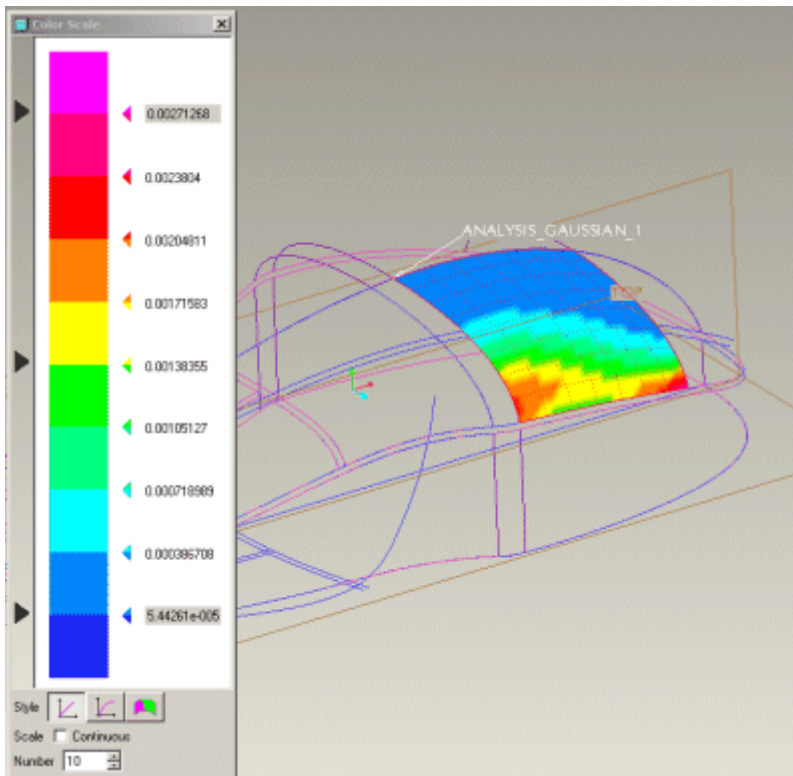


to view the result of the analysis. A shaded plot for the selected surface or surfaces is displayed in the Pro/ENGINEER graphics window.

Example: Shaded Curvature of a Surface

Surface analysis assigns color values in ranges to show curvature. Values toward the red end of the spectrum indicate maximum curvature or slope. Minimum curvature values are shown toward the blue end of the spectrum.

A Shaded curvature is the product of the smallest and largest normal curvatures for every point on a surface.






To Perform a Draft Check on a Part

1. Click **Analysis > Geometry > Draft**. The **Draft** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Surface** collector and select one or more surfaces, quilts, solid geometry, or facets on which you want to perform an analysis. The selected entity is displayed in the **Surface** collector.
3. Click in the **Direction** collector and select a reference plane, coordinate system, curve, edge, or axis to specify the pull direction. The **Color Scale** dialog box opens. You can flip the reference to reverse the direction indicated by the arrow on the model.

The draft is computed and displayed in the result area at the bottom of the **Draft** dialog box. A shaded plot for the selected surface is displayed in the Pro/ENGINEER graphics window.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select a required surface, model, or reference in the collector.

4. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.

5. Optionally, to save the analysis, perform the following steps:
 - a. Select **Saved** from the list at the bottom of the **Draft** dialog box to save the analysis with the model, and to display and dynamically update the analysis while modeling.
 - b. If required, rename the analysis in the box adjacent to the list.
6. Optionally, to customize your color plot, click one of the following in the **Color Scale** dialog box:



—A rainbow type of color plot is displayed for the selected surface or surfaces.



—A gradient type of color plot is displayed for the selected surface or surfaces.



—A two-color type of color plot is displayed for the selected surface or surfaces.




7. Click the **Continuous** check box to customize the color plot by modifying the number of colors for a linear or logarithmic type of plotting style. If you click this check box, then for the gradient type of color plot, you can also modify the minimum and maximum values for color gradation.

If you clear this check box, you can specify the number of colors for the color scale in the **Number** box. Additionally, for a gradient type of color plot, you can also modify the minimum and maximum values for color gradation.

Note: **Continuous** is not available for the two-color type of color plot.

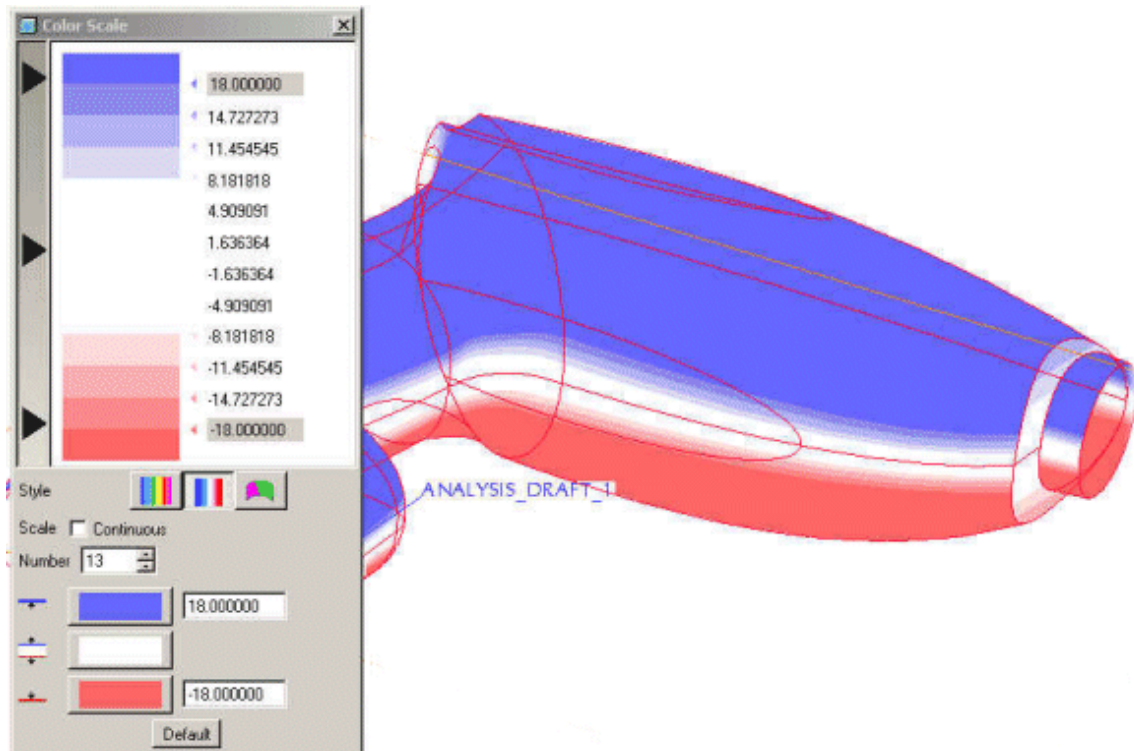
For a gradient type of color plot:

- The color plot is displayed in the Pro/ENGINEER window using the minimum and maximum display angles which by default is three times the specified draft angle value.
 - Geometry areas outside the maximum display angle are displayed in the color shown in the swatch for the maximum color gradation.
 - Geometry areas outside the minimum display angle are displayed in the color shown in the swatch for the minimum color gradation.
 - The insufficient draft geometry areas are displayed in the color shown in the undercut color swatch.
 - You can customize the minimum, maximum, and undercut color swatches using the **Color Editor** that opens when you click on the color swatches.
8. If required, click **Default** in the **Color Scale** dialog box to restore the default values of the plot.
 9. Optionally, to customize or edit your analysis, perform the following steps:

- a. If required, select the draft direction by clicking  for one direction or  both direction and specify a value for the draft angle in the **Draft** box. You can change the draft angle graphically by typing the value directly in the Pro/ENGINEER graphics window.
- b. Select the **Quality**, **Number**, or **Step** as the sampling type in the **Sample** box. **Quality** is selected by default. Adjust the sample values using the slider.
- b. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click  to view the result of the analysis.

Example: Draft Check on a Part

The following example illustrates a draft gradient for both sides where the draft angle is 6 degrees and the upper and lower angles are 18 degrees, respectively. The upper and lower color is set to the default, that is, blue and red, respectively. In the shaded analysis, the red color indicates an adequate draft for the lower half of the mold while blue indicates an adequate draft for the upper half of the mold. White indicates the areas of the mold that have an inadequate amount of draft, while the graded colors indicate areas that are within the draft limit.






To Display the Slope of a Surface

1. Click **Analysis > Geometry > Slope**. The **Slope** dialog box opens. **Quick** is the default analysis type.
2. Click in the **Surface** collector and select one or more surfaces, quilts, solid geometry, or facets on which to perform surface analysis. The selected surface, quilt, or part is displayed in the **Surface** collector.
3. Click in the **Direction** collector and select a reference plane, coordinate system, straight curve, edge, or axis. The **Color Scale** dialog box opens. You can flip the reference to reverse the direction indicated by the arrow on the model.

The slope is computed and the minimum and maximum slope values for each newly selected surface appear in the result area at the bottom of the **Slope** dialog box. A shaded plot for the selected surface is displayed in the Pro/ENGINEER graphics window.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select a required surface, model, or reference in the collector.

4. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
5. Optionally, to save the analysis or create an analysis feature, perform the following steps:
 - a. Click the **Analysis** tab if it is not selected by default.
 - b. Select the desired analysis types from the list at the bottom of the **Slope** dialog box .
 - c. Select **Saved** to display and dynamically update the analysis while modeling or click **Feature** to create analysis feature.
 - d. If required, rename the analysis.
 - e. Click the **Feature** tab to change feature options of the current analysis, if required. You can access the feature options only when you select a **Feature** type of analysis.
6. Optionally, to customize your color plot, click one of the following types of color plots in the **Color Scale** dialog box:



—A rainbow type of color plot is displayed for the selected surface or surfaces.



—A gradient type of color plot is displayed for the selected surface or surfaces.



—A two-color type of color plot is displayed for the selected surface or surfaces.

7. Select one of the following types of styles for the selected surface or surfaces:



—Displays the color plot with a linear scale.




—Displays the color plot with a logarithmic scale.

Note: These plotting styles are not available for the two-color type of color plot.

8. Click the **Continuous** check box in the **Color Scale** dialog box to customize the color plot by modifying the number of colors for a linear or logarithmic type of plotting style. If you click this check box, then for the gradient type of color plot, you can also modify the minimum and maximum values for color gradation.

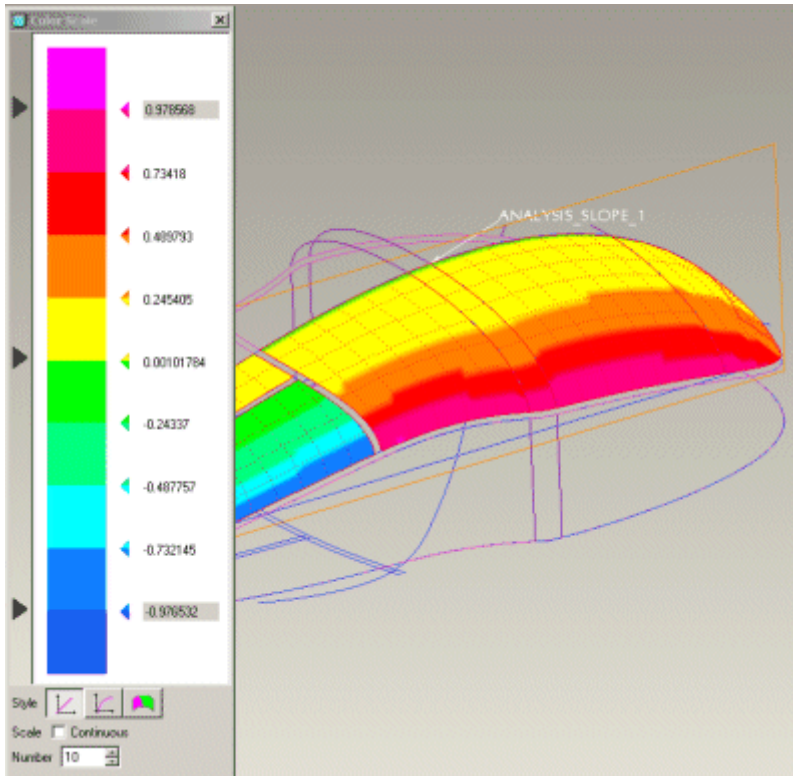
If you clear this check box, you can specify the number of colors for the color scale in the **Number** box and change the base value of the logarithm for a logarithmic scale used for analysis. Additionally, for the gradient type of color plot you can also modify the minimum and maximum values for color gradation.

Note: **Continuous** is not available for the two-color type of color plot.

9. If required, click **Default** in the **Color Scale** dialog box to restore the default values of the plot.
10. Optionally, to customize or edit your analysis, perform the following steps:
- Select the **Quality**, **Number**, or **Step** as the sampling type in the **Sample** list. **Quality** is selected by default. Adjust the sample values using the slider.
 - Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click  to view the result of the analysis.

Example: Slope of a Surface Relative to a Reference Plane

The following example displays the color plot of the surface relative to a reference plane. Values toward the red end of the spectrum indicate maximum curvature or slope. Minimum curvature values are shown toward the blue end of the spectrum.



To Display Curve Reflection




1. Select one or more surfaces, quilts, solid geometry, or facets on which to perform surface analysis.


Note: Reflection curve analysis for the earlier versions is not supported. You must create a new reflection analyses for any old reflection curve analyses that needs to be edited.

2. Click **Analysis > Geometry > Reflection**. The **Reflection** dialog box opens. **Quick** is the default analysis type.

The reflection is computed and a shaded plot for the selected surface is displayed in the Pro/ENGINEER graphics window. The result of reflection analysis is visible only in the OpenGL graphics mode.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select new surfaces, quilts, or parts to create a new analysis.

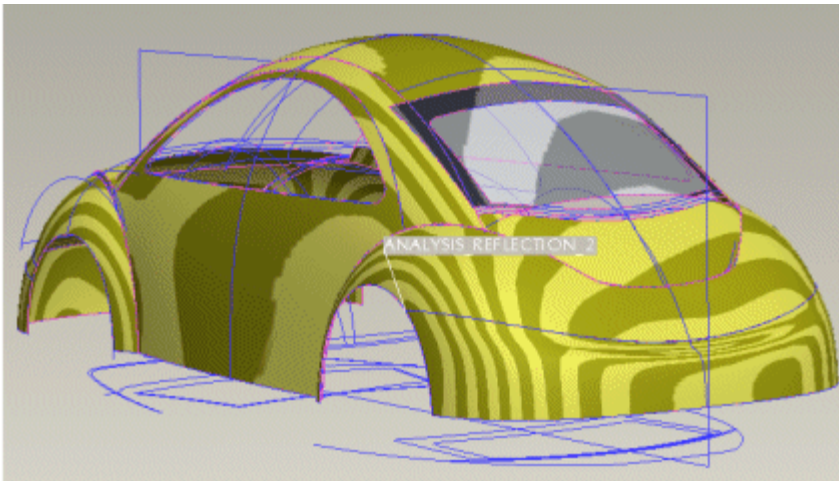
3. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
4. Optionally, to save the analysis, perform the following steps:

- a. Select **Saved** from the list at the bottom of the **Reflection** dialog box to save the analysis with the model, and to display and dynamically update the analysis while modeling.
 - b. If required, rename the analysis in the box adjacent to the list.
5. Optionally, to customize or edit your analysis, perform the following steps:
- a. Specify the number of lights in the **Lights** box. The default value is 8.
 - b. Adjust the angle of lights in the **Angle** slider. The default is 90 degrees.
 - c. Adjust the spacing between the linear source of light in the **Spacing** box, and the width in the **Width** box, as required. The default values are 10 and 5, respectively.
 - d. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to select or adjust multiple analysis options, and click  to view the result of analysis.

Note: If you edit the definition of one saved type of reflection analysis, the definitions of the other existing saved type of reflection analyses also change.

Example: Curve Reflection

The following example shows reflection curves that represent the reflection from linear sources of light on a surface when viewed from a particular direction.







To Display Shadows

1. Click **Analysis > Geometry > Shadow**. The **Shadow** dialog box opens. **Quick** is the default analysis type.
2. Select the desired analysis types from the list at the bottom of the **Shadow** dialog box.

3. Click in the **Surface** collector and select one or more surfaces, quilts, or solid geometry on which you want to perform the analysis. The selected entity is displayed in the **Surface** collector.
4. Click in the **Model** collector and select surfaces, quilts, or solid geometry. The selected entity is displayed in the **Model** collector.
5. Click in the **Direction** collector and select a reference plane, coordinate system, straight curve, edge, or axis. You can flip the reference to reverse the direction indicated by the arrow on the model.

The shadow is computed and a shaded plot is displayed in the Pro/ENGINEER graphics window. The shadow area and ratio values for the selected entity appear in the **Results** box.

Note: To clear the contents of the collector, right-click in the collector and click **Remove** on the shortcut menu. You can then continue to select a required surface, model, or reference in the collector.

6. Click  to complete the analysis or  to cancel the analysis. Alternatively, click  to start a new analysis.
7. Optionally, to save the analysis, perform the following steps:
 - a. Select **Saved** from the list at the bottom of the **Shadow** dialog box to save the analysis with the model, and to display and dynamically update the analysis while modeling.
 - b. If required, rename the analysis in the box adjacent to the list.
8. To customize or edit your analysis, perform the following steps:
 - a. Select the **Quality**, **Number**, or **Step** type of sampling in the **Sample** box. **Quality** is selected by default. Adjust the sample values using the slider.
 - b. **Update** is selected by default. You can see the effects of your selections or changes on the analysis plot as you make them. Clear the **Update** check box to view the analysis for selected options, the compute option is available.
 - c. Select or adjust multiple analysis options, and click  to view the result of the analysis.

Comparing Part or Assembly Files

About Part Comparison

In Part mode, you can compare two part files or two different versions of a part file, and obtain a graphical report on the differences between the two parts or versions. You can perform a feature comparison or a geometric comparison:

- To compare the features of two parts, use **Analysis > Compare Part > By Feature**.
- To measure the geometric deviation of one part from another part, use **Analysis > Compare Part > By Geometry**.

Note: You can compare the features of two parts in the Part mode or two components in the Assembly, Cabling, Piping , or Welding modes.

Comparing the Features of Two Parts

You can compare the features of two parts or two versions of the same part. For example, if you have two part files that contain the same basic part with the same basic features, but the dimensions of the features differ, you can use part comparison to analyze the difference in size between the two parts. A list of features that have been modified, along with any features that exist in only one of the part files, is displayed in the **Part Comparison** dialog box. Pro/ENGINEER also displays an overlay of the second part on the first part and highlights the feature being compared. The default highlight color is red.

Comparing the Geometry of Two Parts

You can measure the geometric deviation between one part and another part. Pro/ENGINEER performs an analysis that generates a point cloud of the first part, and then overlays it on and aligns it to the second part. Then Pro/ENGINEER generates a shaded color display of the deviation between the two parts and displays it in the graphics window.

To perform the geometric comparison, click **Analysis > Compare Part > By Geometry**. The **Diff Geom** dialog box opens. In the **Diff Geom** dialog box, you can control the point density (how far apart the points in the point cloud will be) by typing in a measurement spacing value. The unit of measurement in this box is always the same as the default or user-defined part measurement unit. You can also control the tolerance values for the display. That is, you can choose to display only the deviations between the parts that occur above a specific value. For example, if you are interested in a deviation range of 1.0 millimeter or more, Pro/ENGINEER displays deviations of more than 1.0 millimeter and eliminates deviations of lesser values from the display.

Note: You can choose to export the difference report displayed in the Pro/ENGINEER browser by clicking **Export Report** on the HTML report page. By default, the exported file is saved in the current working directory in the 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 exported file in the rich text format as `<model name>_CMPRO.txt` and `<model name>_CMPRO.txt.1`. You can open these saved files in Microsoft Excel.

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 latest exported file.

To Compare the Features of Two Parts



1. Open the first part.
2. Click **Analysis > Compare Part > By Feature**. The **Open** dialog box opens and lists the part files in the working directory.
3. Select the part file that you want Pro/ENGINEER to compare with the first part, and click **Open**. Pro/ENGINEER opens the second part and performs the comparison analysis.

When the comparison is finished, a list of all the features in the second part that have been modified, along with the features that are present in the second part but missing in the first part are displayed in the Pro/ENGINEER browser. The graphics window displays a model with the second part overlaid on the first part and the differences are highlighted in red.

4. If required, 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.

For User Defined Features and features that are a part of a group, the report displays the group name or the User Defined Feature name with the actual feature name in parentheses in the Item Name column. This enables you to sort the report for similar grouped features by group name and UDF name. For example, Group ABD(AE_1).

In the **Part Comparison** dialog box, you have the following choices:

Command	Action
	Select the next feature in the list and highlight the difference between the two features in the graphics window.
	Select the previous feature in the list and highlight the difference between the two features in the graphics window.
Info	Display information about the selected feature in a separate Information window.
Close	Close the dialog box and quit the comparison operation.

Note: Pro/ENGINEER does not automatically close the second part when you quit the comparison operation. You must close the window manually using **File > Close Window**.

To Measure the Geometric Deviation between Two Parts

1. Open the first part.
2. Click **Analysis > Compare Part > By Geometry**. The **Open** dialog box opens and lists the part files in the working directory.

3. Select the second part file and click **Open**. The **Geom Diff** dialog box opens.
4. In the **Measurement Spacing** box, enter the desired Measurement Spacing value. This value represents the distance between each point in the point cloud that Pro/ENGINEER generates from the first part.
5. In the **Tolerance** box, enter the desired tolerance value. This value is equal to the maximum difference in the compared geometry of the two parts, in part units. Pro/ENGINEER displays deviations that are larger than the specified tolerance value upon comparison.
6. To display only the areas that have changed from the first part to the second part, select **Show only changed areas**.
7. Click **Apply** to begin the part comparison operation. The system compares the geometry of the parts and calculates the deviation of the geometry in the second part to the geometry of the current part.

A shaded color display showing the differences between the second part and the current part is displayed in the Graphics window. The **Color Range** dialog box opens beside the color display, and displays a visual color chart containing the measurement definitions of each color.

8. To perform additional comparisons of the same two parts, you can repeat steps 5 through 8.
9. To close the dialog box and exit the part comparison mode, click **Close** in the **Geom Diff** dialog box.

To Compare the Components or Features of Two Assemblies

1. Click **Analysis > Compare Assembly by feature**. The **Comparison Options** dialog box opens.
2. Select one of the following options:
 - **Retrieve sub models**—Retrieves the submodel from the local library, renames it automatically, and displays the difference report in the Pro/ENGINEER browser window.
 - **Display sub model differences**—Displays a difference report between the two submodels in the Pro/ENGINEER browser window. Displays a difference report in the Pro/ENGINEER browser window for the two assemblies including the items that originated from the submodels.
3. Select the assembly file that you want Pro/ENGINEER to compare with the first assembly, and click **Open**. Pro/ENGINEER opens the second part and performs the comparison analysis.

When the comparison is complete, a list of all the features and submodels that have been modified in the assembly is displayed in the Pro/ENGINEER browser window.

Note: If assemblies to be compared are stored in different local directories, submodels of each assembly are retrieved separately and submodel differences are reported in the difference report. However, if the assemblies to be compared are located in the same local directory, the submodels will be identical in both the assemblies as Pro/ENGINEER does not retrieve a model that is already in session. In this case, submodel differences between the two assemblies are not reported.

The Info Menu

About the Info Menu

In Pro/ENGINEER, you can display information about selected features, parts, or the entire model in the embedded browser.

If information is displayed in text format, you can save the file with a different name and edit the file using the **File**, **Edit**, and **View** menus in the Information Window.

You can retrieve and display information for the following:

- Bill of Material
- Feature
- Component
- Model
- Reference Viewer
- Parent/Child
- Relations and Parameters
- Feature List
- Modification History
- Model Size
- Audit Trail
- Session Info

About the Info Menu

About the Display of Information

Using **Info > Component**, you can display component assembly information and information about its parents, children, relationships, and parameters.

You can display revision information about the current model using **Info > Audit Trail**. Information about the system date and time can also be displayed in the Pro/ENGINEER message area.

To Display Component Assembly Information

1. Click **Info > Component**. The **Component Constraints** and **SELECT** dialog boxes open.
2. Select the component for which you want constraint information. The name of the component appears in the **Component** box. In addition, **Type**, **CompRefs**, **Asm Refs**, and **Offset** information is displayed, if applicable.
3. Click **OK** in the **SELECT** dialog box.
4. Click **Apply** on the **Component Constraints** dialog box. A report displays information about the component in the Pro/ENGINEER browser, by default.
5. If required, sort the 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.

Alternatively, you can view the component information in an Information Window, in a file named `component-name.memb` if the `info_output_format` configuration option is set to `Text`.

If required, use the **File**, **Edit**, and **View** menus in the Information Window to save, edit, or search information.

To Display the Model Revision History

Click **Info > Audit Trail**. A report displays revision information in the Pro/ENGINEER browser. This information is also available in a file displayed in the message area. Each audit trail displays:

- **Item Type**—Lists the type of item such as feature, parameter, layer, dimension, note, and so on.
- **Item Name**—The name of the item.
- **Item Id**—The ID of the item indicating location of the change.
- **Change Type**—Selects the type of model change such as, **Metadata**, **Geometry**, **Drawing**, or **Cosmetic**.
- **Change Description**—Changes associated with the object during the creation of the model.
- **Detailed Info**—Any extra but important information in addition to the change description.
- **Time/Date**—The date and time of the most recent modification to the object in the format `mm/dd/yy hh:mm:ss`.
- **Rev**—The current revision of the object in its product database of origin.
- **Rel Level**—The current release level of the object in its product database of origin.
- **User**—The owner of the object in its product database.

- **Actions**—A list of actions that have been performed on the object, including fetching, storing, renaming, and copying.
- **Comment**—Comments associated with the object during submission, approval, rejection, and so on.

Note: The `info_output_mode` configuration option determines the means by which Pro/ENGINEER creates the audit trails.

If required, sort the 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.

To Display the Date and Time

To display the date and time in the Pro/ENGINEER message area, click **Info > Session Info > Date and Time**.

To Display a List of Messages in the Current Session

Click **Info > Session Info > Message Log**. The **Message Log** dialog box opens showing a list of messages received in your current session.

You can use the **File**, **Edit**, and **View** menus on the Information window to save the information in a file, edit the file, or view line numbers.

Getting Part and Assembly Information

About Displaying Part and Assembly Information

In Pro/ENGINEER, you can display information about selected features, including entity layers and status, using **Info > Feature**. You can also display information about entire models using **Info > Model**.

You can display feature information using an object-action workflow as follows:

- Query the **Info** menu for feature information, then
- Select the feature in the graphics window

Note: You can also display feature or part information by right-clicking the feature name in the Model Tree and using the shortcut menu (**Info > Feature** or **Info > Model**).

About Bill of Materials (BOM)

The Bill of Material (BOM) lists all parts and part parameters in the current assembly or assembly drawing. The BOM also comprises a breakdown and a summary of the report. The Breakdown section lists what is contained in the current assembly or part. The Summary section lists the total quantity of each part included in the assembly, and is the list of all the parts needed to build the assembly from the part level.

You can view a BOM in the following formats:

- BOM Text output format

BOM HTML Output Format

The default BOM HTML format provides hyperlinks to parts in the current assembly. It also provides hyperlinks that allow you to highlight, open, or see additional information about the parts in the assembly.

Breakdown Section

The HTML BOM Breakdown section lists information such as the quantity, type, name (hyperlink), and three actions (highlight, information, and open) about each member or submember of your assembly.

- **Quantity**—Lists the number of components or drawings.
- **Type**—Lists the type of the assembly component (part of subassembly).
- **Name**—Lists the assembly component and is hyperlinked to that item. Clicking this hyperlink highlights the component in the graphics window.
- **Action**—Includes the following three areas:
 - **Highlight**—Highlights the selected component in the assembly.
 - **Information**—Provides model information on the relevant component.
 - **Open**—Opens the component in another Pro/ENGINEER window.

Note: If required, sort the 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.

Summary Section

The BOM HTML Summary section lists information such as the quantity, type, name, links, and three actions (highlight, information, and open) about each member or submember of your assembly.

BOM Text Output Format

The text formatted BOM Breakdown section lists the quantity, type, and name of each member and submember of the assembly. The Summary section lists the quantity, type and name for each part included in the assembly.

BOM Configuration

You can configure BOM output format with the `bom_format` configuration file option.

You can set the `info_output_format` configuration option to `text` to specify text format for the BOM.

To Create a BOM

1. Click **Info > Bill of Materials**. The **BOM** dialog box opens.
2. Under **Select Model**, select **Top Level** or **Subassembly**.

3. If you select **Subassembly**, use the **SELECT** menu to select the subassembly in the current window.
4. Under **Include**, select or clear the **Skeletons**, **Unplaced**, and **Designated Objects** check boxes to indicate whether you want these to appear in the BOM.
5. Click **OK**. The BOM appears in a report in the Pro/ENGINEER browser, by default.
6. If required, sort the 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.

Alternatively, you can view the BOM information in an Information Window if the `info_output_format` configuration option is set to `Text`. If required, use the **File**, **Edit**, and **View** menus in the Information Window to save, edit, or search information.

To Define the BOM Format

1. Using the system editor, create the BOM output format file.
2. Add the following option to the configuration file:

```
bom_format formatname.fmt
```
3. In Pro/ENGINEER, add user-defined parameters to parts and assemblies using **Tools > Relations > Parameters > Add Parameter**.
4. Click **Info > Bill of Materials**. The BOM appears in an INFORMATION WINDOW and is written to file.

Note: Assembly members that are blanked on a layer or suppressed through assembly representation are listed in the BOM for the assembly as if they were displayed.

BOM and Mass Properties Behavior

In Skeleton Models

When working with a skeleton model in an assembly, Pro/ENGINEER generates Bill of Materials (BOM) information and mass properties information that accurately reflects the design models and either the default or user-specified mass properties. However, the assembly BOM and assembly mass properties ignore skeleton models entirely when working on parts.

In Master Representations

To obtain the full BOM or the mass properties of the Master Representation while working with a simplified representation, you must switch to the Master Representation. Pro/ENGINEER includes included components in mass property calculations because they are in session. It does not include excluded components unless they are in session. Mass properties only reflect what is currently on the screen.

The BOM lists all components of assemblies that are in session. Unless the Master Representation is in session, the BOM is not accurate. Pro/PDM provides the full BOM without retrieval of objects. In Pro/REPORT, the BOM is available only when you retrieve an assembly as the Master Representation.

For substituted objects, Pro/ENGINEER has access to the names of both the original object and the substituted object. The mass properties of the substituted component are available because the component is in session. If they have been assigned through Interchange mode, the mass properties of the original object are available in the substituted component.

To Display Feature Information

1. Click **Info > Feature**. The **SELECT** dialog box opens.
2. Click the desired feature on the Model Tree or in the model. A feature information report is displayed in the Pro/ENGINEER browser. The report lists the feature name, parents, children, feature element data, section data, dimension, and generic section data.
3. Right-click the BOM Report to add it to **Favorites** in the Pro/ENGINEER browser. Pro/ENGINEER saves the information in a file named `feature.inf`.
4. If required, sort the 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.

Alternatively, you can view the feature information in an Information Window if the `info_output_format` configuration option is set to `Text`. If required, you can save the file with a different name and edit the file using **File**, **Edit**, and **View** menus in the Information Window.

To List Model Features

Click **Info > Feature List**. A feature information report is displayed in the Pro/ENGINEER browser. The report lists the part name and features of this part.

If required, sort the 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.

For User Defined Features and features that are a part of a group, the report displays the group name or the User Defined Feature name with the actual feature name in parentheses in the Item Name column. This enables you to sort the report for similar grouped features by group name and UDF name. For example, Group ABD(AE_1).

Alternatively, you can view the model information in an Information Window if the `info_output_format` configuration option is set to `Text`. You can save the file with a different name and edit the file using the **File**, **Edit** and **View** menus in the Information Window.

To Display Detailed Model Information

1. Click **Info > Model**. If you are in Part mode, a feature information window opens in the Pro/ENGINEER browser and lists the part name and features of this part.
2. If you are in Assembly mode, the **Model Info** dialog box opens.
3. Under **Select Model**, click the option for which you want information that is one of **Top Level**, **Subassembly**, **Part**, or **Skeleton**.
4. If the **SELECT** dialog box opens, select the subassembly, part, or skeleton that you want information about and click **OK**.
5. Click **Apply** in the **Model Info** dialog box. A feature information report is displayed in the Pro/ENGINEER browser.
6. If required, sort the 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.

For User Defined Features and features that are a part of a group, the report displays the group name or the User Defined Feature name with the actual feature name in parentheses in the Item Name column. This enables you to sort the report for similar grouped features by group name and UDF name. For example, Group ABD(AE_1).

Alternatively, you can view the model information in an Information Window if the `info_output_format` configuration option is set to `Text`. This feature information is saved to a file named `part-name.inf.n`, where `part-name` is the name of the model and `n` is the version number of the file.

The information depends on what you have selected.

Command	Action
Top Level	View model information for the top-level assembly.
Subassembly	View model information for a particular subassembly.
Part	View the following model information for a part: Regular features Suppressed features Coordinate systems Cross sections Reference dimensions Incomplete features Parts are labeled if they are a generic or an instance. To use the Name option on a part, the

Command	Action
	part must have been created or retrieved during the working session.
Skeleton	View model information for a skeleton.

By default, only the names of the objects appear in the **Model Info** window. However, you can use the configuration file option `display_full_object_path` to display the full path name of the objects, along with their object-type and version-number suffixes.

To List All In-Session Models

1. Click **Info > Session Info > Object List**. An INFORMATION WINDOW opens and displays a list of objects.
2. In the INFORMATION WINDOW, you can use the following menus:
 - **File**—To save or close the list
 - **Edit**—To edit the list
 - **View**—To add line numbers to the list

To Display Model Size

1. Click **Info > Model Size**. If you are in Assembly mode, the **SELECT** dialog box opens.
2. Select the model (component) that you want size information about and click **OK**. The length of the bounding box diagonal for the assembly appears in the Message window, and the bounding box appears in green around the selected model.

If you are in Part mode, the length of the bounding box diagonal appears in the Message window and the bounding box appears in green around the selected model.

Note: To remove the bounding box, repaint or rotate the model.

Getting Manufacturing Information

About the Information Window

Click **Info** in various dialog boxes to display information generated by various Pro/ENGINEER functions in an INFORMATION WINDOW.

Sometimes you are prompted to specify whether you want to have information displayed in the window, written to a file, or both. At other times, the INFORMATION WINDOW opens as soon as you select a Pro/ENGINEER command that solicits information.

Use the menus in the Information window as follows:

- **File**—To save the contents of the window to a file
- **Edit**—To edit the contents of the window
- **View**—To display line numbers in the window

To Get Cast Model Information

1. Click **Info > Cast**. The **CAST INF** dialog box opens.
2. Under **Show Info About**, select or clear:
 - **BOM**
 - **Components**
 - **Cavity Layouts**
 - **Split Volumes**
 - **Created Volumes**
 - **Split**
 - **Last Volume**
3. Under **Output Results**, choose **Screen**, **File** or both and edit the file name if desired.
4. Click **Apply** to display the data in an INFORMATION WINDOW, write the data to a file (named in the message area), or both, depending on your selections under **Output Results**.
5. If desired, use the **File**, **Edit**, and **View** menus in the INFORMATION WINDOW to save, edit, or search, then close the **Cast Information** window.

To Get Diagram Information

1. Click **Info > Wire List**. An INFORMATION WINDOW appears and displays the following information:
 - File name for the diagram

- Cable/wire names
 - Conductor names
 - From and To component pin names
 - Spool Names
2. If desired, use the **File**, **Edit**, and **View** menus in the INFORMATION WINDOW to save, edit, or search the information, and to close the component information window.

To Get Sheetmetal Toolpath Information

1. Click **Info > Toolpath Display**.
2. Select **Tool & Path**, **Tool**, **Path**, or **None**.
3. Click **OK**. The toolpath display is highlighted on the model.

To Get Sheetmetal Design Information

1. To show information in HTML format, ensure that the `info_output_format` configuration option is set to `html`. The default is `html`.
2. Click **Info > Model**. The Sheetmetal part information is displayed in the Pro/ENGINEER browser with the following reports:
 - Used K and Y factors by Part
 - Bend Tables Associated with Part
 - Bends Containing Feature Bend Table
 - Bends Allowance
 - Bend Radii
 - Design Rules - Violations Check
3. If the the `info_output_format` configuration option is set to `Text`, click **Info > Sheetmetal**. The **Sheetmetal Info** dialog box opens.
 - a. Under **Report**, select **Bend Report**, **Radii Report**, or **Design Check**.
Note: Design Check checks the part against designated rules and displays any violations. This check requires a rule table.
 - b. Under **Output Results to**, select **Screen**, **File**, or both, and change the file name if desired.
 - c. Click **OK** to generate the information display.

To Get Harness and Cable Information

1. Click **Info > Cabling**.

2. In the **Cabling Information** dialog box, select the **Information Type**:
 - **Harness Items**—To display a list of the harness items
 - **Harness Wirelist**—To display a list of the wires in the harness that shows the cable names, color, length, and From and To locations
 - **Physical Routing**—To display another dialog box with the following buttons:
3. Select **All**, **Individual**, or **Partial** to indicate the amount of information you want.
4. Click **Apply** to display the information in an INFORMATION WINDOW. For individual and partial selection of cables in harness, select the required cable from the **HARNESSES** menu.
5. If required, use the **File**, **Edit**, and **View** menus in the INFORMATION WINDOW to save, edit, or search the information.

To Get Mold Model Information

1. Click **Info > Mold**. The **MOLD INF** dialog box opens.
2. Under **Show Info About**, select or clear the following options:
 - **BOM**
 - **Components**
 - **Cavity Layouts**
 - **Split Volumes**
 - **Created Volumes**
 - **Parting Surf**
 - **Split**
 - **Last Volume**
 - **Shrinkage**
3. Under **Output Results**, choose **Screen** or **File** or both, and edit the file name if desired.
4. Click **Apply** to display the data in an INFORMATION WINDOW, write the data to a file (named in the message area), or both, depending on your selections under **Output Results**.
 5. If desired, use the **File**, **Edit**, and **View** menus in the INFORMATION WINDOW to save, edit, or search, then close the window.

To Get Process Sequence Information

1. Click **Info > Process Sequence**.

2. An INFORMATION WINDOW opens containing information about the process.

To Get Process Setup Information

1. Click **Info > Process Step**. The **Select Step** dialog box opens.
2. Select the step about which you want to see information and click **Info**. An INFORMATION WINDOW opens containing the information.
3. Click **Close** to close the dialog box.

To Get Manufacturing Model Information

To display information about the current, active manufacturing model, from the menu bar:

1. Click **Info > Manufacturing**. The **Manufacturing Info** dialog box opens.
2. Under **Output Information for**, select one of the following options:
 - **Manufacturing Model**—To display information in the INFORMATION WINDOW about the model including:
 - Machining Sequence Report (Manufacturing Type, Stock, and Date)
 - Fixture Setup Information
 - Overall Machining Data
 - Operation List
 - **Operation**—To display list of operations in the **Operation List** box
 - **NC Sequence**—To display a list of NC sequences from either the current operation or all operation
 - **Mfg Geometry**—To display a geometry list for a selected **Geometry Type (Mill Window, Mill Surface, Mill Volume, or Turn Profile)** in the **Geometry List** box
 - **Route**—To display a Route Sheet for the assembly in an INFORMATION WINDOW showing Seq. Machine and Sequence, Rmvd Vol, Set up Time, and Run Time
3. In the **Output Results To** section of the **Manufacturing Info** dialog box, select **Screen, File**, or both and change the file name if desired.
4. Click **Apply** to generate the information, if applicable. (If no features exist to support the selection, an informational message appears.)
5. Click **Filter** to display the **Manufacturing Information: Filters** dialog box, which allows you to select or clear filters for information display.
6. Click **Close** to remove the **Manufacturing Info** dialog box.

Investigating References

About Reference Investigation

Reference investigation allows you to investigate the relationships and dependencies between features and models, enabling increased management of design intent. You can perform reference investigation through the **Reference Viewer** dialog box when working on individual parts or assemblies. Use **Info > Reference Viewer** to access the **Reference Viewer** dialog box. By using filters in the **Reference Viewer** dialog box, you can customize the contents of the **Reference Viewer** dialog box to show only dependencies of interest.

With the **Reference Viewer** you can:

- Identify the features in a model that have external or local references.
- Identify the features in a model that have a chain of dependencies from the feature to the referenced entity.
- Reroute or redefine a feature to break the dependency, if any of the external or local references are not required.
- Investigate regeneration failures by examining parent and child dependencies.
- Obtain information about the models that have external or local references to a specified model and about their relation references. Using the relations reference type as a filter, you can investigate model and feature relations to identify local or external references, or both created by relations.
- Identify references generated by relations and parameters.
- Identify circular references.
- Understand the relationships among various models, components, and features in an efficient and flexible manner.
- Break dependencies between the specified model and its dependent model or between a Data Shearing Feature and its dependent model.

The relationships are displayed graphically in the form of a tree and uses arrow shaped graph connection handles to present multiple dependencies in an easily viewable form. The graph area can display multiple levels of parents and children at one time, allowing you to see an entire assembly structure and all interdependencies among all the models.

You can investigate the following types of references, or dependencies:

- Regular references
- Placement references
- System references
- Relations references, and
- Component references.

Note:

- If the Reference Viewer dialog box is open, you cannot modify an assembly.
- If you close the Reference Viewer dialog box using File > Close and open it again in the same session, the previous reference information is retained and only changes to the assembly are updated.
- If you choose a command from a Pro/ENGINEER menu while the Reference Viewer dialog box is open, the Reference Viewer dialog box automatically closes. However, without closing the Reference Viewer dialog box you can view the information for a feature or model by clicking Info on the shortcut menu that appears when you right click a feature on the Model tree.

Feature References

Feature references are the dependencies between different features in a design. They are created in the context of a single part, or in the context of an assembly.

Feature dependencies can be local references or external references. Local references only reference geometry of the model in which they were created. External dependencies reference geometry outside the model in which they were created.

For external references, the assembly in which the two components existed when the dependency was created is another important characteristic of the dependency. For example, an assembly cut intersecting the assembly components and using their geometry is a local reference in the context of this assembly. However, if an assembly feature of a subassembly component references geometry outside the subassembly, such as a top-level part component, then this type of reference is considered external for the subassembly.

If a feature is set as current, all models that are referenced by this feature or that are referencing this feature appear in the Parent and Children trees. You can expand these models to show the features that participate in dependencies with the current feature. If a component is set as current, all parents and children of its features appear.

Models with only placement dependencies do not expand to features, because their features do not participate in the dependency. Exceptions are feature relations using external parameters and merge references.

Relations References

Relations references consist of dimensions and parameters that are used in relations. The Relations references filter setting displays objects with dependencies resulting from relations. No information about local symbols or other information about relations is given here. The **Reference Viewer** dialog box only provides investigation of references created in relations and references that were created in Program to other objects.

Relations references displays relations using parameters and dimensions from other models or features. For example, if the following symbols are used in the relations of a specified feature:

```
feat_param:FID_DTM1, surf_param:sid_surf1, model_param:5
```

the feature `DTM1`, the surface `surf1`, and the model that corresponds to session id 5 are displayed as parents of this feature when you use the Relations references setting as a filter.

As with parameters, relations can belong to various objects: assemblies, skeletons, parts, features, and patterns.

Because patterns can have relations (`P52:2 = P6:9`, for example), and accordingly, can now have parents, the Global Reference Viewer also shows patterns, as does the Model Tree. Both can be expanded to show their features.

Because a relation can be maintained without the assembly, the full path for a relation appears in the following format:

```
Relation d54:4 = d42:2 + 10 --> Cut id 496 -->
A_WHITE_BRASS_CONNECTION.PRT -->
A_CYL_END_TOP.PRT --> Protrusion id 323 --> Dimension d42:2
```

Component References

Component references are references used to place a component in an assembly. These references are created when components are placed relative to one another in an assembly. When you add a part to an assembly, the system creates a component feature in the assembly. This feature represents the part included in the subassembly, the part entities, and assembly entities used to place the part.

External placement dependencies occur when some geometry that does not belong to the subassembly or its components is used in placing a component of a subassembly. When you investigate a component reference, you may find an external reference exists because the component belongs to one subassembly but has been assembled to a component that does not belong to the same subassembly. If such an external reference is used for placement, the design intent of the subassembly is unclear outside the context of the higher level assembly in which the reference was created. Also, the subassembly will always have an external dependency to this parent assembly, making it difficult to reuse in other designs.

Local placement dependencies occur when placement uses geometry of components of the same subassembly (or their subcomponents).

When you set a subassembly as current, the system first considers it as a component and checks whether its own placement in a higher level subassembly is using some external geometry (in the context of the higher level subassembly). Then, it checks components of the current subassembly (only its own components) to see if it is using some of them for placement geometry out of the current subassembly.

External References

When investigating dependencies in an assembly, features may exist that were created in the context of another assembly. Therefore, these external dependencies point to a component in another assembly.

In the case when the other assembly is in session, the corresponding parent (or child) component has a name of the assembly next to it in parentheses, for example, PARENT.PRT (OTHERASM). Setting such an object as current replaces the current assembly tree with a filtered tree of OTHERASM.ASM. You can investigate other dependencies in the context of this assembly. When you select a previous item from the first assembly tree, you restore the tree of your initial assembly.

In the case when the other assembly is not in session, the system cannot determine which component inside this other assembly participates in an external dependency. It can only show the other assembly as the context of the dependency. You then see the following parent OTHERASM.ASM (you cannot show a child from another assembly that is not in session). If you select this parent and try to set it as current, the system tells you that you should retrieve the assembly first. If you confirm this, the **Reference Viewer** dialog box automatically retrieves it and shows its tree. The system behaves then as described above for an assembly that is in session.

About Deleting References

You can delete references in one of the following ways:

- Using the **Reference Viewer** dialog box—You can delete a reference through the **Reference Viewer** dialog box only when Pro/ENGINEER identifies it as an additional reference that can be safely removed. You can access the **Reference Viewer** dialog box using the menu bar, menu manager, or the Model Tree.
- Using the **Edit Definition** command—You can delete all references, including the references that cannot be deleted through the **Reference Viewer**, by clicking **Edit Definition** on the shortcut menu that appears when you right-click a feature on the Model Tree.

To delete a reference through the Reference Viewer using the menu bar, click **Info > Reference Viewer**, right-click the reference in the **Reference Viewer** dialog box that opens, and click **Delete Reference** on the shortcut menu.

If a feature fails for a missing reference, click **Investigate > Show Ref** on the **RESOLVE FEAT** menu that appears, right-click the reference in the **Reference Viewer** dialog box that opens, and click **Delete Reference** on the shortcut menu.

To delete references through the Reference Viewer from the Model Tree, right-click the feature on the Model Tree and click **Info > Reference Viewer**. Right-click the reference in the **Reference Viewer** dialog box that opens, and click **Delete Reference** on the shortcut menu.

You can delete missing references only for features that have the following types of references:

Feature Type	Reference Type
Round	Edge, Variable round point
Chamfer	Edge

Feature Type	Reference Type
Copy Geometry, Publish Geometry	Edge, Surface, Datum Plane, Datum Axis, Datum Point, Coordinate System, Feature
Annotation Features	Annotation plane references, User references, Leader attachment references, Surface finish reference surfaces
Thru-Point Datum Curve	Reference point of a Thru-point datum curve

Delete Reference is not available if:

- Deletion of a reference changes the structure of the feature. For example, deleting an edge in a round changes the type of round.
- You are deleting references for Thru-Point Datum Curve with Tweak option.
- A reference belongs to a logical collection such as a tangent chain, intent chain, surface loop, surface boundary and so on and the anchor is not a part of selected references.
- A reference belongs to a pattern feature, pattern instance, or leader. To delete references from a leader, suppress the pattern and delete references.
- Reference belongs to restricted annotation elements or dependent annotation elements. However, you can delete annotation plane references from dependent annotation elements if these annotation elements belong to annotation element propagation feature or data sharing feature.

The Applications Menu

About the Applications Menu

The **Applications** menu provides options for switching from one Pro/ENGINEER mode to another and to start-related applications. The modules available on the **Applications** menu vary depending on the current model type open and licenses you have.

Applications include:

Standard—Create parts, assemblies, drawings and more. Available when you launch Pro/ENGINEER.

Cabling—Create a special cabling harness part within an assembly.

Piping—Model complex 3-D piping systems based on either user-defined pipe data or project specifications.

Welding—Model welds in assemblies. Generate report tables about weld parameters and show welding symbols in assembly drawings.

Legacy—Import and update 3D data and 2D drawings in Pro/ENGINEER.

Sheetmetal—Facilitate the design of a sheet metal part.

Scantools—Focus on reverse engineering.

Mechanica—Simulate how a product will function in its intended environment enabling non-specialist design engineers to explore the mechanical performance of design alternatives without building prototypes.

Mechanism—Make a mechanism move and analyze its motion.

Animation—Animate mechanisms previously created in Mechanism Design or any assembly from Pro/ENGINEER.

Plastic Advisor—Rapidly design high quality, manufacturable, injection-molded plastic parts without the need for expertise in plastics processing.

Mold/Casting—Create mold/casting features such as Silhouette Curve, Draft Line, Draft, Tan Draft, Offset Area, 1 Sided Trim, Waterlines, and runners while in Part mode.

Mold Layout—Automate mold design beyond just creating the core and cavity assembly.

NC Post Processor—Start the PTC NC post-processor, G-POST. A post-processor is used to translate CL files generated by NC/Manufacturing into G/M codes for CNC machines.

Template—Activate **Template** mode (drawings only), allowing you to define the attributes for new drawing templates.

Conference—Start a conference. Share files and run a shared application in real time.

The Tools Menu

About the Tools Menu

The **Tools** menu includes options for customizing the Pro/ENGINEER working environment, setting external reference control options, and viewing the model creation history using the **Model Player**. It also includes options for setting configuration options (`config.pro`), and trail or training file playback. You can also choose options for creating and modifying mapkeys, and for working with floating modules and auxiliary applications.

Using the Environment Dialog Box

About Environment Settings

You can set various Pro/ENGINEER environment options using the **Environment** dialog box (**Tools > Environment**). When you make a change in the **Environment** dialog box, it takes effect only for the current session. When you start Pro/ENGINEER, the environment settings are defined by your configuration file, if any; otherwise, by the system configuration defaults.

The **Environment** dialog box has sections where you can set options for:

Setting the Display

The following table lists the options available under **Display** in the **Environment** dialog box.

Command	Action
Dimension Tolerances	Display model dimensions with tolerances.
Datum Planes	Display the datum planes and their names. Note that when you turn off the datum plane display by clearing the check box, this will not affect a datum plane that is set as a reference datum for geometric tolerances.
Datum Axes	Display the datum axes and their names. Note that in Drawing mode, when you turn the datum axis display off by clearing the check box, only the axis names are blanked.
Point Symbols	Display the datum points and their names.
Coordinate Systems	Display the coordinate systems and their names.
Spin Center	Display the spin center for the model.
Notes as Names	With the preceding 3D Notes option selected, select this box to display the names, and clear it to display note text.
Reference Designators	Make each assembly component designated as a connector in Cabling or imported as an ECAD component appear with its reference designator in 3D assembly views.
Thick Cables	Display a cable with 3D thickness. It can be shaded. This option and Centerline Cables are mutually exclusive.
Centerline Cables	Display the centerline of a cable with location points shown in green. This option and Thick Cables are mutually exclusive.
Internal Cable Portions	Display cable portions that are hidden from view by other geometry.
Colors	Display the models in colors.
Textures	Display textures on shaded models.

Command	Action
Levels of Detail	Make use of the levels of detail available in a shaded model during dynamic orientation (panning, zooming, and spinning).

Setting the Default Actions

The following table lists the options available under **Default Actions**.

Command	Action
Ring Message Bell	Ring bell (beep) after each prompt or system message.
Save Display	Save objects with their most recent screen display. This makes object retrieval in new sessions faster, because its graphics do not need to be recalculated.
Make Regen Backup	Make the system back up the current model or models to disk before every regeneration: when you explicitly choose Regenerate , as well as whenever you start a function that ends in an implicit regeneration. At the end of a successful regeneration, Pro/ENGINEER automatically deletes the backup files that it creates.
Snap to Grid	Make points you select on the screen snap to a grid. This is particularly useful in Sketcher.
Highlight Erased Views	Show a box and the view name in place of an erased drawing view.
Keep Info Datums	Control how the system treats datum planes, datum points, datum axes, and coordinate systems that you create on the fly under the Info functionality. If selected, the system includes them as features in the model. If cleared, the system erases them when you exit the Info functionality.
Use 2D Sketcher	Control the initial model orientation in Sketcher mode. If selected, the model is reoriented upon entering Sketcher to a 2-D orientation looking directly at the sketching plane. If cleared, the model orientation upon entering Sketcher is unchanged.
Sketcher Intent Manager	Use the Intent Manager when in Sketcher.
Use Fast HLR	Make possible the hardware acceleration of dynamic spinning with hidden lines, datums, and axes. Performance improvements vary among different

Command	Action
	systems and displays.
Lock View Movement	Disallow the movement of drawing views with the mouse.

You can also change **Display Style**, **Standard Orient**, and **Tangent Edges** settings using the **Environment** dialog box.

To Change Pro/ENGINEER Environment Settings

You can control many aspects of the environment in which Pro/ENGINEER runs with the **Environment** dialog box.

To open the **Environment** dialog box, click **Tools > Environment** on the menu bar or click the appropriate icon in the toolbar.

You can also change many settings by using toolbar buttons. Click **Tools > Customize Screen** to get more information and to place command buttons on the toolbar.

Tip: Ways to Change Environment Settings

You can change settings in several ways:

- You can change many of the display settings on the **Environment** dialog box by using items on the **View** menu.
- You can customize your toolbar to contain additional buttons for changing settings by using the **Customize Screen** command on the **Tools** menu.
- You can change settings for every session by editing your `config.pro` file using the **Options** dialog box, accessed through the **Options** command on the **Tools** menu.
- You can set up mapkeys as shortcuts for Pro/ENGINEER settings by using the **Mapkeys** command on the **Tools** menu, and then use the mapped keys to change settings.

Working with Mapkey Macros

About Mapkeys

In Pro/ENGINEER, a mapkey is a keyboard macro that maps frequently used command sequences to certain keyboard keys or sets of keys. The mapkeys are saved in the configuration file `mapkey`, with each macro beginning on a new line. You can define a unique key or combination of keys which, when pressed, executes the mapkey macro (for example, F8). You can create a mapkey for virtually any task you perform frequently within Pro/ENGINEER.

By adding custom mapkeys to your toolbar or menu bar, you can use mapkeys with a single mouse click or menu command and thus automate your workflow. If you are

a newer user, you can borrow mapkeys from coworkers and eliminate having to remember cryptic abbreviations.

To create a mapkey, you can use the configuration file option `mapkey`, or, on the Pro/ENGINEER menu bar, click **Tools > Mapkeys**, then in the **Mapkeys** dialog box, you click **New** and record your mapkey in the **Record Mapkey** dialog box.

Use the **Mapkeys** dialog box to:

- Define new mapkeys
- View, modify, and delete existing mapkeys
- Execute a mapkey chosen from the list
- Save the mapkeys to a configuration file

The system records your mapkey as you step through the sequence of keystrokes or command executions to define it. After you define the mapkey, Pro/ENGINEER creates a corresponding icon and places it in the **Customize** dialog box under the **Mapkeys** category. To open the **Customize** dialog box, click **Tools > Customize Screen**. On the **Toolbars** tabbed page, select the **Mapkeys** category. You can then drag the visible mapkey icon onto the Pro/ENGINEER main toolbar. You can also create a label for the new mapkey. If you choose to add the newly created mapkey icon to the toolbar, the label appears on the button added to the toolbar. When you mouse over the button, the description appears next to the mouse pointer and in the message area.

You can also nest one mapkey within another, so that one mapkey initiates another. To do so, you include the mapkey name in the sequence of commands of the mapkey you are defining.

Mapkey operations include the ability to do the following actions:

- Pause for user interaction.
- Handle message window input more flexibly.
- Run operating system scripts and commands. The **Record Mapkey** dialog box contains the **OS Script** tabbed page, whose options allow you to run OS commands instead of Pro/ENGINEER commands.

When you define a mapkey, the system automatically records a pause when you make screen selections, so that you can make new selections while the mapkey is running. Also, you can record a pause at any place in the mapkey along with a user-specified dialog prompt, which will appear at the corresponding point while the mapkey is running.

Note: If you create a new mapkey that contains actions that open and make selections from dialog boxes, then when you run the mapkey, it does not pause for user input when it opens the dialog box. To set the mapkey to pause for user input when opening dialog boxes, you must select **Pause for keyboard input** on the **Pro/E** tab in the **Record Mapkey** dialog box before you create the new mapkey.

Mapkeys Dialog Box

To open the **Mapkeys** dialog box, click **Tools > Mapkeys**. The following table defines each command on the dialog box.

Command	Action
New	Allows you to define a new mapkey. Opens the Record Mapkey dialog box.
Modify	Allows you to modify the selected mapkey.
Run	Allows you to run the selected mapkey.
Delete	Allows you to delete the selected mapkey.
Save	Allows you to save all mapkeys in the active file to a configuration file.
Changed	Allows you to save only the mapkeys changed in the current session to a configuration file.
All	Save all the mapkeys.

To Define a Mapkey

1. Click **Tools > Mapkeys** from the menu bar or click the appropriate icon on the toolbar (use **Customize Screen** on the **Tools** menu to place buttons on the toolbar). The **Mapkeys** dialog box opens.
2. Click **New**. The **Record Mapkey** dialog box opens.
3. Type the key sequence that is to be used to execute the mapkey in the **Key Sequence** text box.

Note: To use a function key, precede its name with a dollar sign (\$). For example, to map to F3, type \$F3.

4. Optional: Type the name and description of the mapkey in the appropriate text boxes.
5. Click the **Pro/E** tab. Specify how the system must handle the prompts when running the mapkey by selecting one of the following options:
 - **Record keyboard input**—(default). Record the keyboard input when defining the mapkey and use it when running the macro.
 - **Accept system defaults**—Accept the system defaults when running the macro.
 - **Pause for keyboard input**—Pause for keyboard input in the message area in response to a prompt while running the macro.

Note: The **Pause for keyboard input** option waits for you to enter a value wherever an user input is required while recording the macro.

6. Click **Record** and start recording the macro by selecting menu commands in the appropriate order.
7. If you create a new mapkey that contains actions that open and make selections from dialog boxes, when you run the mapkey, Pro/ENGINEER does not pause for user input whenever the dialog box opens. To pause for user input into dialog boxes, click **Pause** in the **Record Mapkey** dialog box to indicate when to pause while recording the mapkey. The **Resume Prompt** dialog box opens.
8. Type the prompt in the **Resume Prompt** dialog box. Then click **Resume** and proceed recording the mapkey.
9. Click **Stop** when finished recording the macro.

When you run the macro, the system pauses, displays the prompt you typed, and gives you the options to **Resume** recording the macro or to **Cancel**.

The **OS Script** tab allows you to run an Operating System (OS) script through Pro/ENGINEER using a mapkey. This is beneficial because you can start this OS script without having to minimize your window or to place it in the background.

An OS script is analogous to using aliases in UNIX. An example of an OS script is one that copies a configuration file that is commonly used from a directory on the hard disk to the working directory so that you can load the file into your session.

Defining or Editing Mapkeys

The following rules apply when you are creating mapkeys, editing mapkeys that you originally entered in the `config.pro` file or editing mapkeys that you created by using the **Mapkeys** dialog box (**Tools > Mapkeys**):

- Precede each command with a crosshatch sign (#).
- Use semicolons to separate commands or fields.
- To use a function key as a mapkey, precede it with a dollar sign (\$). For example, \$F1.
- If the first nonspace character in a field is not a crosshatch sign (#), the system interprets the rest of the field as keyboard input that you had entered in response to a prompt. However, if the current command does not request keyboard input, this data is ignored.
- If there is no text in a field, the system interprets the field as a carriage return (<CR>).
- The system ignores leading spaces.
- The system treats a series of nonleading spaces as a single space, unless it is treating the field as an input field.
- Entries are case insensitive.

- Keyboard input is case sensitive.
- There is no practical limit to the length of a mapkey. Use the backslash character (\) as a continuation character. For example, you could define the macro "aa" as follows:

```
mapkey aa #feature;#create;#protrusion; \  
#revolve; #done;
```

Nesting Mapkeys

You can nest mapkeys so that one mapkey initiates another.

Mapkey Execution

Three important rules apply for executing mapkeys:

- When you press the mapped key, if the first command is not on the screen, the system checks all the following commands in turn. When it finds one present, it executes the mapkey from the first found command.

Therefore, it is advisable to build command sequences starting from the top-level menu where appropriate.

Note: Sometimes a mapkey cannot be executed because a command specified in the mapkey is currently unavailable in the menus.

- If some of the commands ask for an input from the keyboard (for example, the name of a part), the mapkey pauses. After you enter the data, the system continues executing the mapkey.
- If one of the mapkey commands calls for you to select using the mouse (for example, selecting a sketching plane in the graphics window), the system stops the mapkey execution and exits the mapkey at that point.

In this way, you are able to create object-independent mapkeys for long sequences of repeating choices, provided you do not have to select any geometry.

Naming the Mapkeys

If you have set the configuration option for keyboard command abbreviation to `on`, use caution when naming the mapkeys. For example, if you have mapped a character sequence "cc," Pro/ENGINEER picks the command ChangeWindow from the Main menu as soon as you press the first "c." Therefore, it is advisable to start the mapped sequences with a character that is not used in the command names: slash, bracket, exclamation point, and so forth.

Native Language Support

Mapkeys are supported both for native language versions (by default) and for the English language. If you have written or recorded your macros in English and do not

want to rewrite them in your native language, set the configuration file option `native_kbd_macros` to `no`.

Example: Creating Nested Mapkeys

You can nest mapkeys so that one mapkey initiates another. To nest a mapkey, include the mapkey name preceded by a percent sign (%) in the sequence of commands of the mapkey you are defining.

The following is an example of an edited mapkey with a nested mapkey. The first line shows the mapkey to be nested, as illustrated in the second line.

```
mapkey $F1 #VIEW;#PREVIOUS;
mapkey $F2 %$F1;#VIEW;#REPAINT;
```

Note: Nested mapkeys cannot be more than five levels deep.

This method of calling existing mapkeys is different from the one the system uses when you create mapkeys using the **Mapkeys** dialog box (**Tools > Mapkeys**).

About an Incremental Mapkey

You can record an incremental mapkey by setting the `relation_text_trail_output` configuration option to `yes` and the `relation_tool_mapkey_behavior` configuration option to `increment`. You can record and run an incremental mapkey in the **Relations** dialog box only.

Note: You cannot record an incremental mapkey if you have set the `relation_text_trail_output` configuration option to `no` even if the `relation_tool_mapkey_behavior` configuration option is set to `increment`. However, you can run a mapkey that was recorded earlier with the `relation_text_trail_output` configuration option set to `yes` regardless of this option's setting when you run the mapkey.

The incremental mapkey records the insertion point in the **Relations** dialog box. The incremental mapkey records only the newly added text or action performed at the insertion point. When you run the recorded mapkey, it performs the recorded operations starting at the recorded insertion point. It adds or removes text incrementally to the current text set in the text area. The incremental mapkey does not delete or replace the pre-existing text or relation sets if it is not defined to do so.

With an incremental mapkey you can:

- Add text and relation sets in the **Relations** dialog box by running the mapkey again at the required insertion point. You may add multiple lines of text at the same time.
- Replace text with new text. You can select text and run the mapkey again to replace the selected text with the new text in the existing text set.
- Delete a specific portion of the text from a relation set in the **Relations** dialog box. You can delete the number of characters before or after that current cursor position as recorded in the mapkey.

Customizing the User Interface

To Add or Remove Toolbars

Using the **Toolbars** tabbed page in the **Customize** dialog box, you can add entire toolbars to or remove them from the Pro/ENGINEER user interface.

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens.
2. Click the **Toolbars** tab.
3. To remove a toolbar from the display, clear its check box. Click **OK** to close the dialog box.
4. To add a toolbar, select a toolbar name, for example, **Datum Display**.
5. Specify the position of the toolbar by selecting **Top**, **Left**, or **Right** in the position list (to the right of the toolbar name).

The system moves all the buttons currently in the selected toolbar to their specified positions. For example, if you specify **Right** for the **View** toolbar, the system moves the **Repaint**, **Zoom**, and other buttons in the **View** toolbar to the right side of the window.

6. To save the new settings in the `config.win` file, take one of the following actions in the **Customize** dialog box:

Click the **Automatically save to** check box (selected by default). Then, either accept the default `config.win` file name, type a new file name, or navigate to the `config.win` file in which you want to save the settings.

or

Click **File > Save Settings**. In the **Save Window Config Settings** dialog box, you can accept the default file name and path, or you can type a new file name or navigate to the `config.win` file in which you want to save the new settings. Path names can contain up to 260 characters.

7. Click **OK** to close the **Customize** dialog box and accept your changes.

About Customizing the Pro/ENGINEER User Interface

You can customize the Pro/ENGINEER user interface according to your needs or the needs of your group or company.

Using the **Customize** dialog box (**Tools > Customize Screen**), you can customize the menu bar and toolbars. By default, all commands in addition to the ones available for the active session are also displayed in the **Customize** dialog box. You can add options to the Menu Manager by using the `menu_def.pro` file.

Note: You can also change environment settings for Pro/ENGINEER using the **Environment** dialog box.

Creating Keyboard Macros, Buttons, and Button Images

You can click the **Mapkeys** command on the **Tools** menu to:

- Create keyboard macros, called Mapkeys, (copies of frequently used command sequences that are associated with defined keystrokes).
- Define text labels for the buttons if they are added to a toolbar. The label is displayed in the toolbar when the user chooses to display text in the toolbar buttons. The label is also used when you add the mapkey to a menu using the **Customize** dialog box. Click **Tools > Customize Screen** to access this dialog box.
- Enter descriptions of each macro that help you keep track of the defined macros.

Customizing the Toolbars, Toolbar Buttons, and Message Area Location

You can use the **Customize Screen** command on the **Tools** menu to:

- Add, remove, and move toolbars and toolbar buttons. Many toolbar buttons are available in addition to those that appear initially. You can also create new toolbar buttons, including split toolbar buttons.
- Create your own toolbar buttons and change the images on buttons. You can use the **Customize** dialog box to copy, delete, or edit button images.
- Add, remove, and move mapkeys and top-level commands in menu bar menus.
- Change the location of the message area.
- Choose display settings for the Model Tree.
- Save all window configuration settings in the `config.win` file.

Moving, Copying, or Removing Toolbars, Toolbar Buttons, and Menu Commands

You can change the position of a Pro/ENGINEER toolbar on the screen. You can drag a toolbar by its handle and place it in a docked position on the top, left, or right side of the main window. You cannot float a toolbar as a separate window. You can also move individual toolbar buttons to different positions on the same toolbar, or to other toolbars. You can then save the new toolbar positions as default settings in the `config.win` file. You can also move menu commands to different positions within the same menu or to other menus. You can remove toolbar buttons or menu commands from the user interface by dragging them off of the menu or toolbar area.

You can also reset a button image to its original state and set the display of menu bar or toolbar commands as text only, image only, or text with image. You cannot modify the appearance of a Menu manager command.

Editing the Configuration Files

You can load and edit configuration files to set the look and feel of the Pro/ENGINEER window (using the `config.win` file) and to specify how you want Pro/ENGINEER to work in every session (using the `config.pro` file). To edit the `config.pro` file, click

Options on the **Tools** menu. To edit the `config.win` file, use **Customize Screen** on the **Tools** menu.

Customizing the Menu Manager Options

You can customize the Menu Manager. If a system default does not exist for a particular menu, you can set your own default using a `menu_def.pro` file. You can also add options and blank them to make them unavailable in the Menu Manager.

Like the configuration files, this file is loaded during Pro/ENGINEER initialization. While loading, the system default directory is searched first, then the current working directory. If a menu default entry (`menu_def.pro`) is defined for a particular menu in the system, then any conflicting definitions in the local file are ignored.

To Add a Mapkey to the Menu Bar or Toolbars

You can add commands to toolbars or to the menus in the menu bar. The new button or command can be an existing Pro/ENGINEER command or a user-defined mapkey.

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens and the **Commands** tab is displayed by default.
2. Select **Mapkeys** in the **Categories** list.
3. Under **Mapkeys**, do one of the following tasks:
 - To add a mapkey, drag it to the desired location in the menu bar.
 - To remove a mapkey from the menu bar, drag it outside the menu bar.
4. To save the new settings in the `config.win` file, choose of the following actions in the **Customize** dialog box:
 - Select **Automatically save to** (selected by default), and then accept the default `config.win` file name, type a new file name, or navigate to the `config.win` file in which you want to save the settings.
 - Click **File > Save Settings**. In the **Save Window Config Settings** dialog box, accept the default file name and path, or type a new file name, or navigate to the `config.win` file in which you want to save the new settings. Path names can contain up to 260 characters.
5. Click **OK**.

To Add a Split Button to a Toolbar

A split button is a series of toolbar buttons that contain closely related commands and is stored in a group. Only the first command button in the series is visible. The button contains an arrow which, when clicked, displays the remaining command buttons. Rather than creating individual toolbar buttons for each command, you can create split buttons to save space or to group closely related commands. Using the following procedure, you first create a split button, and you then add commands to it. The split button displays only the first command in the series; the other commands appear when you click the arrow.

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens and the **Commands** tab is displayed by default.
2. Under **Categories**, select **New Menu**.
3. Under **Commands**, click and drag the **New Flyout** button to the desired toolbar in the Pro/ENGINEER window. As you drag, an insert indicator previews the placement position of the button. You can insert the button anywhere in an existing toolbar, but you cannot insert the button inside of another split button or as a separate toolbar. Release the mouse button to place the **New Flyout** button in the indicated position.
4. Assign commands, including mapkeys, to each new button as desired. To do so, take one of the following actions:

Select a new category in the **Categories** tree, and then under **Commands**, click and drag the desired icon onto the button.

or

In the Pro/ENGINEER window, click and drag an existing menu item or toolbar button onto the compound button.

Note: You must add a command to the button. If the button is empty, it is removed from the toolbar area when you exit the **Customize** dialog box.
5. To save the new settings in the `config.win` file, click **File > Save Settings**.
6. Click **OK**.

To Change the Appearance of Menu Bar and Toolbar Commands

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens. The **Commands** tabbed page is displayed by default.

Note: You can use the **Customize** dialog box to copy, delete, or edit button images. You can also reset a button image to its original state and set the display of menu bar or toolbar commands as text only, image only, or text with image. You cannot modify the appearance of a Menu Manager command.
2. Under **Categories**, select **Mapkeys**.
3. In the **Mapkeys** area, select a specific button and click **Modify Selection**. Select from the following options:
 - **Delete**—Delete the selected button.
 - **Copy Button Image**—Copy the image of the selected button to the clipboard.
 - **Paste Button Image**—Paste a copied button image on the selected button.
 - **Reset Button Image**—Reset the selected button to its original image.

- **Choose Button Image**—Open the **Pick Mapkey Icon** dialog box to choose from a set of created images.
- **Edit Button Image**—Open the **Button Editor** dialog box to edit the pixels in the image on the selected button.
- **Text Only**—Display text only on the selected button.
- **Image and Text**—Display image and text on the selected button.
- **Image Only**—Display the image only on the selected button (the default).

You can also select a system-supplied command and click **Modify Selection**. In this case, the only available option is **Copy Button Image**.

4. Click **OK**.

To Move a Command from One Location to Another

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens.
2. In the Pro/ENGINEER window, click and drag a toolbar button or menu item to a different location within the same or different toolbar or menu. You can also move entire menus and toolbars from one location to another. If you move a command from one menu or toolbar to another, the command is copied to the new menu or toolbar and now exists on both toolbars or menus.

To Rename a Menu Bar Menu

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens and the **Commands** tab is displayed by default.
2. In the Pro/ENGINEER menu bar, right-click the menu you want to rename, and then choose **Rename** from the shortcut menu.
3. In the **Rename** dialog box, type the new name for the menu. To place a mnemonic (shortcut key) in the new menu name, include an ampersand symbol (&) just before the letter you want to designate as the mnemonic. For example, if you want to rename the **Edit** menu to **Modify**, and you want to make the letter *o* a mnemonic, you would type `M&odify`. The new menu name would appear `Modifofy` on the menu bar, and pressing ALT+O would open the menu.
4. Click **OK**. The menu is renamed. If you designated a mnemonic, the specified letter is underlined. You can press ALT+<the letter> to open the menu rather than using the mouse.

Note: You cannot rename menu command, such as **Exit** on the **File** menu.

To Remove Commands from the User Interface

You can remove individual commands or entire toolbars or menus.

To Remove a Single Command

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens.
2. In the Pro/ENGINEER window, drag the toolbar button or menu command outside of the toolbar or menu bar. The button or command is removed.

Note: When you remove a toolbar button, make sure you drag the button outside of the toolbar area. If you drag the toolbar button to another location within *any* toolbar, the button is relocated or copied rather than removed.

To Remove an Entire Toolbar

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens.
2. Click the **Toolbars** tab, and then clear the check box next to the name of the toolbar you want to remove. The toolbar is removed from the Pro/ENGINEER window.
3. Click **OK**.

To Remove an Entire Menu from the Menu Bar

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens to the **Commands** tabbed page.
2. In the menu bar, right-click the menu you want to remove, and then choose **Delete** from the shortcut menu. The menu is removed.
3. Click **OK**.

To Change the Position of the Message Area

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens.
2. Click the **Options** tab.
3. Under **Dashboard position**, click **Above graphics area** or **Below graphics area**.
4. To save the new settings in the `config.win` file, do one of the following actions:
 - Select **Automatically save to** (selected by default), and then accept the default `config.win` file name, or enter a new file name, or navigate to the `config.win` file in which you want to save the settings.
 - Click **File > Save Settings**. In the **Save Window Config Settings** dialog box, you can accept the default file name and path, or you can type a new file name or navigate to the `config.win` file in which you want to save the new settings. Path names can contain up to 260 characters.
5. Click **OK**.

Note: The `config.pro` file option `windows_scale` lets you scale the Pro/ENGINEER windows using a coefficient.

Customize Dialog Box

You can customize the user interface with the **Customize** dialog box. To open the **Customize** dialog box, click **Tools > Customize Screen**.

You can use this dialog box to:

- Load different window configuration settings by loading a different `config.win` file.
- Specify the command buttons that appear in the window toolbars.
- Add mapkeys and other commands to the menus and toolbars, move them from one location to another location, and remove them from the menus or toolbars.
- Show or hide toolbar categories and customize the locations of toolbars on the screen.
- Show commands for all categories using the **All Commands** option under the **Categories** section.
- Specify whether the message area is located below or above the graphics window and whether icons appear next to their commands in the menu bar menus.
- Control the location of the Model Tree and Navigation tabs.
- Switch the display of icons in the top-level menus, using the **Options > Menu display > Show icons** command in the **Customize** dialog box.

Note: You can save the settings you make in the **Customize** dialog box. All settings are saved in the `config.win` file. To save the settings, select **Automatically save to** (selected by default), and then accept the default file name, or enter a new file name, or navigate to the `config.win` file in which you want to save the settings.

The following table lists the available commands in the **Customize** dialog box, along with a description of each command.

Command	Action
File > Open Settings	Opens the Open dialog box, from which you can load a different <code>config.win</code> file.
File > Save Settings	Opens the Save Window Config Settings dialog box, in which you can save your current windows configuration settings as a new, custom configuration file with a new file name.
View > Show Mode Commands Only	Lists all commands that are applicable to active model of the current session.
Toolbars tabbed page	Allows you to add, remove, or reposition the available Pro/ENGINEER toolbars
Commands	Allows you to add (and delete) new menus and custom

Command	Action
tabbed page	<p>commands (including mapkeys) to the user interface. Also allows you to edit the image and text on a toolbar button, delete commands from the menu bar menus or toolbars, and set the relative position of individual toolbar buttons.</p> <p>When you select a command name under Categories, its icon is displayed under Commands. The available commands are most helpful in the core modules, for example, Drawing, Sketcher, and Manufacturing. You can also place buttons that you have defined as mapkeys on the toolbar or on a menu bar menu.</p>
Navigation Tabs tabbed page	Allows you to set placement and size of the Navigation tabs and the Model Tree, and show History by default.
Browser tabbed page	Allows you to set the width of the Pro/ENGINEER browser, expand and animate it upon opening or closing Pro/ENGINEER.
Options tabbed page	Allows you to customize the position of the dashboard, change the size of secondary windows, and customize the menu display.
Automatically save to	Allows you to save settings automatically, in the <code>config.win</code> (or custom named <code>config.win</code>) file of your choice, each time you make changes. The advantage of this option is that a <code>config.win</code> could be easily modified and saved automatically, then read automatically when Pro/ENGINEER is started at a later time.

Customizing the Menu Manager

About Customizing the Menu Manager

Your system administrator may use a configuration file called `menu_def.pro` to customize the Menu Manager according to the needs of your workgroup. For example, the administrator may add options to the Menu Manager, blank (make unavailable) certain options, or set default option selections for the menus. The administrator would store the `menu_def.pro` file in the system default directory `<loadpoint>/text`, so that the identical Menu Manager configuration is loaded during initialization (startup) for all members of your workgroup.

You too can customize the Menu Manager for your local Pro/ENGINEER session, by creating a `menu_def.pro` file and storing it locally in your working directory. The local settings will affect only your local session and not the sessions of others in your

workgroup. At startup, the system searches for all `/menu_def.pro` files. It searches in the same three directories in which it searches for other configuration files, and in the same order—the system default directory `<loadpoint>/text`, the home directory, and the current working directory. The system then customizes the Menu Manager according to the settings in all found `menu_def.pro` files. Note that any settings in the loadpoint `menu_def.pro` file supercede similar settings in the files located in the home and working directories. For instance, if the loadpoint directory `menu_def.pro` file contains a setting for making **Feature** the default selection on the **PART** menu, and your local `menu_def.pro` contains a setting for making the **Set Up** choice the default selection on the **PART** menu, **Feature** will be the default selection.

- You can create a menu item and place it on the Menu Manager to implement a sequence of existing commands or a user-defined mapkey (macro). Adding menu options provides easy visual access to your keyboard macros or other existing commands; menu options are easier to remember and more likely to be used. You can also set up these menu items to access other commands or external software applications.

Note: If you want to add an option to the menu bar menus or to the toolbars, use **Tools > Customize**, and then use the **Customize** dialog box.

- You can set a Menu Manager menu option as the default choice when a particular menu is opened. For example, the default menu option on the **FEAT CLASS** menu is **Solid**, and it is selected (and highlighted) automatically when you choose **PART > Feature > Create**.

To Create a menu_def.pro File

1. Open the text editor of your choice (such as Microsoft Word or Notepad).
2. Create a new file, and then add the desired commands to customize the Menu Manager. You can add options, blank existing options, and set a default selected menu option.
3. Save the file to your working directory as a text file with the filename `menu_def` and the extension `.pro`. If you are working in an editor that does not allow you to use the `.pro` extension, use the extension `.txt` and rename the file using Windows Explorer or an equivalent application after the file is saved.

The system loads your `menu_def.pro` settings each time you start Pro/ENGINEER.

To Add an Option to the Menu Manager

You add menu items to the Menu Manager by using a `menu_def.pro` file. To add an item, use the **setbutton** command.

1. Open a new or existing `menu_def.pro` file.
2. Edit the file by adding a line in one of the following formats:

```
setbutton menuname menuitem "ActionDefinition" "ShortHelp"
```

or

```
setbutton menuname menuitem "ActionDefinition"
```

The elements of these definitions are as follows:

- **Menuname**—Menu title. It is usually, but not always, the same as that shown in the title box at the top of a menu. To be absolutely sure of what the `menuname` is, enable the configuration file option `button_name_in_help`. As you place the cursor over a menu option, the associated `menuname` and `menuitem` is shown in the status bar. If the title consists of more than one word, substitute # for spaces in the `menuname` (for example, `set#up` for `set up`).
- **Menuitem**—Name of the new menu option, as you want it to appear in the menu. If the option consists of more than one word, substitute # for spaces in the `menuitem` (for example, `shade#obj` for `shade obj`).
- **ActionDefinition**—Sequence of menu picks, dialog-box picks and user input to be activated by the new option. The syntax for the `ActionDefinition` is the same as that for keyboard macros. When a `setbutton` definition does not fit in one line, type `[\<CR>]` (backslash carriage-return) and continue on the following line. The total length of a `setbutton` definition must be fewer than 250 characters.
- **ShortHelp**—Text string used as one-line help for the new menu option. If omitted (when the second format is used), the default string `User defined action is used for one-line help`.

Referencing Mapkeys in menu_def.pro settings

You can also reference mapkeys in a `menu_def.pro` line with the following syntax:

```
%<mapkey name>
```

For example, if you have your `config.pro` file contains the following mapkey definition:

```
mapkey repaint ~ Activate `main_dlg_cur` `View.psh_view_repaint`
```

you can reference your `menu_def.pro` button definition using the following syntax:

```
@setbutton PART Update#Model %repaint
```

where `%repaint` is the direct reference to the mapkey.

Note: To add menu items to the menu bar or toolbars, use the **Customize** dialog box (**Tools > Customize Screen**).

Rules for Adding Options to the Menu Manager

- You cannot use `setbutton` to modify existing Pro/ENGINEER menu options (that is, menu items existing in the system menu files).
- Make sure that the user-defined item name is unique among all items on the screen at any given moment, and that all commands referenced in `ActionDefinition` are accessible at the moment of action execution. For

example, if you shade the object (see the example below) in Drawing mode, it will fail.

- When you are using a user-defined menu option in another `setbutton` definition or a keyboard macro, remember to change the # characters, if any, back to spaces.
- The maximum number of characters in `menuitem` permitted by the system is 19. You may want to use only 12 characters to fit in the standard Pro/ENGINEER menu, or modify your menu width.
- You must enter `menuitem` and `ShortHelp` exactly as they appear on the screen. The elements for `menuname` are case insensitive. `ActionDefinition` is case insensitive except for keyboard input.

Example: User-Defined Menu Manager Options

The next three examples illustrate the proper syntax for adding options, blanking options, and setting default menu options in **Menu Manager** menus.

Example 1: Adding Options

The following example is part of an actual `menu_def.pro` file. It contains a setting for adding a button to the Menu Manager that cosmetically shades a model.

```
@setbutton PART#SETUP COSMETIC#SHD \
"~ Activate `main_dlg_cur` `View.psh_view_shade`" "Temporarily Shade the Model."
```

Example 2: Including Mapkeys

The following example is from an actual `menu_def.pro` file. Note that the settings contain mapkeys in the definitions. The mapkeys are preceded by a % sign.

```
!-----
! Adding and modifying standard parameters
!-----
@setbutton PART#SETUP MOD#DPARAMS "%!mod_dparms" \
"Modify standard design parameters."
@setbutton PART#SETUP MOD#TPARAMS "%!mod_tparms" \
"Modify standard t parameters."
@setbutton ASSEM#SETUP MOD#DPARAMS "%!mod_dparms" \
"Modify standard design parameters."
@setbutton ASSEM#SETUP MOD#TPARAMS "%!mod_tparms" \
"Modify standard t parameters."
@setbutton PART#SETUP ADD#PARAMS "%!addp1;%!addp2" \
"Add standard parameters and designate them."
@setbutton ASSEM#SETUP ADD#PARAMS "%!addp1;%!addp2" \
"Add standard parameters and designate them."
!
```

Example 3: Setting Default Menu Manager Menu Options

This example illustrates two `menu_def.pro` settings for setting default menu selections in Menu Manager menus. A default menu selection is chosen and highlighted when its menu is opened.

The first line sets the option **Solid** as the default selection on the **FEAT CLASS** menu (accessed when you click **PART > Feature > Create**).

```
@setbutton FEAT#CLASS SOLID
```

The following line illustrates the syntax you would use to set the option **Protrusion** as the default selection on the **SOLID** menu (accessed by clicking **PART > Feature > Create > Solid**). It contains a setting for one-line help.

```
@setbutton #SOLID PROTRUSION;\
```

```
Create a protrusion.
```

About Setting Default Menu Options in the Menu Manager

You can set any **Menu Manager** menu command as the default choice for a menu. The specified menu command will be selected automatically whenever that menu is chosen. For example:

The default menu option in the **FEAT CLASS** menu is **Solid**, and it is selected automatically when you choose **PART > Feature > Create**. You can change the default option to any option in the **FEAT CLASS** menu, such as **Surface** or **Cosmetic**.

If there is no system default for a particular menu, you can set your own default.

Examples

In the drawing mode on selecting **Table > Repeat Region**, the **TBL REGIONS** is activated and **Simple** is selected by default.

Rules for Setting Default Menu Manager Options

When setting default Menu Manager menu options using `menu_def.pro`, be aware of the following points:

- To avoid creating serious problems consider what result setting a certain menu default may produce. Always remember what menu items the file has set as defaults. For example, setting **Create** as the default choice for the **Feature** menu results in a continuous loop of feature creation, which can only be stopped by exiting the Pro/ENGINEER session.

Setting a certain drawing size as the default for the **DWG SIZE** menu is another example of a dangerous choice. That drawing size is automatically selected whenever a drawing is created, and could easily lead to a drawing being completed on the wrong size sheet before the mistake is realized.

- The system default directory `<loadpoint>/text` is searched first, then the current working directory. If a menu default entry has already been defined for a particular menu in `menu_def.pro`, any conflicting definitions in the local file are ignored.

- You cannot set menu defaults for compound menus.
- Insert only one space between a `menuname` and the `menuitem` specified as its default.
- The entries you make in the `menu_def.pro` file are case insensitive.
- If during runtime a user-defined menu default is inaccessible, it is ignored at that moment. If the same menu name is used in the `menu_def.pro` file twice, the later entry overrides the earlier one. The system does not report any error messages for incorrect menu defaults.

Tip: Controlling Automatic Loading of a `menu_def.pro` File

The system looks for and loads a `menu_def.pro` file automatically at startup. To avoid automatic loading, you can do one of the following actions:

- Create a `menu_def.pro` file only when you are certain that you will use it in the next Pro/ENGINEER session. Rename or remove the file when the session ends.
- Keep the file in the working directory under a different file name. When you are about to begin a Pro/ENGINEER session during which the file is used, rename it to `menu_def.pro`. Rename or remove the file when the session ends.
- Keep in mind that your company system administrator may already have created a `menu_def.pro` file. In this case it is stored in the loadpoint directory and its properties can only be modified by the system administrator.

To Set a Default Menu Manager Menu Option

1. Create or open an existing local `menu_def.pro` file.
2. Add to the file a line in the following format: `menuname menuitem`

where

- `menuname`—Title as shown at the top of a menu. If the title consists of more than one word, spaces between the words must be changed to # (for example, the `menuname` for menu **DIM PNT PNT** would be entered as `dim#pnt#pnt`).
- `menuitem`—Option in the menu `menuname`. If the option consists of more than one word, you must change spaces between the words to # (for example, enter the `menuitem` `Edge Tangent` as `edge#tangent`).

Remember to include only one space between the `menuname` and the `menuitem`.

The entries you make in the `menu_def.pro` file are case insensitive.

3. Save the file. The new settings will take effect the next time you start Pro/ENGINEER.

Note: Any settings in a `menu_def.pro` file created and stored in the loadpoint directory supercede your local `menu_def.pro` file settings.

Using Configuration Files

About Configuration Files

You can customize the look and feel of Pro/ENGINEER and the way in which Pro/ENGINEER runs by setting options in a configuration file. Pro/ENGINEER contains two important configuration files: `config.pro` and `config.win`. The `config.pro` file is a text file that stores all of the settings that define the way in which Pro/ENGINEER handles operations. The `config.win` file is a database file that stores window configuration settings, such as toolbar visibility settings and Model Tree location settings. Each setting in the configuration files is called a configuration option. Pro/ENGINEER provides default values for each option. You can set or change the configuration options. Some of the options you can set are:

- Tolerance display formats
- Calculation accuracy
- The number of digits used in sketcher dimensions
- Toolbar contents
- Relative button order on a toolbar
- Location and size of the Model Tree

Note: `Config.sup` is a protected system configuration file. Your company system administrator uses this file to set configuration options that are used on a company-wide basis. Any values that are set in this file cannot be overridden by other (more local) `config.pro` files.

About menu_def.pro

The `menu_def.pro` file is a configuration file that controls the look and feel of the Menu Manager. Typically, your company system administrator would set up this file in order to customize the Menu Manager according to the needs of your workgroup. You can also create your own `menu_def.pro` file and further customize the Menu Manager locally.

How Pro/ENGINEER Reads Configuration Files at Startup

Pro/ENGINEER reads configuration files automatically from several areas. If a particular option is present in more than one configuration file, Pro/ENGINEER applies the most recently loaded/read setting.

At startup, Pro/ENGINEER first reads in a protected system configuration file called `config.sup`. It then searches for and reads in configuration files (`config.pro`, `config.win`, `menu_def.pro`) from the following directories in the following order:

1. `loadpoint/text` (`loadpoint` is the Pro/ENGINEER installation directory) – Your system administrator may have put configuration files in this location to support company standards for windows configuration settings, formats and libraries. Any user starting Pro/ENGINEER from this `loadpoint` uses the values in this file.

2. Login directory – This is the home directory for your login ID. Placing your configuration files here lets you start Pro/ENGINEER from any directory without having a copy of the files in each directory.
3. Startup directory – This is your current or working directory when you start Pro/ENGINEER.

Note: The local configuration files (`config.pro`, `config.win`, and `menu_def.pro`) in your startup directory are the last to be read; therefore, they override any conflicting configuration file option entries. The `config.pro` file does not, however, override any `config.sup` entries.

You can create and store custom configuration files in your current working directory. These custom configuration files usually apply to specific projects.

About `config.pro` Options

Options in the `config.pro` file generally use the following format:

```
config_option_name value
```

Where `config_option_name` is the option that you want to set, and `value` determines the setting for that option.

For example, the configuration file option `prompt_on_exit` tells Pro/ENGINEER whether to prompt you to save your working files when you click **File > Exit**. By default, Pro/ENGINEER exits without a prompt. To change this default behavior, you can add the following line to your configuration file:

```
prompt_on_exit yes
```

Pro/ENGINEER reads this configuration file option when it starts and subsequently prompts you to save any modified files before exiting.

Setting `config.pro` Options

Each configuration file option in `config.pro` contains a default value that is set by Pro/ENGINEER. If you do not change an option, Pro/ENGINEER uses the default value.

There are two methods to change a `config.pro` option:

- Click **Tools > Options** and use the **Options** dialog box.
- Open the `config.pro` file from within a text editor such as Notepad or Microsoft Word, and add or change configuration options directly.

In general, you should change `config.pro` file settings before you start a Pro/ENGINEER session. If you want to change the environment during a session, it is usually more convenient to use the **Environment** dialog box, accessed by clicking **Tools > Environment**. However, you can change some `config.pro` options only by editing the file directly, by using one of the two methods mentioned previously.

You can save multiple versions of the `config.pro` file, with each file containing unique settings. This approach is useful if you want to use different configuration settings for different models, and you do not want to change multiple options for the

same `config.pro` file as needed. You save different versions by using the **Options** dialog box.

Setting `config.win` Options

You can make window configuration changes to Pro/ENGINEER only during an active session. You can make changes using only the **Customize** dialog box. Open the **Customize** dialog box by clicking **Tools > Customize Screen**.

As with the `config.pro`, the `config.win` file uses a hierarchical approach to configuration file settings. First, it looks in the `loadpoint/text` and home directories, then at the local directory. The settings are reflected as it moves down the hierarchy.

You can retain different window configurations by saving multiple versions of `config.win` and loading the desired configuration file during a session, rather than by changing a single `config.win` file every time your window configuration needs to change.

To Save `config.win` Changes Automatically

All changes in the Pro/ENGINEER user interface are saved in the `config.win` file located in your current working directory.

1. Click **Tools > Customize Screen**. The **Customize** dialog opens.
2. Select **Automatically Save To** if it is not already selected.
3. In the **Automatically Save To** box, type the name of the custom `config.win` file in which you want to store the current settings, or accept the default file.
4. Click **OK** to close the **Customize** dialog box and accept your changes.

Note: If you want to see the full path of your current working directory, do the following:

1. Start Pro/ENGINEER in an empty directory.
2. Open the **Customize** dialog box (click **Tools > Customize Screen**).

At the bottom of the **Customize** dialog box, the full path name of your current working directory is displayed.

Note: Path names can contain up to 260 characters.

To Load a `config.win` File

Pro/ENGINEER loads the default `config.win` file at startup if it does not find any files named `config.win`. To load a file named other than `config.win` or located in one of the non-default search locations, you must load custom configuration files each time you start Pro/ENGINEER and open the project to which the configuration file(s) apply. Use the following procedure:

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens.
2. Click **File > Open Settings**.

3. In the **Open** dialog box, navigate to and select the windows configuration file you want to open. Then click **Open**.
4. Click **OK**. The new `config.win` settings are loaded and visible.

To Set config.pro Options

1. Click **Tools > Options**. The **Options** dialog box opens. Settings from the most recently loaded `config.pro` file are displayed. The icon next to the option shows whether the edit is applied immediately, applied to the next instance of an object created, or applied to the next session.
2. Locate the option you want to set. When you highlight an option in the list, it appears in the line labeled **Option:** in the lower part of the dialog box.
3. In the **Value** text box, enter the new value. If the value entries are fixed, for example *Yes* or *No*, you can use the drop-down list at the bottom of the dialog to see what values are available. If the value requires an integer, type it in. If the value requires a path or file, type it in or use the Browse option to locate it.
4. When you have edited the value, click **Add/Change**. The new value is applied in the options list window. The status icon for the affected line changes to show that an edit to the default has been made.
5. When you have finished making edits to the config option values, click **Apply**, or **OK**. Changes that can be applied immediately appear in the user interface.

Additionally, when you edit a value during the current session, a configuration file called `current_session.pro` is automatically created in the working directory. This file contains only the configuration options you changed with their new settings. If the file already existed, it is edited or appended with the new settings.

Using the Options Dialog Box

Use the **Options** dialog box to work with Pro/ENGINEER configuration files in the following ways:

- Load a different configuration file (the default is `config.pro`).
- Modify values for configuration options in Pro/Engineer configuration files, including `config.pro` files and other files such as drawing setup files (available in Drawing mode only).
- Toggle the display of all configuration options and values, or display only those options that are off-default and loaded locally.
- Determine the source configuration file for each configuration option displayed.
- Save copies of configuration files with custom settings for current sessions.

To access the **Options** dialog box, click **Tools > Options**.

The **Showing** list box displays the last `config.pro` file read, and the left pane shows the cumulative off-default settings that have been read from any `config.pro` files

found in the search path while Pro/ENGINEER is loading. The value, status, and source are shown for each option. The status is either *applied* (solid dot) or *conflicting values* (crosshatched dot). Conflicting values means that two configuration files have differing values for the same option. In this case the system default setting is used until you reset the value for the current session.

The source is defined as the configuration file and path that the option and value come from. You can use the list to show only the configuration options from each of the sources.

When Current Session is displayed in the **Showing** box, uncheck the **Show Only Options** check box to show all options and values referenced during the current Pro/ENGINEER session. When the box is checked, only those options and values loaded from `config.pro` files are displayed.

Application Effective Icons

The icon to the left of each option indicates whether a change is applied immediately, or on the next startup. The lightning icon means immediate effect. The wand icon means the change is applied to the next object created. The screen icon means the change is applied in the next session.

Editing Options and Values

When you make edits to options or values, the edits are saved to the file currently displayed in the **Showing** menu when you click **Apply**. If Current Session is displayed, the changes are saved to a file named `current_session.pro`, which is automatically created in the current working directory.

To edit a value, select it in the list window. The option appears in the Option: text field. The value appears in the Value: field. If the value is fixed, for example `Yes` or `No`, use the drop down list box to select from the choices for the option. If the value requires an integer, type it in.

To reset a value to the system default, select the value and click **Delete**. When the option and value are deleted, the system returns to referencing the default values.

Navigating the Options

- Use the **Sort** list box to sort the options alphabetically or by category.
- Use the **Find** button to open the **Find Option** dialog box and search for options using a text string and wildcard characters. For example, if you type `'ecad'` in the Type keyword field, the dialog box lists all options using `ecad` in their strings. You can also use keywords and wildcard characters to search the descriptions. You can change values for the selected configuration options from within the **Find Option** dialog box.

Note: You must still click **Apply** in the **Options** dialog box to save changes to the session.

To Search for a Configuration Option

You can use the **Find Option** dialog box to search for configuration options by typing a keyword and wildcard characters.

1. To open the **Find Option** dialog box, click the **Find** button in the **Options** dialog box.
2. Under **Type keyword**, type a string that exists within the option you want to find. You can use wildcard characters to assist you in your search. For example:
 - If you type the letter `a` with no wildcard character, the search returns all of the configuration options that contain the letter `a`. This is the most convenient way to find all options that contain a certain word.
 - If you type the letter `a` and an asterisk `a*`, the system lists all of the configuration options that begin with the letter `a`.
 - If you type an asterisk and the letter `a*`, the system lists all of the configuration options that end with the letter `a`.

3. Click **Find Now** to perform the search.

Under **Choose Option**, the system lists all of the configuration options that contain the specified string (keyword).

4. Select a configuration option under **Choose Option**.
5. To set a value for the specified option, use the **Set Value** box.
6. Click **Add/Change** to update the configuration option display, and then click **Close** to close the **Find Option** dialog box.
7. In the **Options** dialog box, Click **Apply** to apply your changes. If you do not click **Apply**, your changes are lost. The system updates the setting in the configuration option.
8. Click **Close** to close the **Options** dialog box.

Working with Licenses



About Pro/ENGINEER Licenses

In Pro/ENGINEER, there are two types of licenses:

- **Locked Licenses**—A locked license allows Pro/ENGINEER to run only on the machine for which the codes were generated. On UNIX systems, the **DISPLAY** setting of Pro/ENGINEER is also forced to the machine for which the codes were generated. With this type of license, a license server is not required.
- **Floating Licenses**—A floating license allows Pro/ENGINEER to run on any machine on your network, provided a machine can communicate with the license server. The license server keeps track of how many licenses are checked out and will grant or deny licenses to clients based on license availability

To List, Get, or Release a Floating Module License

To obtain the floating module licenses, click **Tools > Floating Modules**. The **Floating Modules** dialog box opens, and lists the floating modules.

Command	Action
	Select all the modules.
	Cancel the selection of all the modules.

Using Auxiliary Applications

About Auxiliary Applications

The Auxiliary Applications command on the **Tools** menu is used to manage Pro/TOOLKIT and J-Link applications. It is an entry point for these external applications you create for use in Pro/ENGINEER.

The Auxiliary Applications command opens the **Auxiliary Applications** dialog box. It allows you to

- Search for registry files (.dat files) in order to register external applications.
- List all available external applications.
- Start, stop, or delete registered applications.

For more information about Pro/TOOLKIT or J-Link, refer to the Pro/TOOLKIT and J-Link online documentation.

To Work with Auxiliary Applications

To start, stop, list, register, or delete an auxiliary application:

1. Click **Tools > Auxiliary Applications**. The **Auxiliary Applications** dialog box opens, listing the existing auxiliary applications and their status.
2. Click one of the following:
 - **Register**—Retrieve a file by displaying the **Register auxiliary application** dialog box.
 - **Start**—Start a selected application
 - **Stop**—Stop a selected application.
 - **Delete**—Remove a selected application.
 - **Info**—Display information about any auxiliary applications in the Information window.

Using Pro/Web.Link

About Pro/Web.Link

Pro/Web.Link enables you to build custom Web-based applications that interact with Pro/ENGINEER through the Netscape Web browser. Pro/Web.Link extends the capabilities of this browser so it can recognize and execute JavaScript commands that communicate with a Pro/ENGINEER session.

During a Pro/ENGINEER session, you set up the access control for Pro/Web.Link with the **Pro/Web.Link Security** dialog box (**Tools > Pro/Web.Link > Set Access Permissions**).

To Set Access Permissions for Pro/Web.Link

Pro/Web.Link links the World Wide Web to Pro/ENGINEER, enabling you to use the Web as a tool to automate and streamline parts of your engineering process.

To set access permissions for Pro/Web.Link:

1. Click **Tools > Pro/Web.Link > Set Access Permissions**. The **Pro/Web.Link Security** dialog box opens.
2. Check any of the following actions you want the Web pages support:
 - **Read from Pro/ENGINEER Session**
 - **Make changes in Pro/ENGINEER Session**
 - **Read from File System**
 - **Write to File System**

Prompt as needed for new permissions (if autoprompting is turned on and you call a function that requires a security level beyond the current settings).

3. Click **OK**.

To Remove a Pro/Web.Link License

When you remove the ownership of unused Pro/Web.Link licenses (because of a program crash, power failure, and so on), click **Tools > Pro/Web.Link > Release License**.

Working with Trail and Training Files

About Trail and Training Files

Trail Files

A trail file is a record of all menu choices, dialog-box choices, selections, and keyboard entries for a particular working session. Trail files allow you to view the

record of activity so that you can reconstruct a previous working session or recover from an abrupt termination of a session. Trail files are editable text (.txt) files.

When you run a trail file, the system replays the selections in the exact order in which they were originally made.

Note: Before running a trail file, you must rename it. You cannot rename a trail file within Pro/ENGINEER.

Training Files

Training files are customized trail files that can be used to help train Pro/ENGINEER users. Unlike trail files, training files contain directives that provide the user with additional information to help guide them through sample exercises or operations in Pro/ENGINEER. These directives can include pauses and textual comments that are displayed in an INFORMATION WINDOW. Training files use the extension .txa. You can train yourself to use Pro/ENGINEER by creating training files. A training file is a trail file that contains information that guides you through a sample exercise or operation in Pro/ENGINEER. When you run a training file, the comments that you inserted are displayed in an INFORMATION WINDOW.

Trainer Text Files

The trainer text file contains the text that is displayed while you use the training file. Each training file can reference a single trainer text file. The training text file contains text blocks that are displayed in a text window during execution of a training file. The text blocks are used to describe what is happening in the system. They also provide clues for performing an operation and present data that may not appear graphically.

The trainer text file is divided in sections. The format for each section is as follows:

```
#begin number
text
#end
```

In the trainer text file format, text is the text to be displayed on the screen at a particular point in the training file.

To Run a Trail or Training File

1. Rename the trail file before running it.

Note: You cannot rename a trail file from within Pro/ENGINEER. On a Windows NT system, use Explorer, and on a UNIX system, use the terminal window to either move or copy the file to the new name. Assign the extension .txa to the training file.

2. In Pro/ENGINEER, click **Tools > Play Trail/Training File** or click  in the toolbar (click **Tools > Customize Screen** to place buttons on the toolbar). The **Open** dialog box opens.

3. Select a trail or training file, and then click **Open**.

To Create a Training File

1. Open an existing trail file within an external text editor such as Microsoft Word, and then rename the file with the extension `.txa`.
2. Open the new training (`.txa`) file within any text editing application, such as Microsoft Word or Notepad.
3. Add directives to the file.
4. Save the file when you are finished adding directives.

To Pause a Training File During Execution

To set a training file to pause after each line in the file, set the configuration option `set_trail_single_step` to `yes`.

To set a training file to pause after only certain lines, include the following syntax in the directive of the line at which you want the file to pause:

```
`PAUSE
```

Directives for Creating Training Files and Trainer Text Files

The following directives are available to create training files and trainer text files. The directives are listed in alphabetical order.

Breakpoint	Marker Name	Text
Cont At	Menu Options	Textwindow Hints
Display Menu	Nostop Typefile	Unset Window Output
File	Pause	User Input
Graphics Resume	Prompt	User Menu Pick
Graphics Stop	Set Window Output	User Pick
Ignore	Stop Typefile	

Note: You must include the open single quote (```) preceding the directive.

Breakpoint

Syntax	<code>`breakpoint</code>
Description	Stops the rapid execution of the training file and returns the trainee to single-stepping. Add this directive after the menu selection at which you want to break.

Cont At

Syntax	<code>`cont_at <label_name></code>
Description	Causes the training file to skip all commands and statements until the specified label is found. Execution continues from that point on. The argument <code>label_name</code> can point to a label further ahead in the training file, but cannot point back.
Example	<code>`cont_at get_out</code> Transfers training file execution to the label <code>get_out</code> .
See Also	Marker Name

Display Menu

Syntax	<code>`display_menu menu_name</code>
Description	Displays a menu that has already been defined with a <code>`menu_options</code> command. For the item selected, the action specified in the <code>`menu_options</code> command is performed.
Example	<code>`display_menu choices</code> Displays the menu named <code>choices</code> . It is assumed that the menu has already been defined with the <code>`menu_options</code> command.
See Also	Menu Options

File

Syntax	<code>`file name</code>
Description	Specifies the trainer's text file to be used with the training file. Only one such file can be specified; subsequent <code>`file</code> commands have no effect. You must enter the full path name only if the text file is not in the same directory (path names can contain up to 260 characters). If there is no path specified with the name, the system searches for the

	<p>file in the following directories, in order:</p> <ul style="list-style-type: none"> • The current directory • The directory in which the training file resides • The training directory, if defined in the <code>config.pro</code> file with the <code>TRAIN_PATH_FILE</code> configuration option
Example	<pre>`file text.txt</pre> <p>The file <code>text.txt</code> is associated with the training file.</p> <pre>`file /usr/users/train/text.txt</pre> <p>The file <code>text.txt</code>, located in the directory <code>/usr/users/train</code>, is associated with the training file.</p>

Graphics Resume

Syntax	<code>`graphics_resume</code>
Description	Restores the graphics mode after a <code>`graphics_stop</code> command. If the system does not encounter a previous <code>`graphics_stop</code> command, nothing happens.
See Also	Graphics Stop

Graphics Stop

Syntax	<code>`graphics_stop</code>
Description	<p>Stops putting new graphics on the screen. This command can be used to conceal a particular sequence of operations from the trainee.</p> <p>You can execute all the commands in the trail file between this directive and <code>`graphics_resume</code>, but they are not displayed on the screen. The geometry that is displayed in the main window is frozen at the point before the command preceding the <code>`graphics_stop</code> directive. The menus are not displayed. When the trail file</p>

	reaches the directive <code>`graphics_resume</code> , the system resumes normal geometry and menus are displayed.
Example	
See Also	Graphics Resume

Ignore

Syntax	<code>`ignore</code>
Description	Can be used in other commands to replace an action or argument. If an action is replaced, the system does nothing. If an argument is replaced, the system uses a default value.
Example	<code>`menu_options choices continue `ignore `ignore skip `cont_at dome_feature</code>

Marker Name

Syntax	<code>`marker label_name</code>
Description	Defines a label in the training file, which can then be referenced as a target for a jump.
Example	<code>`marker get_out</code> Defines a label called <code>get_out</code> in the training file.
See Also	Cont At

Menu Options

Syntax	<code>`menu_options menu_name item 1_name action1 {arg1} {item2_name action2 {arg2}...</code>
Description	Allows you to define a menu to be displayed to the trainee when running the training file. The argument <code>menu_name</code> is the name by which the menu is identified. The menu can have any number of items. Each item definition must have an action

	<p>command and an argument, if necessary. You can use <code>`ignore</code> for both actions and arguments.</p> <p>The number of characters in <code>menu_name</code> and any of its items cannot exceed 20.</p>
Example	<pre>`menu_options choices quit `cont_at get_out continue `ignore `ignore skip `cont_at dome_feature.</pre>
See Also	Marker Name

Nostop Typefile

Syntax	<code>`nostop_typefile</code>
Description	Returns the control of the INFORMATION WINDOW display to Pro/ENGINEER after the <code>`stop_typefile</code> command. If no previous <code>`stop_typefile</code> command has been encountered, nothing happens.
Example	<pre>`nostop_typefile</pre> <p>The INFORMATION WINDOW (other than the window with the trainer's text) scrolls quickly.</p>
See Also	Stop Typefile

Pause

Syntax	<code>`pause line number</code>
Description	Stops the execution of the training file until you click Resume to continue the execution of the training file again.
Example	<code>`pause 74</code>

Prompt

Syntax	<code>`prompt number</code>
Description	Displays a prompt message in the Message Window. The <code>number</code> argument is the number of blocks in the trainer's text file.

Example	<code>`prompt 77</code> Displays the first line of text block number 77 in the associated text file in the Message Window.
See Also	File Text

Set Window Output

Syntax	<code>`set_window_output</code>
Description	Restores regular training file operations for message output.
See Also	Unset Window Output

Stop Typefile

Syntax	<code>`stop_typefile</code>
Description	<p>Passes the control of the INFORMATION WINDOW display to the trainee, providing the ability to page back and forth, and so on.</p> <p>When the training file runs, Pro/ENGINEER information (such as the feature's information and mass properties calculations) is displayed in the INFORMATION WINDOW. If there are multiple pages in the messages in the INFORMATION WINDOW, they scroll by quickly, and there is no time to view them. This directive allows the trainee to control the display.</p> <p>This command does not affect the display of trainer's text. To switch this command off, use the <code>`nostop_typefile</code> command.</p>
See Also	Nostop Typefile

Text

Syntax	<code>`text <number></code>
--------	-----------------------------------

<p>Description</p>	<p>Specifies which block of the text file is to be displayed in the INFORMATION WINDOW at a specified interval when running the training file. Add this directive after the menu pick at which you want to break.</p> <p>Any text block can be displayed at any time, in any order. Repetitive block displays are also allowed.</p>
<p>Example</p>	<pre>\text 44</pre> <p>Displays text block number 44.</p>

Textwindow Hints

<p>Syntax</p>	<pre>\textwindow_hints {px py} {rows cols}</pre>
<p>Description</p>	<p>Allows you to change the size and position of the INFORMATION WINDOW with the trainer's text. In the syntax, <code>px</code> and <code>py</code> represent horizontal and vertical positioning of the text window (its lower-left corner), with respect to the main window. The argument <code>rows</code> specifies the number of rows in the text window, and <code>cols</code> specifies the number of columns in the text window.</p> <p>You can omit any pair of parameters, in which case the system defaults are used. (To omit the first pair, replace it with two <code>\ignore</code> commands.) If any parameter is out of range, the system uses the default values. The table Command Parameters lists the system defaults and valid ranges for the parameters.</p>
<p>Example</p>	<pre>\textwindow_hints 0.1 0.1 8 80</pre> <p>The INFORMATION WINDOW size will be 8 rows by 80 columns. Its origin will be offset at 0.1, 0.1 with respect to the main window.</p> <pre>\textwindow_hints \ignore \ignore 30 70</pre> <p>In this example, only the window size is specified (30 rows by 70 columns). The INFORMATION WINDOW origin will be at the default location.</p>

See Also	Ignore
----------	--------

Unset Window Output

Syntax	<code>`unset_window_output</code>
Description	Removes the Text INFORMATION WINDOW and diverts all message output to the Message window.
See Also	Set Window Output

User Input

Syntax	<code>`user_input keyboard_data {fail} {succ} {giveup} {retries}</code>
Description	<p>Forces the trainee to enter some keyboard data at some point while running the training file.</p> <p>The arguments are as follows:</p> <ul style="list-style-type: none"> • <code>keyboard_data</code>—The keyboard data line duplicated from the training file. • <code>fail</code>—The message to be displayed if the user fails to enter the expected data. The format is that of a <code>`text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>`ignore</code>, the default message is displayed. • <code>succ</code>—The message to be displayed if the user succeeds in entering the expected data. The format is that of a <code>`text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>`ignore</code>, the default message is displayed. • <code>giveup</code>—The message to be displayed if the user fails the maximum allowed number of retries, which is specified by the argument <code>retries</code>. The format is that of a <code>`text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>`ignore</code>, the default message is displayed.

	<ul style="list-style-type: none"> • <code>retries</code>—The maximum number of attempts allowed. If you do not specify this argument, or replace it with <code>ignore</code>, the system uses the default value (5). <p>The <code>`user_input</code> command line is a duplicate, not a replacement, of an existing line with keyboard data in the training file. To compose this command, copy a keyboard data line, and add the <code>`user_input</code> command prefix. Place the command above the actual keyboard data line.</p>
Example	<pre>`user_input 0.5 0.5</pre> <p>This example shows the basic <code>`user_input</code> command that uses the default message specifiers. The second line is the actual trail file line.</p> <pre>`user_input 0.5 'text 67 `ignore `ignore 2 0.5</pre> <p>This example defines the failure message (as the first line of block number 67 in the associated text file), and allows two tries. The second line is the actual trail file line.</p>
See Also	<p>Ignore</p> <p>User Menu Pick</p> <p>User Pick</p>

User Menu Pick

Syntax	<pre>`user_menu_pick command {fail} {succ} {giveup} {retries}</pre>
Description	<p>Forces the trainee to perform a menu pick at some point while running the training file.</p> <p>The arguments are as follows:</p> <ul style="list-style-type: none"> • <code>command</code>—The menu command line duplicated from the training file. The # character is retained.

	<ul style="list-style-type: none"> • <code>fail</code>—The message to be displayed if the user fails to make the expected menu pick. The format is that of a <code>`text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>`ignore</code>, the default message is displayed. • <code>succ</code>—The message to be displayed if the user succeeds in making the expected menu pick. The format is that of a <code>`text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>`ignore</code>, the default message is displayed. • <code>giveup</code>—The message to be displayed if the user fails the maximum allowed number of retries, which is specified by the next argument. The format is that of a <code>`text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>`ignore</code>, the default message is displayed. • <code>retries</code>—The maximum number of attempts allowed. If you do not specify this argument, or replace it with <code>ignore</code>, the system uses the default value (5). <p>The <code>`user_menu_pick</code> command line is a duplicate, not a replacement, of an existing menu pick line in the training file. To compose this command, copy a menu pick line and add the command prefix <code>`user_menu_pick</code>. Place the command above the actual menu pick line.</p> <p>Note: This function is available for standard Pro/ENGINEER options only, not user-defined options (which are defined using the functions Menu Options and Display Menu).</p>
Example	<pre>`user_menu_pick #DIMENSION #DIMENSION</pre> <p>This example shows the basic <code>`user_menu_pick</code> command that uses all the default message specifiers. The second line is the actual trail file line.</p>

	<pre>\user_menu_pick #REGENERATE \ignore \text 78\ignore 3 #REGENERATE</pre> <p>This example defines the success message (as the first line of block number 78 in the associated text file), and allows three retries. The second line is the actual trail file line.</p>
See Also	<p>Ignore</p> <p>User Input</p> <p>User Pick</p>

User Pick

Syntax	<pre>\user_pick x y b {fail} {succ} {giveup} {retries}</pre>
Description	<p>Forces the trainee to perform a graphics pick at some point while running the training file.</p> <p>The arguments are as follows:</p> <ul style="list-style-type: none"> • <i>x</i>, <i>y</i>, <i>b</i>—The XY coordinates and button code of a pick duplicated from the training file. • <i>fail</i>—The message to be displayed if the user fails to make the expected pick. The format is that of a <code>\text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>\ignore</code>, the default message is displayed. • <i>succ</i>—The message to be displayed if the user succeeds in making the expected pick. The format is that of a <code>\text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>\ignore</code>, the default message is displayed. • <i>giveup</i>—The message to be displayed if the user fails the maximum allowed number of retries, which is specified by the next argument. The format is that of a <code>\text</code> command. Only the first line of the specified text block is displayed. If you replace this argument with <code>\ignore</code>, the

	<p>default message is displayed.</p> <ul style="list-style-type: none"> • <code>retries</code>—The maximum number of attempts allowed. If you do not specify this argument, or replace it with <code>`ignore</code>, the system uses the default value (5). <p>The <code>`user_pick</code> command line is a duplicate, not a replacement, of an existing pick in the training file. To compose this command, copy a pick line, and add the <code>`user_pick</code> command prefix. Place the command above the actual pick line.</p>
Example	<pre>`user_pick 0.427022 0.495110 L 0.427022 0.495110 L</pre> <p>This example shows the basic <code>`user_pick</code> command, with all the default message specifiers. The second line is the actual trail file line.</p> <pre>`user_pick 0.427022 0.495110 L `text 43 `ignore `ignore 15 0.427022 0.495110 L</pre> <p>This example defines the failure message (as the first line of block number 43 in the associated text file), and allows fifteen retries. The second line is the actual trail file line.</p>
See Also	<p>Ignore</p> <p>User Menu Pick</p>

Example: Sample Training File

The following example of a training file contains five directives. The directives are displayed in red. The following key describes the function of each directive.

KEY:

- Lines 1 and 2 are the header.
- The first directive is the name of the file with the trainer's text.
- The second directive specifies the size of the text window.
- The third directive displays the first portion of the file `train1.txt`.
- The fourth directive stops the graphics display at that point.

Pro/ENGINEER Fundamentals – Help Topic Collection

- The fifth directive resumes the graphics display.

```
!trail file version No. ###  
  
!Pro/ENGINEER TM Version ##### Copyright 1999-2002 by Parametric  
Technology Corporation All Rights Reserved.  
  
!Select a menu item.  
  
#PART  
  
`file train1.txt  
  
`textwindow_hints 0.1 0.1 8 80  
  
#RETRIEVE  
  
!Enter Part name? [-Quit-]:  
  
block  
  
#FEATURE  
  
#CREATE  
  
#CUT  
  
`text 1  
  
#DONE  
  
#THRU_ALL  
  
#DONE  
  
`graphics_stop  
  
!Select or create a SKETCHING PLANE.  
  
0.610000 0.435000 L  
  
@ sel view 0  
  
0.668082 0.000000 -=0.668077 -0.385709 0.771437 -0.385712 0.545486  
0.545477 0.545491 416.488903 412.228321 -354.564531 1.000000  
  
!Arrow shows direction of feature creation.  
  
Pick FLIP or OKAY  
  
#OKAY  
  
!Select or create horiz. or vertical REFERENCE for sketching plane.  
  
0.535000 0.150000 L  
  
@ sel view 0  
  
0.668082 0.000000 -0.668077 -0.385709 0.771437 -0.385712 0.545486  
0.545477 0.545491 416.488903 412.228321 -354.564531 1.000000  
  
!Confirm selection.  
  
#CREATE  
  
!Enter Section name?:
```

```

@ ok
`graphics_resume
0.480000 0.510000 L 0.490000 0.485000 M 0.480000 0.510000 M 0.510000
0.435000 M
#DIMENSION 0.525000 0.575000 L
@ sel view 0
0.000000 0.000000 -1.474721 0.000000 1.474721 0.000000 1.474721 0.000000
0.000000 7s21.208185 90.062723 -405.548339 1.000000
@ sel2d CIRCLE 0 4 0.525000 0.625000 M

```

Distributed Computing Technology in Pro/ENGINEER

About Distributed Computing Technology in Pro/ENGINEER

Pro/ENGINEER uses distributed computing technology to augment your existing hardware so that it can perform computationally intensive tasks.

Distributed computing technology is comprised of the following:

- A session of Pro/ENGINEER (the controller)
- Participating networked workstations
- A daemon running on each participating workstation
- One or more agents running on the workstations (Pro/ENGINEER sessions running as server processes)
- A task (a collection of jobs)

The following steps outline how distributing computing technology works:

1. The controller communicates with the daemon on each remote workstation in order to begin an agent process.
2. The agent process establishes a connection with the controller.
3. The controller decides which agent receives the next job. The controller dispatches the jobs to the agents in order to most efficiently distribute the computing of the task.
4. The daemon on each workstation enables data streaming between the controller and the agents, provides statistics, provides job execution feedback, reports workstation loads, and monitors workstations on the network.
5. As each job is completed, the results are streamed to the controller.

Interactions between the Controller and the Agent

Distributed computing technology optimizes task distribution. The controller communicates with the daemon on each workstation and determines load averages.

Agent sessions are launched automatically, depending on load averages and the number of processors. These agents are started only once per distributed computing task, not once per job.

Data communication is also optimized. Data (models, information, instructions) are efficiently streamed directly to each agent via the daemon. Files are not copied to workstations prior to executing a job. As subsequent jobs are dispatched to the same agent, only data that is different between the jobs is streamed. If the models involved are the same, they are not streamed again.

To Specify Host Machines for Distributed Computing

You can specify which host machines you want to use for distributed computing by selecting them from a master list configured by your system administrator.

1. Click **Tools > Distributed Computing**. The **Distributed Computing Hosts Selection** dialog box opens and lists all host machines designated for distributed computing. Information is provided regarding the host name, the IP address, the hardware platform, the number of processors, and the CPU load for each workstation.
2. Use the **Distributed Computing Hosts Selection** dialog box to do the following:
 - Configure the current Pro/ENGINEER session so that it uses all or a subset of the designated workstations.
 - Use **Save** to save the selected hosts in a file on disk. To load the files at a later time, use **Load**.
 - Change the selected host machines at any time during a session.
 - Use **Refresh** to recreate the list of host machines. The system removes any machines that are currently off the list.
 - Select a column heading, or use the **Sort by** box, to sort the host machine list.

To interrupt or cancel a distributed computing task, click **Stop**. Participating agent workstations are made free within a few seconds so that they can be used by others who want to execute a distributed computing task.

Distributed computing is suitable only for Multi-Objective Design studies. Distributed computing tasks automatically use the dialog box settings. When you begin executing a task, a progress dialog opens and displays the progress of the execution, statistics about how many jobs are being processed by agents on each of the participating workstations, and the status of each workstation.

To Configure Workstations to Use with Distributed Computing

Note: This procedure should be performed by your company System Administrator.

After installing Pro/ENGINEER release 2000i2 or later, your system administrator must configure workstations to use with distributed computing by performing the following tasks:

1. From the available workstations at your company, choose one that can participate in distributed computing sessions. Any mix of UNIX and Windows workstations can be used.
2. Execute the `dcadsetup` command script on each of the participating workstations. Running this script starts a daemon process that is required for distributed computing. No other setup task is required.
3. To shut down the daemon process and thus disable a workstation from participating in distributed computing sessions, the `dcadshutdown` command script is available.
4. Optionally, your system administrator may register the `dcadsetup` command within the boot script of a workstation to ensure that the workstation is always enabled after shutdown and restart procedures.

Viewing Model Creation History




About Viewing Model Creation History

The **Model Player** option on the **Tools** menu lets you observe how a part is built. It aids in the diagnosis of bad features in a part by allowing you to view the creation history of a model. You can use this option at any time, or instead of regenerating a model after you modify it. The Model Player allows you to do the following:

- Move backward or forward through the feature-creation history in the model to observe how the model was created. You can also start the model playback at any point in its creation history.
- Regenerate each feature in sequence, starting from the specified feature, as the model moves forward.
- Display each feature as it is regenerated or rolled forward.
- Update (regenerate all the features in) the entire display when you reach the desired feature or when the playback process is complete.
- In Manufacturing mode, recalculate CL data for each NC sequence during regeneration.
- Obtain information about the current feature (the feature that was currently regenerated when you stopped the model playback process). You can show dimensions, obtain regular feature information, investigate geometry errors, and enter Fix Model mode.

To View Model Creation History

1. Click **Tools > Model Player**. The **Model Player** dialog box opens.

2. Select one of the following options for controlling feature regeneration and display:
 - **Regenerate features**
 - **Display each feature**
3. Using the control buttons or the slider, select the point in the model creation history (that is, select the feature) at which you want to begin the regeneration process. To select a starting feature from the Graphics window or the Model Tree, click . You can also choose **Feat #** and type a feature number to specify a starting feature. Use  and  to choose a starting feature by stepping backward or forward through the playback operation one feature at a time.

The feature you specify becomes the starting feature. The system suppresses all features that occur after it.


4. To stop playback, click **Stop**.
5. Click a command at the bottom of the Model Player to get information about the current feature (the feature that was currently regenerated when you stopped the model playback process).



If a feature is one that intersects any other feature on the part, the system displays the geometry of the feature in red wireframe before intersection.


Using the Model Player


To open the **Model Player** dialog box, click **Tools > Model Player**.

- Select one of the following commands for controlling feature regeneration and display:
 - **Regenerate features**—Regenerates each feature in sequence, starting from the specified feature, as the model moves forward. If this box is cleared, features will be rolled forward without regenerating. If you have made changes to the model without regenerating, the command is checked and unavailable.
 - **Display each feature**—Displays each feature in the Graphics window as it is being regenerated or rolled forward. Future features are not displayed until they are regenerated. If this box is cleared, the entire display is updated only when the desired feature is reached and the model playback is complete, or when you stop model playback.
 - **Compute CL**—(Available in Manufacturing mode only) When selected, the CL data is recalculated for each NC sequence during regeneration. If cleared, no CL data is calculated during regeneration.
- Select one of the following commands to select the place (feature) in the model creation history at which to start the regeneration process:


—Moves immediately to the beginning of the model (suppresses all features).



—Moves immediately to the end of the model (resume all features). Use  to step backward through the features one at a time.

—Steps backward through the model one feature at a time and regenerates the preceding feature in the model.

—Steps forward through the model one feature at a time and regenerates the next feature in the model.

Slider Bar—Drags the slider handle to the first feature at which you want model playback to begin. The features are highlighted in the Graphics window as you move through their position with the slider handle. The feature number and type are displayed in the selection panel (such as #16 (CHAMFER)), and the feature number is displayed in the **Feat #** box. When you release the slider, the model immediately rolls or regenerates to that feature.

—Lets you select a starting feature from the graphics window or the Model Tree. Opens the **SELECT** dialog box and **SELECT FEAT** menu. After you select a starting feature, its number and ID are displayed in the selection panel, and the feature number is displayed in the **Feat #** box. When you select a feature, the model immediately rolls or regenerates to that feature.

Feat #—Lets you specify a starting feature by typing the feature number in the box. After you enter the feature number, the model immediately rolls or regenerates to that feature. Use  and  to step forward or backward through the playback one feature at a time.

The feature you specify becomes the starting feature, and all features that occur after it are suppressed.

- To stop playback, click the **Stop** button.
- You can use the following commands to obtain information about the current feature (the feature that was currently regenerated when you stopped the model playback process):
 - **Show Dims**—Displays the dimensions of the current feature.
 - **Feat Info**—Provides regular feature information about the current feature in an Information window.
 - **Geom Check**—Investigates the geometry error for the current feature. This command is accessible only when the system encounters a geometry error.
 - **Fix Model**—Activates Resolve mode by forcing the current feature to abort regeneration. When you exit Resolve mode, Pro/ENGINEER returns to the

Model Player at the current feature. You can then continue to move backward or forward through the model.

- **Close**—Closes the Model Player and enters Insert mode at the current feature (the last regenerated feature). You can choose this command anytime during the model playback.
- **Finish**—Closes the Model Player and returns to the last feature in the model. Pro/ENGINEER restores all features. You can choose this command anytime during model playback.

If a feature is one that intersects any other feature on the part, the system displays the geometry of the feature in red wireframe before intersection.

Working with External Reference Control

About External Reference Control

Pro/ENGINEER provides reference control tools for specifying system behavior when you create external references among features in a design. You can specify the reference control settings in the **External Reference Control** dialog box that opens when you click any of the following:

- **Tools > Assembly Settings > Reference Control**
- **Edit > Setup > PART SETUP > Ref Control**, for a part in Part mode
- **Edit > Setup > ASSEM SETUP > Ref Control**, for a subassembly within an assembly

You can:

- Specify global scope settings that apply to all components in the current session except objects that have an object-specific scope setting and objects that have a less restrictive object-specific scope control.
- Specify object-specific scope settings for external references.

To Set External Reference Control Scope Globally

In Part or Assembly mode, you can define external reference control scope globally for the general Pro/ENGINEER environment.

1. Click **Tools > Assembly Settings > Reference Control**. The **External Reference Control** dialog box opens. The **Objects** tab is selected by default.
2. Under **Components Allowed to be Referenced**, select the appropriate option to create external references to any component, to components in the same subassembly, or to the skeleton of any high level subassembly.
3. Click the **Geometry** tab.
 - Under **Geometry to be Referenced**, select the appropriate option to restrict reference selection of external references to only published geometry, for all models, or only for models with published geometry.

- Under **Allowed Placement References**, specify if you want to reference entire geometry with component constraints or only component interfaces as component constraints.
4. Click the **Selection tab**.
- Under **Exclude from Selection Bin**, select the required settings to include or exclude the forbidden references.
 - Under **Colors of Forbidden References**, specify the color settings as follows:
 - To highlight all prohibited components, select **Change Color of Not Allowed for Backup**, and then click the icon to the right of the text to open the **Color Editor** dialog box, in which you can set the highlight color.
 - To highlight all components from which local copies must be made, select **Change Color of Allowed for Backup**, and then click the icon to the right of the text to define the highlight color.
 - To disable the selection of out-of-scope references or invalid components, clear both the check boxes.

Note: If there are object-specific settings in addition to the environmental setting for scope control, the system enforces the more restrictive setting for the object.

To Set External Reference Control Scope for Specific Objects

You can specify a particular scope setting and external reference-handling scheme for a specific object. The information is stored with the object and is in effect for each part or assembly file in which the object appears.

Note: You can set external reference control scope both at model and component level.

1. Use one of the following methods to control the reference scope and handling for a component.
 - To set the reference scope and handling for a part in Part mode, click **Edit > Setup > PART SETUP > Ref Control**.
 - To set the reference scope and handling for a subassembly within an assembly, click **Edit > Setup > ASSEM SETUP > Ref Control** and then select **Inside Subassembly** in the **External Reference Control** dialog box.
 - In the Model Tree, right-click to select **Reference Control** for a part, a part within an assembly, or a subassembly within an assembly.

Each of these methods opens the **External Reference Control** dialog box.

2. The **Accessible** tab is selected by default. If not, click the **Accessible** tab.

Under **Components Allowed to be Referenced**, define the scope of components you want to be referenced by selecting one of the following:

- **All**—Allows you to create any external references to any other component. This setting is equivalent to having no scope control at all.
 - **Inside Subassembly**—Allows you to create external references only to components of the same subassembly.
 - **Skeleton Model**—Allows you to create external references to skeleton models only.
 - **None**—Allows no external references to be created.
3. Click the **Backup Forbidden References** check box to create out-of-scope references. In this case, you can create an out-of-scope reference as an invisible local copy only.

Else, clear **Backup Forbidden References** to prohibit the creation of external references that violate the scope. In this case, if creation of an out-of-scope reference is attempted, the action stops and a message identifying the part or model whose scope has been violated appears.

4. Click the **Shared** tab.
- Under **Geometry Allowed for Referencing**, if you want to restrict selection of external references to only published geometry, select **Published Geometry**. You can also specify whether you want to restrict selection for all models or only for models with published geometry.
 - Under **Allowed Placement References**, specify if you want to reference entire geometry with component constraints or only component interfaces as component constraints.

Note: If there are object-specific settings as well as an environmental setting for scope control on an object, the system enforces the more restrictive setting for the object.

The Window Menu

About the Window Menu

The **Window** menu contains commands for activating, opening, closing, and resizing Pro/ENGINEER windows. You can also switch between open windows by selecting the desired window at the bottom of the **Window** menu.

Activating a Window

To use all the Pro/ENGINEER features that are applicable to an object, you must activate the window containing the object. You can activate a window by doing one of the following:

- Press CTRL+A.
- **Note:** If a list is available, CTRL+A selects all items in the list. Otherwise, it activates the current window.

- Click **Window > Activate**.
- Select an object from the list in the **Window** menu.

In an active window:

- The word "Active" appears on the title bar after the model name.
- All applicable menu commands on that window become available.
- If you minimize the active window, all windows are minimized.
- The file name associated with the object is selected in the list of open windows on the Window menu.

Opening a New Window

You can create a Pro/ENGINEER window that contains the same object that is in the current window. You can then modify the model and rename it.

In Part or Assembly Mode

To create a new window in Part or Assembly mode, use **Window > New**. The object in your current Pro/ENGINEER window is also present in the new window. The new Pro/ENGINEER window becomes the active window.

In Other Modes

The procedure for creating a new window varies, depending on the module or mode you are in at the time. For module-specific procedures, refer to the Help topics on the specific module.

Closing a Window

You can remove a window from the screen and still retain its object in memory by doing one of the following:

- Click **Window > Close**.
- Click **File > Close Window**.

If there was an object in that window, the object remains in memory throughout the session, or until you click **File > Erase > Current**.

If only one Pro/ENGINEER window is open, the object is removed, and the window remains open.

Resizing a Window

You can resize a Pro/ENGINEER window using several methods:

- Drag any side or corner of the window.
- Click the maximize or minimize button on the title bar.

- Use the Pro/ENGINEER **Maximize**, **Default Size**, and **Restore** commands on the **Window** menu.

To Open a System Window

Click **Window > Open System Window**.

You can open a system window (called a command prompt window, in Windows) directly from within Pro/ENGINEER. Use a command-line editor to edit a configuration file or to carry out other operating-system commands. You must exit the system window before you can continue using other Pro/ENGINEER functions.

The Help Menu

About the Help Menu

Use the **Help** menu to access the Help Center home page, context-sensitive Help, release information, and customer service information. The following commands are available:

Help Center—Opens the home page for the Pro/ENGINEER Help Center. From here you can navigate to any Help topic. Topics are arranged by functional area. You can search for information within a functional area or multiple functional areas.

What's This?—Enables context-sensitive Help mode.

Menu Mapper—Opens the Menu Mapper, an interactive tool that facilitates learning of the improved menu structure of Pro/ENGINEER Wildfire.

Online Resources—Opens the Online Resources page.

What's New—Opens the What's New document that highlights the new functionality.

Technical Support Info—Provides product information, including the release, license information, installation date, and customer support contact information.

About Pro/ENGINEER—Displays Pro/ENGINEER copyright and release information.

Context-Sensitive Help

To Get Context-Sensitive Help

1. Click **Help > What's This?** or .

Note: In some cases, it may be easier to open the dialog box or menu first and then click **Help > What's This?**

2. Click the icon, dialog box, dialog box option, or menu command for which you want help. Pro/ENGINEER displays Help for the item.

Note: When you are working with the Menu Manager, you can also right-click an active command, and click **GetHelp**.

Working with the Model

Model Tree


About the Model Tree

The Model Tree is a list of every feature in a part file including the datums and coordinate system. In a part file, the Model Tree shows the part file names and each feature in the part below it. In an assembly file, the Model Tree shows the assembly file names and the included part files beneath it.

The model structure is displayed in a hierarchical (tree) format with the root object (the current part or assembly) at the top of its tree and the subordinate objects (parts or features) below. If you have multiple Pro/ENGINEER windows open, the Model Tree contents reflect the file in the current window.

The Model Tree lists only the related feature- and part-level objects in a current file and does not list the entities such as edges, surfaces, curves, and so forth that comprise the features.

Each Model Tree item contains an icon that reflects its object type, for example, hidden, assembly, part, feature, or datum plane (also a feature). The icon can also show the display or regeneration status for a feature, part, or assembly, for example, suppressed or unregenerated.

By default, the Model Tree is located in the Pro/ENGINEER main window. Click  on the toolbar to show or hide the Model Tree. Click **Settings** to customize the appearance of the Model Tree.

You can save all of the information in the Model Tree as a text (.txt) file. Subsequently, you can print the file, import it into a text editor and format it for review, import it into Microsoft Excel, and apply scripts to it. You can add columns to the Model Tree that display selected information about the model.



Selection in the Model Tree is object-action oriented; you select objects in the Model Tree without first specifying what you intend to do with them. You can select components, parts, or features using the Model Tree. You cannot select the individual geometry that makes up a feature (entities). To select an entity, you must select it in the graphics window.

By default, the Model Tree displays each embedded datum as a feature subnode in the creation order that is the last created item is displayed first. For a feature, the display of embedded datums and other subnodes on the Model Tree is in the order of sketch, annotation, embedded datums, and transformed or patterned features. The patterned, mirrored, or moved features are automatically grouped when you embed datum features in them. They are displayed in the Model Tree with labels that include the tool names, such as Patterned Features, Mirrored Features, or Moved Features. The embedded datum features are displayed as sub-nodes of the group header, above all other group members. The asynchronous datums with their respective feature are displayed as a feature subnode on the Model Tree. In the Model Tree using the drag-and-drop technique, you can convert a stand alone datum to an embedded datum and vice versa.

About Manipulating the Order of Features on the Model Tree

The Model Tree displays a hierarchical structure of all the features in a Model.

In addition to selection, you can drag nodes on the Model Tree. Based on the context, you can drag nodes to subsequently perform different operations such as reordering features on the Model Tree, creating or manipulating groups, embedding datum features, activating the insertion mode, and so on. The context is determined by the node that you select for the drag-and-drop operation and the location where you want to place the dragged node.


If the new location where you want to place the dragged feature is not valid, the pointer changes to . If the location where you want to place the dragged feature is a valid location, an internode indicator () appears on the Model Tree.



Note: Reordering existing features can change the order of features on the Model Tree and in turn, the geometry of the model.



During the drag-and-drop operation, Pro/ENGINEER maintains the referential integrity. That is, you cannot place a child feature before its parent feature or a parent feature after its children. In order to maintain this referential integrity, Pro/ENGINEER dynamically handles the dependencies between the features. That is, the interfering parent or children in the sequence on the Model Tree are automatically selected and dragged with the dragged node, and the parent-child relationship is maintained.

While performing a drag-and-drop operation on the Model Tree, Pro/ENGINEER highlights the selected members and their dependent members based on the relation or properties of the selected members. For example:

- If you drag a feature over a collapsed group node and hold it there for a few seconds without placing it over the node, the group node expands automatically.
- While dragging a feature over an expanded group node, the color of the group header changes to dark grey. This color indicates that you are placing the feature in the group. Similarly, the color of the group header changes to dark grey when you drag a member of that group to place it outside the group.
- If you drag a feature over a collapsed group and the group contains a dependent feature of the feature you are moving, the color of the group header changes to light blue indicating that the dependent features will also be moved out of the group. For example, if you drag a parent feature over a collapsed group containing a child of that feature and place it outside the group, the child feature is placed with its parent outside the group.
- If you drag a datum over other features on the Model Tree, the color of the datum header changes to light grey as soon as you move it over a feature in which you can embed the datum. However, while embedding, the feature number does not change and thus the order of regeneration of features may be different from the order in which the features appear on the Model tree.
- When you drag a feature that is related to other features by one or more dependencies, the other dependent features are automatically selected on the

Model Tree. A  sign appears on the icons of the dependent features to indicate that those features were automatically selected.

For example, when you drag a child feature (Extrude) above the parent feature (Datum), the parent feature is highlighted on the Model Tree and its icon changes to  to indicate that the feature was automatically selected. Similarly, when you drag a parent feature below its child, the child feature is automatically highlighted and its icon changes to a .

- If you select multiple items on the Model Tree to perform a drag-and-drop operation, a  appears next to the pointer.
- You cannot reorder some features, such as suppressed features or 3D notes.
- If you try to place the insertion locator ( Insert Here) after a suppressed feature, it is automatically placed before the suppressed feature or any other suppressed features that exist before it. Additionally, if you try to place the insertion locator after a group whose last member is suppressed, Pro/ENGINEER activates the insert mode and places the insertion locator before the group.



Note: The insertion locator on the Model Tree indicates where a feature will be inserted upon creation.

Understanding Glyphs on the Model Tree

The hierarchical structure of the Model Tree contains feature names and their corresponding icons. In addition to the feature icons on the Model Tree, Pro/ENGINEER uses glyphs to depict the different states of the features when you perform various operations on the features. A glyph, located next to the feature icon, is a modifier that Pro/ENGINEER uses to indicate whether a component is suppressed, not complete, frozen, and so on.

Pro/ENGINEER uses different glyphs depending on whether you are working on a part, assembly, manufacturing model, drawing, format, report, diagram, layout, markup, and so on.

The following table describes some of the glyphs on the Model Tree:

Glyph	Description
	<p>Indicates that the feature is currently being created or redefined.</p> <p>You create models by building features. To create features, you can create a new sketch, part, assembly, manufacturing model, drawing, format, report, diagram, layout, markup, or interchange file in Pro/ENGINEER using the New and New File Options dialog boxes.</p>
	<p>Indicates that the feature is not fully defined (incomplete).</p> <p>You can suspend feature creation without losing data that you have already defined. This produces an incomplete feature, which you can complete later when the design intent is clear or the required</p>

Glyph	Description
	references are available. A fully defined feature cannot be made incomplete.
☒	<p>Indicates that the feature has failed regeneration.</p> <p>Regeneration can fail for reasons such as bad geometry, broken parent-child relationships, and missing or invalid references.</p>
□	<p>Indicates that the feature is a packaged component.</p> <p>A packaged component is an assembly component that is not fully constrained. All moving assembly components are packaged.</p>
☒	<p>Indicates that the feature is a child of a packaged component.</p> <p>When you place a component in an assembly by referencing a packaged parent, the newly placed component is called a child of the packaged component. The component is shown as Child of Packaged in the Model Tree Status column.</p>
✖	<p>Indicates that you are duplicating and placing a component in an assembly by copying and pasting it.</p> <p>While placing the component, the placement references of the source component are maintained but the assembly references must be redefined.</p>
■	<p>Indicates that the feature is suppressed.</p> <p>Suppressing features is similar to removing them from the model temporarily. However, you can unsuppress (resume) the suppressed features at any time. You can suppress features in a model to simplify the model and reduce regeneration time.</p>
☐	<p>Indicates that the component placement uses mechanism joints, and is not fully constrained.</p>
	<p>Indicates that the feature is frozen. (Assembly Mode only)</p> <p>Freezing is similar to suppressing a feature. Freezing keeps the selected feature in its current location. You can freeze only the components of an assembly.</p>
☒	<p>Indicates that the feature is a child of a frozen feature.</p> <p>Children of suppressed components can be frozen in their original location until they can be regenerated successfully, either by resuming the parent, redefining the placement, or rerouting the child.</p>

Basic Model Tree

Model Tree Display

About Searching in the Model Tree

When you search for model properties or other information the results are highlighted in the Model Tree. To perform a search, you use the **Search Tool** dialog box, accessed from the **Find** command on the **Edit** menu.

About Adding Information to the Model Tree


You can add informational columns to the Model Tree window, such as columns containing parameters and values, assigned layers, or feature name for each item. You can use the cells in the columns to perform context-sensitive edits and deletions. The following types of information can be added:

- **Info**—Use this option to add information, including Status, Feat #, Feat ID, Feat Type, Feat Name, Copied Refs, Model Size, Feat Subtype and Designate Name.
- **Model Params**—Use this option to display any established parameters that affect an entire model. You can also define new model parameters using this option.
- **Database Params**—Use this option to view local, Windchill and Pro/INTRALINK database parameters.
- **Feat Params**—Use this option to create feature parameters in the Model Tree. You can also display the existing parameters by specifying the existing parameter name in the **Name** field.
- **Annotation Elem Params**—Use this option to create annotation element parameters in the Model Tree. You can also display the existing parameters by specifying the existing parameter name in the **Name** field.
- **Simplified Reps**—Use this option to view simplified representations in the current assembly.
- **Ext. Simp. Rep**—Use this option to view external simplified representations in the current assembly.
- **Display Styles**—Display Style, DEFAULT
- **Layer**—Use this option to display layer status and name of each layer in the current model.
- **Note**—Use this option to display information about notes, including Note ID, First Line, Note Display, any URL that has been assigned to the note, and Note Type.
- **Cable Info**—Use this option to set and display default Cabling information such as Designation, Diameter, Logical From, Logical To, Physical From, Physical To and Length.

- **Cabling Params**—This option allows you to enter and display user-defined cable parameters.
- **ECAD Params**—Use this option to display parameters specifically associated with ECAD parts and features. For more information on the specific parameters you can add, refer to the Help on ECAD.
- **Reference Control**—Use this option to display the references that are set to your current model. You can also choose to display shared model references.
- **Mass Properties Params**—Use this option to display the Mass Properties in your model. If no properties are assigned, you can type them into a drop-down list box. If more than one property is assigned, you can select from them a drop-down list box.
- **Zones**—(Assembly mode only) Use this option to display components inside the zone in the current assembly.

You access the **Model Tree Columns** dialog box by clicking **Settings > Tree Columns**.

To Add Columns to the Model Tree

1. Click **Settings > Tree Columns**. The **Model Tree Columns** dialog box opens.
2. In the **Type** list, select a type (such as **Info**, **Layer**, or **Note**) and a subtype (such as **Feat #** or **Feat ID** for the **Info** type, or **Layer Names** or **Layer Status** for the **Layer** type).
3. Click  to add types to the **Displayed** list. The new parameter is added to the **Displayed** list.
4. Click **Apply > OK**. The dialog box closes and the new parameter column is added to the Model Tree. The status of a feature (such as regenerated or failed) is indicated next to its icon.

About Model Tree Operations

You can perform the following operations using the Model Tree:

- Rename files on the Model Tree
 - Click the icon next to the filename or double-click the name. A type box opens.
 - Type the new name in the type box provided.
 - Press **Enter**.
- Select features, parts, or assemblies, and perform object-specific operations on them using the shortcut menu.

Note: You cannot use the Model Tree to select individual geometry that is used to make a feature (entity). To select entities, you must use the graphics window.

- Filter the display by item type or status, for example, showing or hiding datum features, or showing or hiding suppressed features.
- Open a part within an assembly file by right-clicking the part on the Model Tree.
- Create or modify features and perform other operations such as deleting or redefining parts or features, rerouting parts or features, and so on, using the shortcut menu (accessed by right-clicking a part name).
- Search the Model Tree for model properties or other feature information by clicking **Edit > Find** on the **Pro/ENGINEER** menu bar.
- Show the display or regeneration status for a feature, part, or assembly, for example suppressed or unregenerated.

Note: Each Model Tree item contains an icon that reflects its object type, for example, assembly, part, feature, or datum plane. The icon can also show display or regeneration status, for example, suppressed or unregenerated.

About Selecting in the Model Tree

You can use the Model Tree to select features or parts for editing. This is especially useful when the selected feature or part is not visible in the graphics window. You can also right-click the feature or part to open a shortcut menu, from which you can select object-specific commands relative to the selected object.

You can also select features and parts in the Model Tree even when their selection is disabled in the graphics window.

Note: You cannot select individual geometry (entities) in the Model Tree. The Model tree lists only features and not individual entities, as they are the components of features.

The Model Tree Insertion Locator

The Model Tree insertion locator, indicated on the Model Tree by *Insert Here*, indicates where features will be inserted upon creation. By default, its position is always after all items listed in the Model Tree. You may drag it higher or lower in the Model Tree to insert features between other features in the list. When you move the insertion locator, the model does not reorient irrespective of backward or forward movement of the insertion locator in the Model Tree. The graphical display of model in current view is displayed.

After it is moved, you can return the insertion locator to its default location by positioning the pointer over it and selecting **Cancel** on the shortcut menu or by dragging the insertion locator to the bottom of the model tree.

To Move the Model Tree Insertion Locator

1. Position the pointer over *Insert Here* on the Model Tree.
2. Press and hold the left mouse button and drag the pointer to the desired location. The insertion locator moves with the pointer.

3. Release the left mouse button. The insertion locator is in the new location and model orientation of the current view is retained. The model does not roll back to the new location. Pro/ENGINEER displays a message in the message area that indicates `Insert mode has been activated.`

Use the following procedure to return the insertion locator to its default position:

1. Position the pointer over `Insert Here` on the Model Tree.
2. Right-click to open the shortcut menu.
3. Select **Cancel**. The insertion locator returns to its default position. Pro/ENGINEER displays a message in the message area that indicates `Insert mode has been cancelled.`

Alternatively, you can return the insertion locator to its default position by dragging the insertion locator to the bottom of the model tree.

About Model Tree Status Display

You can display the model status in the Model Tree by setting **Tree Columns > Info > Status** and **Tree Columns > Database Params > Database Status**.

The status is displayed in the Model Tree column and describes the object state, for example, Regenerated. The database status describes whether the object is Editable or Read Only.

When a PDM model is displayed, the Pro/ENGINEER title bar displays the part name, revision and its status for example, In Work.

To Select Objects Using the Model Tree

You can use the mouse or the keyboard to select objects on the Model Tree.

Using the Mouse

1. Click **Show > Highlight Model** in the Pro/ENGINEER navigator window to check the **Highlight Model** option.
2. Click the name of a feature or part in the Model Tree. Pro/ENGINEER activates the window containing that feature or part and highlights the selected object in the graphics window.



Note: Selections in the graphics area are cleared when items are selected using CTRL+Click in the Model Tree.

You cannot use CTRL + A to select all items in the Model Tree.

Using the Keyboard

1. Click the Model Tree window to activate it. A dotted rectangle appears around the top-level Model Tree object.
2. Press the arrow keys to move the rectangle to other Model Tree objects.
3. Press the space bar once the rectangle is on the object you want to select.

To Change Model Tree Column Order

1. Click **Settings > Tree Columns**. The **Model Tree Columns** dialog box opens.
2. Under **Displayed**, select the column you want to reorder and click   to move the column to the left or right on the Model Tree.
3. Click **Apply > OK**. The selected columns are moved to their newly specified positions in the Model Tree.

To Change Model Tree Placement

1. Click **Tools > Customize Screen**. The **Customize** dialog box opens.
2. Click the **Navigation Tabs** tab.
3. Select from the Placement list in the Model tree settings window.
4. Select desired placement option.
5. Click **Apply settings > OK**.

To Display or Hide Items On the Model Tree

To hide items, select the item or items on the Model Tree, right-click, and select **Hide**. The selected item or items are temporarily hidden in the graphics window and added to the **Hidden Items** layer.

To display the hidden items, select the item that you want to display, right-click, and use one of the following methods:

- On the Layer Tree, under **Hidden Items**, right-click and select **Unhide**.
- On the Model Tree, right-click and select **Unhide**, or click **View > Visibility > Unhide**.

You can also display or hide the Model Tree by using one of the following methods:


- On the Model Tree, click the plus sign or minus sign to expand or collapse the Model Tree, respectively.
- On the Layer Tree, click **Show > Model Tree**.

To Expand or Collapse the Model Tree

Click **Show > Expand All**, or **Show > Collapse All**. This expands or collapses all branches and displays only the highest, or parent part, assembly, or feature, in the Model Tree. To expand or collapse single branches, click the plus or minus sign in the tree node.

To Remove Columns from the Model Tree

1. On the Model Tree, click **Settings > Tree Columns**. The **Model Tree Columns** dialog box opens.



2. Under **Displayed**, select the column you want to remove and click .
3. Repeat step 2 for each column you want removed from the Model Tree.
4. Click **Apply > OK**.

To Reorder Features in the Model Tree

1. Select one or more features in the Model Tree.
2. Drag the selected feature or features to a new location in the feature list.

Note: Some features cannot be reordered, such as suppressed features or 3D notes. If you try to move a child part to a higher position than its parent part, the parent part moves with the child part in context, and the parent/child relationship is maintained. Also, if you move a parent part to another position, the children parts move with the parent part in context, so the parent/child relationship is maintained.

To Show or Hide Items by Type on the Model Tree

1. Click **Settings > Tree Filters**. The **Model Tree Items** dialog box opens.
2. Under **Display**, click or clear the desired object state options.
3. Under **Feature Types**, click a tab and click the feature types you want to display. You can click  to select all feature types to display, or  to hide all feature types.
4. Click **Apply > OK**.

About Suppressing Items in the Model Tree

Suppressing a feature temporarily removes it from the model, both physically and visually. You can use the Hide command to visually remove a feature. However, if you want to temporarily remove a feature, for example, in order to try another one in its place. Suppressing and resuming features allow you to temporarily remove a single or group of features and restore them later.

To Suppress and Resume Features

1. Right-click the feature in the Model Tree. The shortcut menu appears.
2. Click **Suppress**.

Note: If suppressed features are hidden in the Model Tree, click **Settings > Tree Filters** and select the **Suppressed Objects** checkbox. When suppressed features are shown, you can select them and click **Resume** from the shortcut menu to return them to the model.

Model Tree Notes

To Create a Model Note Using the Model Tree

Use the following procedure to create a note in your model from the Model Tree:

1. Right-click a feature or part in the Model Tree. A shortcut menu opens.
2. Click **Setup Note > Feature** (or **Assembly, Part,** or **Component** in Assembly mode). The **Note** dialog box opens.
3. Specify the name for the note in the **Name** field, or use the default.
4. Type the note text in the **Text** field, or click **Insert** to insert text from a file or from another note.
5. (Optional) Click **Symbols** to select from a gallery of symbols to add to the note.
6. (Optional) Under **Placement** select one of the following for attaching the note to the displayed object:
 - **Place**—Places note on the model
 - **Move Text**—Moves the note text location
 - **Mod Attach**—Redefines the attachment point of the note with reference to an entity or surface
 - **Move**—Moves the leader in 360 degrees to change the location and size of the leader with reference to the attachment point.

Optionally, you can also select **Place Note Flat to Screen** while creating a note so that the note text remains flat to screen. This allows you to read the note text even when the model is rotated in different directions.

7. Click **Place**. The **Select** dialog box opens.
8. Select a datum plane or symbol to which the new symbol will be parallel. The **DIRECTION** menu appears.
9. Click **Okay** or middle-click to accept the direction for operation, or **Flip** to reverse the direction. The **NOTE TYPES** menu appears.
10. Select the required note type or use the defaults.
11. Click **Done**. The **ATTACH TYPE** menu appears.
12. Select the required options or use the defaults.
The **Select** dialog box opens.
13. Click the required placement area in the main window to place the note.
14. Click **OK** in the **Select** dialog box.
15. Click **Done** in the **ATTACH TYPE** menu.
16. Click on the graphics area to create the note and place it in the selected area.

17. Click **OK** in the **Note** dialog box.

To Create a Note Hyperlink Using the Model Tree

1. Click **Hyperlink** in the **Note** dialog box. The **Edit Hyperlink** dialog box opens.

OR

Right-click the note in the Model Tree to which you want to attach a URL. A shortcut menu appears.

2. Click **Add Link**. The **Edit Hyperlink** dialog box opens.

3. Enter a URL or Internal link in the text box and click **OK**.

4. Select the required note in the Model Tree and right-click the note in the main window or Model Tree. A shortcut menu appears.

5. Click **Open URL**. The Web site attached to the note opens in the Pro/ENGINEER browser.

6. To add the address to your favorites, right-click the Web page and select **Add to Favorites**.

Similarly, you can edit or remove hyperlinks from notes using the shortcut menus. You can also **Move**, **Erase**, or **Delete** the selected note. Use **Text Style** to change the text properties of the note.

Saving Model Tree Information

To Save Model Tree Information

1. Click **Settings > Save Model Tree**. The **Get Name** dialog box opens.

2. Accept the default file name `treetool.txt`, or type another name and click **OK**. The text file is created and saved in your working directory.

Note: Pro/ENGINEER does not automatically save Model Tree information to a text file. To update the text file, you must perform this procedure.

To Save Model Tree Settings

You can save your Model Tree setup and apply it to other models. With more than one assembly, part or drawing open in your session, do the following:

1. Click **Settings > Apply Settings from Window** in the Model Tree navigator. The **Change Config Settings** dialog box opens.

2. Select the appropriate model from the list.

3. Click **Apply > OK**.

4. Click **Settings > Save Settings File** to save these settings to a file. The **Save Model Tree Configuration** dialog box opens.

5. Accept the default file name, that is, `tree.cfg`, or type a new name.
6. Click **Save**.

Advanced Model Tree

Configuration

To Save a Model Tree Configuration

1. Click **Settings > Save Settings File**. The **Save Model Tree Configuration** dialog box opens.
 2. Accept the default file name `tree.cfg`, or type a new name.
 3. Click **Save**.

To Retrieve a Saved Model Tree Configuration

1. Click **Settings > Open Settings File**. The **Load Model Tree Configuration** dialog box opens.
 2. Click **Open**. If necessary, navigate to the directory where the desired configuration file resides, and then double-click the file name to open it, or select the file and click **Open**.

To Load a Model Tree Configuration at Startup

You can set the configuration file option `mdl_tree_cfg_file` to specify a Model Tree configuration file to be loaded at startup.

1. Click **Tools > Options**. The **Options** dialog box opens.
2. Under **Option**, type the configuration option `mdl_tree_cfg_file`.
3. Under **Value**, enter the path and file name of the Model Tree configuration file (`tree.cfg`, or a user-defined name).


The Model Tree configuration loads automatically the next time you start Pro/ENGINEER.

Parameters

To Remove a Parameter Subtype In the Model Tree

You can use the following procedure to remove a parameter if you added a parameter column to the Model Tree.

1. In the Model Tree, click **Settings > Tree Columns**. The **Model Tree Columns** dialog box opens.
2. In the **Displayed** box, select the parameter you want to remove.

3. Click . The parameter is moved back under the **Not Displayed** box.
4. Click **Apply > OK**.

To Create Parameters Using Model Tree Columns

1. Select an item in the Model Tree to which you want to apply the new parameter.
2. Click **Settings > Tree Columns**. The **Model Tree Columns** dialog box opens.
3. In the **Type** list, select one of **Model Params**, **Database Params**, **Feat Params**, **Annotation Elem Params**, **ECAD Params**, or **Mass Properties Params** type of parameters.
4. To create a new parameter for the selected item, under **Not Displayed**, specify a name for the new parameter in the **Name** field.
5. Click **>>** to add the new parameter to the **Displayed** list.
6. Click **Apply > OK**. The dialog box closes and the new parameter column is added to the Model Tree.
7. Click anywhere in the new parameter column opposite the selected item. If the parameter does not yet exist, the **Parameter Properties** dialog box opens.
8. Select a parameter format (**String**, **Real Number**, **Integer**, or **Yes No**) and click **OK**.
9. Type the required parameter value according to the parameter format that you have selected. For a real number variable, Pro/ENGINEER automatically completes the value to six decimal places. For a Yes No variable, Pro/ENGINEER automatically types a YES or NO.
10. From the **Units** list, select the required unit for the parameter.
11. Under **Access**, select the type of access, that is, full, limited, or locked that you want to give to the parameter.
12. Click **OK**. The parameter for the selected feature or model is added to the Model Tree.


Layers

Layer Basics

About the Layer Tree

In the Layer Tree, you can manipulate layers, their items, and the display status. Use the **View** menu to control the display of layers.

You can access the Layer Tree using the following methods:







- Click  or **View > Layers** to display the Layer Tree in the navigation window or in a separate dialog box. To view the layer tree in the Layers dialog box, set the `floating_layer_tree` configuration option to `yes`. If you set this option to the default of `no`, then the Layer Tree is displayed in the navigator window.
- Click **Show > Layer Tree** in the Model Tree navigator window. The **Layer** command is available while you are working below the highest-level functional menu of any mode (Drawing mode, Assembly mode, Part mode, and so forth) so that you can manipulate the layer display status or layer membership as required without having to return to the Part, Assembly, or top-level menu.

Note: The **Layer Tree** command is available only when `floating_layer_tree` configuration option is set to `no`.

Use the shortcut buttons on the Layer Tree toolbar, right-click on items listed in the Layer Tree, or use the following drop-down menus to perform layer related functions:

- **Show**—Display the Model Tree or items in different levels of layers, including nested and sub-model layers within your assembly.
- **Layer**—Create new layers, set rules, or edit properties with the Layer Properties dialog box. In addition, you can propagate, delete, remove, cut, copy, and paste items or layers.
- **Settings**—Add non-local items to layers being defined or sub-model layers.

The following symbols appear in the Layer Tree to indicate the type of layer associated with items:

-  Hidden Items—Items temporarily hidden from the Model Tree.
-  Simple layers—Items are manually added to the layer.
-  Default layers—Created using the `def_layer` configuration option.
-  Rules Layers—Layers defined primarily with rules.
-  Nested Layer—Layer that primarily contains other layers
-  Same Name Layer—Holds all same name layers of all components in the assembly.

Using Hidden Items Layer

When you manually hide items using the Model Tree or create asynchronous items, such items are automatically added to a layer called **Hidden Items**, if it already exists. If it does not exist, a new layer called **Hidden Items** is automatically created and the hidden items are added to it. This layer is always created at the top in the Layer Tree list.

Note: You cannot change the name or properties of the **Hidden Items** layer, or copy other layers to this layer.

You can select any item in the **Hidden Items** layer and perform the **Copy Item**, **Cut Item**, or **Paste Item** operations using the shortcut menu to add or remove items from this layer. When you unhide items using **Unhide All**, all the manually hidden items that are listed under the **Hidden Items** layer are removed and displayed on the graphics screen.

Note: The status of the **Hidden Items** layer is saved when you save the status of the other layers. But, you cannot delete the **Hidden Items** layer.

About Layer Names

Layers are identified by name. These names can be expressed in numeric or alphanumeric form, with a maximum of 31 characters. When layers are displayed in the tree, numeric layer names are sorted first and then alphanumeric names are sorted. Layer names in alphanumeric form are sorted alphabetically.

About Selecting the Active Model in the Layer Tree

You can select the active layer model from the:

- Drop-down list
- Model Tree
- Graphic Window


In Part mode, the active model is the part itself. You do not need to identify it before working with layers.

In Assembly mode, you may select the active model before working with layers.

In Assembly mode, if you are in the process of modifying a part, subassembly, or skeleton (that is, you have selected a sub-model and click **Activate** in the shortcut menu to set it as the active model), the Active Object is automatically set to the model being modified.

To Create a New Layer

You can create a new layer and then use the **Layer Properties** dialog box to edit the name and ID of the layer and add to or remove items from the layer.

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree appears.
2. Click **Layer > New Layer** or right-click within the Layer Tree and use the shortcut menu. The **Layer Properties** dialog box opens.
3. Type a name for the new layer, or accept the default name. The **Layer ID** field is empty.
4. Select what you want to include or exclude in the layer using the graphics window, search tool, layer tree, or rules tab.
5. Click **Ok**. The system creates a new layer with the specified name and adds the layer to the active object. As you add a layer, it is placed alphabetically in the

tree, and the tree scrolls vertically to the location of the layer that was added last. The **Layer Properties** dialog closes.

6. Click **Pause** to temporarily suspend selection of items to the layer and perform other operations. Click **Resume** to continue adding or removing items to the layer.
7. Click **Cancel** to close the dialog box. Changes to the layer properties dialog box are ignored.

Note: A layer can be defined as a non-ruled layer if it exists in the top level model.

Using a Template Layer Model

To use predefined ruled layers, you must define a template model with the required ruled layers specified. This model is designated as the template layer model. To access the template layer model set the `default_layer_model` config option to `<layer_model_name>`. Where `<layer_model_name>` specifies the model name that would be used to define predefined ruled layers for all models (`*.prt`, `*.asm`, and `*.drw`) in session. You must specify the name of the default layer models without an extension. You can create template layer models for a part (`*.prt`), assembly (`*.asm`), and drawing (`*.drw`). For example, if the `default_layer_model` config option is set to `DEF_LAYER_MODEL_NAME`, then for a part, Pro/ENGINEER will search for `def_layer_model_name.prt`, and for an assembly, Pro/ENGINEER will search for `def_layer_model_name.asm` in the working directory, workspace, and template directory.

If you create a new entity in a part, assembly, or drawing that matches the criteria of any of the predefined rules in a template layer model, a new layer is added to the current model automatically, and the new entity is added to that layer. However, if the template model is not available in the current session and if it is undesignated, then the ruled layers are no longer driven by the template layer model.


When the template layer model is not available, then the existing ruled layers become non-ruled layers and do not collect any items or objects. Additionally, a layer driven from the template layer model (or default layer config) will have the option of making the layer independent of the default layer model. Models that already have layers before a default layer model is used will always have layers that are independent of the default layer model if rules are already defined for the previous layers.







To Activate a Layer

You can control the layer on which drawing items are placed by setting the layer as active. After the layer is activated, any drawing items created, shown, or pasted in the drawing are placed on the layer. If a layer is activated in a 3D model, all the newly created 3D elements are added to this layer.

A layer remains active for the duration of the session until you deactivate it, change to another layer, or end the session.

If you manually create a layer, you must manually activate it. You can manually activate a layer by doing the following:

1. On the Model Tree, click **Show > Layer Tree**. Any existing layers are listed.
2. Right-click the layer that you want to activate and click **Activate**.  appears next to the activated layer. Any drawing items created, shown, or pasted in the drawing are placed on the active layer. The following table lists different states of the default and nested layers.

Default Layers		Nested Layers	
	Default layer is activated		Nested layer is activated
	Default layer is isolated and activated		Nested layer with hidden components is activated
	Default layer with hidden components is activated		Nested layer is isolated and activated

Note:

- You can automatically activate a default layer by creating it with the `def_layer` configuration option. If you set the `def_layer` configuration option to `layer_all_detail_items`, you can activate 2D elements. To activate 3D elements, set the `def_layer` configuration option to `layer_3d_items`.
- If an active layer exists and you activate another layer, then the first active layer is deactivated.
- If you activate a layer that has the same name as another layer, then each layer with that name is activated.
- If the top-level drawing layer is Layer X and the part layer is also Layer X, then activating Layer X in the drawing activates Layer X in the part. However, Layer X in the drawing contains all detail items while Layer X in the part contains all 3D entities.
- If you open a new model or drawing in the current Pro/ENGINEER session, then no layer is active for that new model or drawing.
- If Drawing A has an active layer and you open Drawing B, then no layer is active in Drawing B. However, if you switch back to Drawing A, then the active layer is still displayed.
- If you open a model in a new window, then no layer is active for that model.
- If you switch to a model that did not have an active layer, then no layer is active for that model.
- Pro/ENGINEER remembers the active layer status for the top-level model layer and any existing same name layers of submodels and retains the status in the active window. However, if you open a submodel in a new window, then no layer is active for that submodel.
- If a submodel does not have a same name layer as that of the top-level model layer that is active, a new layer is created but not activated. The submodel does not have an active layer.

To Deactivate a Layer

A layer remains active for the life of the session or until you deactivate it. You can deactivate layers in the following ways:

- Delete the setting for the `def_layer` configuration option.
- Rename the active layer. The layer is automatically deactivated.

Deactivate the Layer Manually

You can also deactivate the layer by doing the following:

1. On the Model Tree, click **Show > Layer Tree**. All existing layers are listed.
2. Right-click the layer you want to deactivate, and click **Deactivate**. The layer is deactivated. Any items created or shown after this point will not be automatically placed on the layer.

Note: If the settings for the `def_layer` configuration option are changed for the layer that was activated manually, you cannot deactivate such layers.

Managing Layers with the Same Name

Items on layers with the same name are treated similarly. You can set the display of model layers to mirror the display of drawing layers with the same name. For example, if you blank the drawing layer `datums`, Pro/E also blanks all items on the `datums` layers for the model components.

If the model is an assembly and you add a component with a layer of the same name, Pro/ENGINEER automatically sets the display of any items on a model layer named `datums` to be the same as the drawing layer. To keep one component `datums` layer from being affected, put the desired layer directly on another drawing layer.

To set the display of layers with the same name, do one of the following:

- On the Model Tree, click **Settings > Drawing Layer Status**, and select the second check box (Change display of model layers with the same names in drawing only) in the **Layer Status Control** dialog box.
- Set the `draw_layer_overrides_model` drawing setup file option to `yes`. If you set this option to `no` (the default), Pro/ENGINEER ignores nondrawing layers when you set the display status of layers in the drawing model with the same name.

With the exception of ruled layers, if you activate a layer that has the same name as another layer, each layer with that name is activated. For example, if the drawing and the drawing model have a layer named `my_layer`, activating one of these layers causes the other to be activated. Designated items supported in both modes (driven dimensions, surface finish symbols, and geometric tolerances) are placed on both layers, while unsupported items are only placed on the drawing layer.


In an assembly, you cannot activate a layer that exists only in a submodel. However, if you activate the same name layer at the top level, all submodel layers are activated.

To Delete a Layer


You can delete the selected layer or layers from their objects.

1. Select one or more layers in the Layer Tree.
2. Click **Layer > Delete Layer** or right-click the layer name and click **Delete > Layer**. The **CONFIRMATION** dialog box opens.
 3. Click **Yes** to delete the selected layer or layers or click **No** to cancel the deletion.


To Copy an Item Between Layers

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree opens.
2. Right-click the items to copy and click **Copy Item**. The item or items are copied to the clipboard.
3. Select one or more existing layers or click **New Layer** to create a new layer.
4. With all destination layers highlighted, click **Layer > Paste** (or use the right mouse button). The item or items are placed on the layers.

To Copy All Items on a Layer

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree opens.
2. Right-click the layer whose contents you want to copy. The shortcut menu appears.
3. Click **Select Items**.
4. Click **Copy Item**. The contents of the selected layer are copied to the clipboard.
5. Select the layer on which you want to paste the copied items, or create a new layer by using **Layer > New Layer**.
6. Click **Layer > Paste**. The items are copied onto the selected layer.

To Set Properties for a Layer

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree opens.
2. Click **Layer > Layer Properties** or right-click the layer name and click **Layer Properties**. The **Layer Properties** dialog box opens.
3. The current layer name is displayed. You can select it and change the name.
4. The current ID is displayed in the Layer ID box. You can change the ID.

The following tabs allow you to set properties for layers:

- Contents
- Rules
- Notes

Layer properties in the dialog box always correspond to the selected layer. The Layer Tree scrolls vertically so that the selected or expanded layer is always visible. If you select more than one layer, selection and properties step down the list, starting with the highest layer.

Note: If you want to change the names of all same name layers in a top level assembly, select the top level name node, and right-click Rename in the shortcut menu. This command is only available for name nodes.


To Obtain Information about a Layer

Right-click one or more layers in the Layer Tree and click **Layer Info** in the shortcut menu. An Information window opens, showing layer display status for each selected layer.


The information reflects the current status of the Layer Tree even with respect to drawing setup file options such as `ignore_model_layer_status`.

To Set a Layer Interface ID

In the **Layer Properties** dialog box, you can set an interface layer ID. With this ID, a layer is recognized when you are exporting the file to a different format, such as IGES.

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree appears.
2. Right-click one or more layers in the Layer Tree. The shortcut menu appears.
3. Click **Layer Properties**. The **Layer Properties** dialog box opens. If more than one layer is selected, the **Layer Properties** dialog boxes open sequentially for each selected layer.
4. Type a new interface ID for the selected layer or layers of the current object in **Layer ID** box. Enter an integer in the range of 1 through 99,999,999.
5. Click OK to close the **Layer Properties** dialog box.

To Remove a Layer Interface ID

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree appears.
2. Select one or more layers in the Layer Tree. If more than one layer is selected, the **Layer Properties** dialog boxes opens sequentially for each selected layer.
3. Remove the Layer ID from the box.

- Click OK to close the **Layer Properties** dialog box.

Layers Remove Item Commands

You can use **Layer > Remove Item** and **Layer > Remove All Items** in the Layer Tree to remove a selected item or items from all layers, from all 3-D layers, or from all drawing layers. When you use these commands to remove items from a layer, the **CONFIRMATION** dialog box prompts you to confirm before the system removes the selected item or items.

About Layer Display

Use the Layer Tree to view layers in a part or assembly, view items assigned to a layer, and control how layers are displayed in the model. After you change the layer display status, click **View > Visibility > Save Status** to save your changes. If you make further changes to the display status after you save it, you can revert the display status to the saved status by clicking **View > Visibility > Reset Status**.

Hiding and showing layers do not affect model geometry, because these functions only affect features that do not affect mass properties, such as datum planes, axes, and coordinate systems. You can select datum axes that are created with geometry to add to layers either graphically or using the search tool.

You can set the display status of a layer to one of the following:

- **Shown Layers**—(Default) Displays the selected layer.
- **Hidden Layers**—Hides the selected layer.
- **Isolated Layers**—Displays the selected layers and treats all non-isolated layers as hidden.
- **Layers Shown in Hidden Line**—(Assembly and Manufacturing mode only) Hides the components in the hidden layers according to the environment settings for hidden-line display. Other items on those hidden layers are not affected. The rules are as follows:

Environment Setting for Line Display	Hidden Layer Component Display
Wireframe	Shown
Hidden Line	Hidden Line
No Hidden Line	Hidden
Shading	Shown

In Assembly mode, if you set a specific layer or layers to Isolate, Pro/ENGINEER hides all components. In addition, Pro/ENGINEER hides all other items that are assigned to any non-Isolate layer.

Note: In Assembly mode, if you have components on layers that you subsequently set to **Hide**, Pro/ENGINEER hides all nongeometry items (datum planes, datum axes, feature axes) even if they are also on the displayed layers.

In all other modes, if you set a specific layer or layers to **Isolate**, Pro/ENGINEER also hides all other layers. However, Pro/ENGINEER continues to display items that are not assigned to any layer.

You can hide the following types of items in any modeling mode (such as Part, Assembly, or Manufacturing):

- Datum features, such as planes, axes, curves, and points
- Feature axes, such as the axes for holes on the layer
- Cosmetic features
- Quilts

Note: Isolate has priority over Hide. Therefore, if a member is on two layers, one set to Isolate and the other set to Hide, the member is shown. However, if a feature consists of several entities (for example, datum curves), individual entities are not shown if the entire feature is in a layer that is set to Hide, even if the entities themselves are in layers that are set to Isolate.


The only features on a layer that layer display operations affect are datum and surface features. Solid geometry is not affected. For example, in Part mode, if you put a hole on a layer and hide the layer, only the datum axis of the hole is hidden. You can hide the hole itself only by suppressing it directly or by suppressing its layer. The only exception to this rule is that, in an assembly solid, you can hide components.

If you add a new member to an assembly, and a default layer for it does not exist, Pro/ENGINEER does not automatically add it to an existing layer. If a layer in the assembly is already set to Isolate when the member is added, the new member is not shown until you either add it to a layer that is explicitly isolated and repaint the display or clear all the isolated layers. In the latter case, which is the recommended action, all the members of the assembly are shown.

In Assembly, Show affects the level of the member and levels above it; Hidden affects the level of the member and levels below it.

To Set Layer Display Status

You can set the display status of layers using the **Save Status** command, by setting configuration file options, or by using the layer file.

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree appears.
2. Click **Layer** on the Layer Tree toolbar and select one of the following commands to set the display status:

Hide—Hides selected features, components, and layers.

Unhide—Shows selected features, components, and layers.

Isolate—Isolates the selected layers and treats all non-isolated layers as hidden.

In the Layer Tree, icons next to a layer name indicate whether items on that layer are visible, isolated, or hidden.

Hidden Lines—(Assembly and Manufacturing mode only) Hides the components in the hidden layers according to the environment settings for hidden-line display.

Alternatively, set the status to show or display the layer by right-clicking the selected layer and clicking **Layer > Hide** and **Layer > Unhide**.

3. In 3D modes, repaint the screen to see any layer status changes take effect.

In Drawing mode, the graphics display is automatically repainted.

4. If you want to save the display status, click **View > Visibility > Save Status**. Pro/ENGINEER saves the current layer display status.

Note: By default, Pro/Engineer does not save the layer display status unless you save the layer status before saving the object.

5. If you want to change the display status after you have saved a status, click **View > Visibility > Reset Status** to roll back the display status.

To Create Sublayers

A layer can contain a number of other layers that are considered sublayers. When a layer has been included as a sublayer, selecting the name of the top layer automatically selects the sublayers as well.

Sublayers can be associated with different layers. Thus, the sublayer can be associated with both the DETAIL layer and any other layer.

Multilevel layer structure can be created in one of two ways:

Include a layer as an item on another layer

1. Select the layer which is the sublayer from the Layer Tree and right-click **Copy Item** in the shortcut menu to copy the selected layer.
2. Select the layer that you want to add the sublayer and right-click **Paste Item** in the shortcut menu. This adds the first layer as a sublayer to the second layer.

Note: You can also open the layer properties of the layer where you want to add the sublayer. In the **Layer Properties** dialog box, you can select other layers to be added to the top level layer.

Deleting a row in the layer file using **Settings > Setup File > Edit** does not delete this layer from the model. However, adding a row by specifying a new layer name automatically creates a new layer with that name.

To Control the Display Status of Sublayers

To control the display status of items in lower-level models, you must first put them in layers in their native models.

Usually, then, you would have to navigate down to each model in turn to change the display status of its layers.

However, if you assign common names to layers throughout the Model Tree, changing the display status of an upper-level layer simultaneously changes the status of those layers of the same name in layers below the one changed. Note that layers of the same name in parallel branches or above the selected one are not affected.

Layer Items

To Add Items to a Layer

1. In the Layer Tree, right-click the layer to which you want to add items.
2. Click **Layer Properties** on the shortcut menu. The **Layer Properties** dialog box opens.
3. Select the required items to place on the selected layer. Select items in the graphics window, Model Tree, or use the search tool.

If you are in Assembly mode and you select an item to add to a layer and the item does not belong in the top level model, then the **Place External Item** dialog box opens. Use this dialog box to add such items to a layer.

Note: You can add datum axes and coordinate systems to layers as entities either through the Pro/ENGINEER graphics window or by using the search tool. Selecting a layer highlights the entities on that layer.

To Add Items to Layers Using Rules

You can also add items to layers by using the **Rules** to create rules that define the items of a layer. The layer can either store only the items gathered by the rule, or it can store the items and the rule itself. If the rule is stored, then the layer will update its items with new items that meet the rule criteria.

1. Open the **Layer Properties** dialog box of a new or existing layer by right-clicking any item within the Layer Tree.
2. Click the **Rules** tab. The layers are dependent and non-associative by default. If the layer is independent and the **Rules Enabled** check box is selected, then the layer is updated each time an item matches the criteria and is added to the model.
3. Click **Options** to define the rule settings by selecting one of the following:
 - **Associative**—Adds all the existing and newly created items to the **Query Builder** box that currently meet the rule criteria.

- **Rules Enabled**—Controls whether items matching the rules can be automatically added to the layer. The newly created items are added either to the default layers or to the user-defined layers based on the rule criteria they meet.
 - **Independent**—The layer becomes independent and rules are created using **Edit Rules**.
4. Select **Edit Rules**. The **Rule Editor** dialog box opens.
 5. Use the **Rule Editor** dialog box to create the rules for gathering items. For example, type `Name is equal to A*` to gather all component names starting with the letter A.
 6. Click **OK** in the **Search Tool** dialog box.
 7. Create a rule in the Rules area of the **Layer Properties** dialog box.
 8. Select **Contents** to see items added to the layer. The icon in the Status column of the Model Tree indicates how the item was included in the layer (by rule or manually). The Contents page is updated only when you close the **Layer Properties** dialog box.

Note:


- To define a rule the layer must be independent.
- You can add one item to multiple layers by adding the item to one layer first using its **Layer Properties** dialog box. After one layer contains the item, you can select the item in the Layer Tree and right-click **Copy Item** on the shortcut menu. Select all the other layers to copy to this item and use **Paste Item** on the shortcut menu to copy the item to all the selected layers.

To Add Items to Multiple Layers

You can add one item to multiple layers by adding the item to one layer first using its **Layer Properties** dialog box. Once one layer contains the item, complete the following steps.

1. Select the item in the Layer Tree, and right-click **Copy Item** in the shortcut menu.
2. Select all the other layers to copy to this item.
3. Right-click **Paste Item** in the shortcut menu to copy the item to all the selected layers.

To Remove Items from a Layer

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree appears.
2. Select one or more items under layers in the Layer Tree. Items do not have to be in the same layer.

3. Click **Layer > Remove Item**. You will be prompted to verify that you want to remove the selected items.

Note: You can also remove items from a layer when the Layer Properties dialog is open by selecting items in the Contents list, and using the Remove button.


You can also remove all items from one or more layers.

1. Select one or more layers in the Layer Tree.
2. Click **Layer > Remove All items**. Pro/ENGINEER prompts you to confirm the removal of all items.
3. The selected items are removed from the selected layer or layers.

To Purge Items from a Layer

1. Click **Show > Layer Tree** on the Model Tree navigator window. The Layer Tree appears.
2. Select one or more layers in the Layer Tree.
3. Click **Layer > Remove All Items**. The system prompts you for confirmation of the purge operation.
4. Click **Yes** to remove all the items in the appropriate objects from the selected layer or layers, or click **No** to cancel the deletion.

To Copy Items from One Layer to Another Layer

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree opens.
2. Use the **Copy** and **Cut** commands on the Layer Tree Edit menu to move items from one layer to another in the active model.
3. Select one or more layer items in the Layer Tree.

NOTE: A shortcut to select all items of a single layer is to first select the layer, and then right-click **Select Items** in the shortcut menu.

4. Click **Layer > Copy** and **Layer > Cut** or use the shortcut menu.
5. Specify destination layers for the item or items. Only layers from the parent object of the item are available.
6. Click **Layer > Paste** or right-click **Paste Item** in the shortcut menu to copy the item to the selected layers.

Notes:

- If the item is not hidden, it is highlighted in red.
- If the item is a hidden geometry feature (such as a hole or round), or a hidden component, the geometry is still visible on the screen, but remains highlighted in red. Pro/ENGINEER displays that the item is hidden in the message area.

- If the item is hidden and is not visible on the model (such as a datum or axis), the item is not highlighted on the model.
- You can select items that are shown or hidden. However, you cannot select suppressed items.
- If the items are visible on the model (whether or not they are hidden), they are highlighted in red after you select them.
- If you select a hidden component in an assembly, the component appears on the assembly highlighted in red.
- If items such as datums and axes are hidden, they are not visible on the model and so are not highlighted.

To Add External Components to Layers

You can add external components to top-level assembly layers. External components are located below the first level of the top-level assembly. For example, components within a subassembly are considered external components. You can either add all external components to the top-level layer or add external components to sub-model layers.


Note: You cannot add external components from one branch to another branch of an assembly structure.

Two methods to add external components to top-level layers are available.


- Click **Settings > Item Selection Prefs > Favor Current Layer**. Select the **Favor Current Layer** checkbox to automatically add all selected sub-level components to the top-level layer.
- To individually add external components to the top level layer:
 1. In the Layer Tree, right-click the layer to which you want to add items. The shortcut menu appears.
 2. Click **Layer Properties** in the shortcut menu. The **Layer Properties** dialog box opens.
 3. Select an external component to include in the top-level layer from the graphics window or Model Tree, or use the search tool. The **Place External Item** dialog box opens listing layers of the sub-assembly to which the component belongs.
 4. Select **Place component on current layer** to add the selected external component to the top-level layer. Otherwise, select one or more layers of the sub-assembly model and then select one of the following options to include the components so they become local to that sub-assembly:
 - **Place item on all selected layers**—Places the selected component on all layers in the active model selected in the dialog box and closes the dialog box.
 - **New Layer**—Creates a new layer in the active model and places the selected component on this new layer.

- **Ignore Item**—Ignores the selected component.
- **Ignore All**—Ignores all the selected components that are outside the active model.

To Delete a Feature or Component Using the Layer Tree

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree appears.
2. Select the features or components to be deleted, right-click, and click **Select Items** on the shortcut menu.
3. Click **Edit > Delete > Delete** to delete the selected features or components.

To Suppress a Feature or Component Using the Layer Tree

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree appears.
2. Select the features or components to be suppressed, right-click, click **Select Items** on the shortcut menu.
3. Click **Edit > Suppress > Suppress** to suppress the selected features or components.

Default Layers

About Default Layers

Pro/ENGINEER can automatically place certain types of items on specified layers when they are created. You can establish up to 32 default layers in a model. To do this, set the configuration file option `create_numbered_layers` to `yes`.

You can then add these items to layers automatically as you create the items by setting the configuration file `, def_layer <type-option layername>`, where `type-option` is the item type, and `layername` is the name that you assign to the layer. Whenever you create an entity of a given item type, Pro/ENGINEER automatically adds it to the specified default layer.

About Creating Layers for Subtractive Features

Subtractive features (cuts, holes, and so forth) in an assembly must be created on intersected models. A new layer is created if this condition is met, and the default layer is defined in the configuration option.

When you create a subtractive feature, the visibility of the feature must be set to a level other than the top level. For example, in assembly A with parts B and C, if you create a hole in assembly A that goes through part B, you must go to the intersection section and specify that the cut be visible at the B part level. This action

adds the feature as an assembly cut to part B, and a default layer is created in part B.

Valid Options for Item Type in a Default Layer

The following table lists valid options for specifying item type in the `def_layer` configuration file option.

Option	Item Type
<code>layer_assem_member</code>	assembly members
<code>layer_feature</code>	all features
<code>layer_axis</code>	features with axes and cosmetic threads
<code>layer_geom_feat</code>	features with geometry
<code>layer_skeleton_model</code>	skeleton model
<code>layer_nogeom_feat</code>	features without geometry
<code>layer_cosm_round_feat</code>	Dieface cosmetic round feature
<code>layer_cosm_sketch</code>	cosmetic sketches
<code>layer_surface</code>	surface features
<code>layer_datum</code>	datum plane features
<code>layer_datum_plane</code>	datum plane
<code>layer_ext_copy_geom_feat</code>	external copy geometry feature
<code>layer_point</code>	datum point features
<code>layer_copy_geom_feat</code>	copy geometry
<code>layer_curve</code>	datum curves
<code>layer_csys</code>	datum coordinate systems
<code>layer_curve_ent</code>	curve entities
<code>layer_dim</code>	all dimensions
<code>layer_parameter_dim</code>	parameter dimensions
<code>layer_driven_dim</code>	driven dimensions
<code>layer_draft_dim</code>	draft dimensions
<code>layer_refdim</code>	all reference dimensions

Option	Item Type
layer_part_refdim	part reference dimensions
layer_draft_refdim	draft reference dimensions
layer_note	drawing notes
layer_gtol	geometric tolerances
layer_ribbon_feat	ribbon feature
layer_solid_geom	solid geometry
layer_symbol	symbols
layer_sfin	surface finish symbols
layer_thread_feat	cosmetic thread feature
layer_trim_line_feat	Dieface trim line feature
layer_weld_feat	weld feature
layer_detail_item	draft items that include datums, notes, balloons, GTOLs, symbols and groups
layer_draft_entity	all draft entities
layer_draft_constr	draft construction entities
layer_draft_geom	draft geometry entities
layer_draft_hidden	draft hidden entities
layer_draft_others	all other draft entities
layer_draft_grp	draft groups
layer_draft_dtm	draft datums
layer_quilt	quilts
layer_dgm_wire	diagram wires
layer_datum_point	datum points
layer_dgm_conn_comp	diagram connector components
layer_dgm_highway	diagram highways

Option	Item Type
layer_dgm_rail	diagram rails
layer_dwg_table	drawing tables
layer_assy_cut_feat	assembly cut
layer_chamfer_feat	chamfer
layer_corn_chamf_feat	corner chamfer
layer_cut_feat	cut
layer_draft_feat	draft
layer_hole_feat	hole
layer_protrusion_feat	protrusion
layer_rib_feat	rib
layer_round_feat	round
layer_shell_feat	shell
layer_slot_feat	slot
layer_snap_line	snap lines
layer_comp_design_model	design model configuration file option <code>DEF_LAYER</code> or "SetDefLayer" inside the layer interface
layer_comp_workpiece	workpiece configuration file option <code>DEF_LAYER</code> or "SetDefLayer" inside the layer interface
layer_comp_fixture	fixture configuration file option <code>DEF_LAYER</code>
layer_Solid_geom	solid geometry in a part
layer_intchg_funct	functional interchange component
layer_intchg_simp	simplified interchange component
Layer_nongraph_annot_elem	graphical annotation element

Using Layer Status Files

About Using Layer Status Files

The file format for a layer file is as follows:

- The first line of a layer file must always begin with

```
!LAYER SETUP nnn
```
- The number `nnn` is a version number that can change. Verify this number in the trail file for the model before you start to edit the layer file outside of Pro/ENGINEER.


The following rules apply to the layer file format:

- The first column must always contain the layer name.
- There must not be any blank lines in the file.
- The display status is optional. To specify the display status, enter one of the following: `HIDE`, `UNHIDE`, and `ISOLATED`.
- Interface ID is optional. A number sign (`#`) precedes the ID. You must set this if you intend to export layer information.
- You can specify any number of sublayers.

If you use a system editor to create the file, make sure that there is at least one space between each entry.

If you are using Pro/TABLE, put only one entry in each cell

To Save Layer Display Status Settings as a File

1. Click  or **Show > Layer Tree** in the Model Tree navigator window. The Layer Tree appears.
2. Click **Settings > Setup File > Save**. The **Save a Copy** dialog box opens
3. Use **Look In** in the Search Tool (**Edit > Find**) to select the directory to which to save the status file. You can accept the default `.pro` file name or enter a file name. The default file type is a configuration file.
4. Specify a file name and click OK. The system saves the layers of the active model and their current display status to a file.

Saving the information to a file makes the layers and their display status available for retrieval or to be used by other objects.

Note: If you enter the name `layer_file`, in response to the prompt, the actual file name is `layer_file.pro`.

To Open a Layer Status File

1. Click **Show > Layer Tree** to open the **Layer Tree**.

2. Click **Settings > Setup File > Open** to apply the file to the model.

To Edit a Layer Status File

1. Click **Show > Layer Tree** to open the **Layer Tree**.
2. Click **Settings > Setup File > Edit** to manipulate a layer by modifying parameters in the layer file in Pro/TABLE.

Using Default Layer Status Files

Layer status files let you:

- Change the display status of a number of layers and sublayers in one operation.
- Associate sublayers with other layers, by editing the table.
- Define a default setting for layer status whenever you start Pro/ENGINEER.

You use Pro/TABLE to edit layer files. When the file is saved, the layer display status is saved with it. If you later end the session, then start a new one and retrieve the model, the system displays the model in the settings that you last saved.

A `layersetup.pro` file can be used to establish company standards for layer names and layer structure. The system retrieves this file whenever you begin to create a new object with Pro/ENGINEER.

You can also create a customized version of `layersetup.pro` and store it in your startup or home directory. When you start creating the object, Pro/ENGINEER searches for the `layersetup.pro` file in directories in the following order:

- Current
- Home
- Pro/ENGINEER
- `loadpoint/text`

This ensures that the company standard is not overridden because later settings overwrite earlier ones.

A default `layersetup.pro` file is supplied with Pro/ENGINEER in the `loadpoint/text` directory. It contains 32 layers with names 1 through 32 inclusive.

Example: A Typical Layer File

```
! LAYER SETUP 1061
! Comment lines must begin with a "!"
!
! The first column must always contain the layer name.
! As an option, entering one of the following can specify the display
status: HIDE, UNHIDE, and ISOLATED.
```



```


! Interface ID can optionally be specified by preceding the ID with #
! You can specify as many sublayers as you want.
!
! eg.
! DATUMS SHOWN #100 DTM_PLANES AXES POINTS COORD_SYSTEMS
!
! NAME | STATUS | #ID | SUB-LAYER NAMES
!
LAY0001 SHOWN
LAY0002 SHOWN
LAY0003 HIDDEN

```

Using the Layer Tree

To Include or Define Rules in Layers

1. Specify rules in layers or include items defined by the rules.
2. Right-click the layer name and click **Layer Properties**. The **Layer Properties** dialog box opens.
3. Click **Rules**. The **Rules Enabled** option under **Options** is selected by default and allows you to update the layer each time an item that matches the criteria is added to the model.
4. Select **Independent**, the **Edit Rules** option is enabled. The layers are dependent and non-associative by default.
5. Click **Edit Rules**. The **Rule Editor** dialog box opens.
6. Search and select rule conditions using the **Rule Editor**.
7. Click **Preview Results** to preview rule condition results or click **OK**. The rule conditions appear in **Rules**.
8. Click **OK**. Layers and their associated items appear in the Layer Tree.

Note: The symbol  appears in the Layer Tree indicating the layer is defined primarily with rules. Additionally, rules can be combined using AND and OR operations.

To Nest Layers within Layers

Use the **Layer Properties** dialog box or copy and paste commands to nest layers within layers.

1. In the Layer Tree, right-click the layer name. The shortcut menu appears.

2. Click **Layer Properties**. The **Layer Properties** dialog box opens. The **Include** button is selected by default.
3. Select the layer to nest from the Layer Tree. This layer name appears in the **Contents** list. However, items of the nested layer are not listed.

Note: To show items of nested layers in the layer tree, click **Settings > Items on Nested Layers**.

Alternatively, you can use copy and paste commands to nest layers.

To Display and Select Items in the Layer Tree

Use the following commands in the Layer Tree to display and select items:

Under **Show**:

- **Highlight Geometry**—When selected, items in the Layer Tree are highlighted in red in the graphics window. The setting is remembered in the session independently for 3D modes and in Drawing mode. That is, settings do not return to the default, so that when you open the dialog box again, the Highlight setting is the same as it was the last time the dialog box was closed.
- **Find**—Search for layers containing or controlling specific items.

Under **Settings**:

- **All Submodel Layers**—Shows layers in the active object and all associated submodels.
- **Submodels layers if in the Active Model**—Shows active object layers in all the associated submodels.
- **No Submodel Layers**—Shows layers in the active object only.
- **Layer Items**—Shows all layer items in the Layer Tree.
- **Items on Nested Layers**—Shows items of nested layers in the Layer Tree.

To Display a Layer Status File

1. Click **Show > Layer Tree** to open the Layer Tree.
2. Select a layer.
3. Click **Settings > Setup File > Display**. An INFORMATION WINDOW dialog box showing the current display status and contents of selected layers. Using the File menu in the Information window, you can save this information to a tmp.pro default file or save it as a text file with a specified file name.

To Expand or Collapse the Tree Menu

Click **Show** and select one of the following:


- Expand All
- Collapse All


Search Tool

About the Search Tool

The **Search Tool** allows you to find entities within your model without selecting from the graphics window or Model Tree. You can specify rules using the Attributes, History, Status, and Geometry tabs to help find specific information. You can create a layer of the found items, or select all or a subset of the found items by transferring them to the active selection section of the search tool.

You have the option to find items that are not valid for the current selection. For example, if you want to find a horizontal reference for a sketching plane, the collector filters out all datum planes not orthogonal to the sketching plane. **Filtered Selections**, on the **Options** menu allows you to apply the filters.

The **Search Tool** is available by clicking  on the Pro/ENGINEER toolbar or by clicking **Edit > Find** menu. You can select items by attributes such as name, ID, or number, annotations, or the last feature you created and create rules using wildcards. For example, *Name is equal to A** searches for item names that begin with the letter A. The Criteria section displays the various comparison options, values, categories and other criteria based on the rule set for searching items.


Search results and corresponding locations are listed under **Items Found** of the **Search Tool** dialog box. The results are also highlighted in the Model Tree and graphics window. To select items, use  to transfer items from **Items Found** to the Items Selected section of the **Search Tool** dialog box.

Note: If you are working outside a tool (no action specified), selected items are not cleared when you open the **Search Tool**. For example, if you select the feature and click **Find Now**, that feature is listed in the item selected section of the search tool. If the **Search Tool** is used while a tool is active, clicking the **Close** button appends only the search results and overrides any current selection.

The rules and values in the **Search Tool** reappear when a search is performed and you reopen the **Search Tool** to perform another search.

The **Search Tool** dialog box contains the following tabs and buttons:

- **Attributes**—Search for items based on their attributes (Name, Type, Parameters, Size, Description, or Property)
- **History**—Search for items based on their creation order or ID (including Last Feature, Failed Feature, and All, when applicable)
- **Status**—Search for items based on their status (Regeneration, Layer, Display, Parent/Child, Copied Refs)
- **Geometry**—Search for items based on geometrical relationships (Zone, Distance, Exterior Components)
- **Look for**—Specify the type of item to search for, for example, feature, component, dimension, and so on.

- **Look by**—Create a more detailed query. For example, if you set **Look for** to Feature and set **Look by** to Dimension. The rules apply to dimensions but the found items are features that own the found dimensions.
- **Look in**—(Available in Assembly and Drawing mode only) Select the active model. If the scope of your selection is already defined, this field is not displayed. The arrow button allows you to select the desired model from the graphics window or the Model Tree. Click the **Include submodels** check box to search the sub-models of the Look In model.
- **Find Now**—Search by identified rules and display results under **Items Found**. If only one item is in the found list, it is automatically highlighted in the found list. If only one item is to be selected, select it from the list, and click  to append it to the selection bin or double-click the item. You can also middle-click to add it to the selection bin and close the dialog box.
- **New Search**—Start a new search.
- **Options**—Set options, build, and save queries. Save the components gathered by a rule or set of rules into a layer using **Save Query**. This prompts for a new layer name, and creates a new layer in the layer tree. Build a query by combining rules that use Boolean **AND** and **OR** operations.

About Rules and Criteria for Searching Entities

You can search for an entity in a model by using the Search tool. To refine the search, use the different rules and their corresponding criteria available in the Search Tool. These rules are available for each of the **Attribute**, **History**, **Status**, or **Geometry** tabbed pages in the **Search Tool** dialog box. The **Criteria** section displays the various comparison options, values, categories and other criteria based on the rule set for searching items. These rules and criteria change according to the selected **Look for** and **Look by** values.

You can refine the search by building a query with a combination of one or more than one rule and the corresponding criteria in the required tabbed page. You can also create rules using wildcards. For example, the rule `Name is equal to A*` searches for entity names that begin with the letter A.

Example: Using Rules and Criteria for Comparing Entity Names

You can search for items by name in the **Search Tool** dialog box. To search for an entity by name, click the **Attributes** tab in the **Search Tool** dialog box and under **Rules**, select **Name**. Under **Comparison**, specify a condition such as, is equal to, is not equal to, is less than, is greater than, is less than or equal to, and is greater than or equal to, for comparing the name specified in the criteria with other entity names in the model.

In the **Search Tool** dialog box, under **Criteria**, you can set the **Value** in one of the following ways:

- Specify the exact name that you want to search for. Pro/ENGINEER searches for items in the model whose names match exactly with the specified name.

- Use wildcards. Pro/ENGINEER searches for all items whose names contain the specified character. For example, the rule `Name is equal to A*` searches for entity names that begin with the letter A. Similarly, the rule `Name is equal to B*A` searches for entity names that begin with the letter B and end with the letter A.
- Use asterisk only. Pro/ENGINEER searches all the items that have a default feature name and for items for which you have explicitly assigned a feature name in the model. For example, the rule `Name is equal to *` searches for all entities that have a feature name defined in the model.

While searching for an entity by its name, Pro/ENGINEER compares the names character by character. However, for groups of digits, the values of the groups are compared.

Some examples of how Pro/ENGINEER compares the names, while searching for an entity by its name, are:

- If the model includes datum features named DTM1, DTM2, DTM003, DTM0004, and DTM005 and you have created the following rules to build a query:

Rule 1: `Name is greater than DTM1`

Logical operator: AND

Rule 2: `Name is less than DTM005`

The result of the search includes DTM2, DTM003, and DTM0004.

- If you create a rule `Name is equal to DT*`, the result includes DTM1, DTM2, DTM003, DTM0004, and DTM005 along with the other entities that have their names starting with DT.


A few more examples of how Pro/ENGINEER compares the names, while searching for an entity by its name, are:

- The name `abc` is less than the name `abcd`.
- The name `abcd` is less than the name `abcde`.
- The name `12ab` is greater than the name `2a`.

Similarly, you can use the rule-criteria combination to search for entities based on their attributes, history, status, and geometry properties.

To Search for Objects in the Model Tree

1. Click **Edit > Find**. The **Search Tool** dialog box opens.
2. If you are searching an assembly or a drawing, select an item from the **Look in** and **Look by** lists, and click the **Include submodels** check box, if required.
3. Click the tab to select the criteria by which you wish to search (**Attributes**, **History**, **Status**, **Geometry**).

4. Click **Find Now**. The found items and corresponding location list in the **items found** section of the **Search Tool** dialog box. The results are also highlighted in the Model Tree and graphics window.
5. To select items, Click  to transfer items from Items Found to the Items Selected section of the **Search Tool** dialog box.
6. Click **Close**.

To Build a Query in the Search Tool

You build a query in the **Search Tool** to combine rules for a search you want to perform. Use the following procedure to build a query:

1. Click **Edit > Find**. The **Search Tool** dialog box opens.
2. Click **Options > Build Query**. The **Query Builder** dialog box opens.
3. Select an item type from the **Look for** list.
Note: To enable the **Geometry** tab, **Component** must be selected in the **Look for** list.
4. Select a model in the **Look in** list, if relevant.
5. Select from the available tabs and list boxes to customize the rules.
6. Click **Add New** to add the defined rule to the condition builder.
7. Create a new rule using the tabs and list boxes to be combined with the previous view using **and** or **or** operations.
8. Click **Add New**. The next rule is added to the condition builder.
Note: The default operator is **or** when adding new rules. The operator can be changed to **and** by clicking the **Operator** cell and selecting from the list.
9. When all the desired rules are combined, click **Find Now**. The search results from the combined query and corresponding locations are listed in items found section of the **Search Tool** dialog box.

Note: You can click **Options > Save Query** to save this query for later use.

To Save Query in the Search Tool

You can save a query once you save it under a specified layer name.

1. Click **Edit > Find**. The **Search Tool** dialog box opens.
2. Click **Options > Build Query**. The **Query Builder** dialog box opens.
3. Set the rules for the query.
4. Click **Options > Save Query**. The **Save Rules** dialog box opens.
5. Enter a layer name. The layer is created and a message displays `Current Query has been saved in layer [layer name]`.

About Searching for Multiple Values to Edit


You can use the search tool to find and select items for multiple value editing. After you specify rules to find items, you can select items, and then use the **Edit Value** command in the shortcut menu to edit multiple values. The following items are considered valid for multiple editing:

- Dimension Values
- Surface Finish Values
- Geometric Tolerances
- Parameters


Multiple values must meet the following conditions to be edited:

- Same type values must exist (linear or angular)
- Values must be either driven or non-driven
- Read only and regular files cannot be integrated.

To Search and Edit Multiple Values

1. Click **Edit > Find**. The **Search Tool** dialog box opens.
2. Specify the criteria and rules to find items with values to edit.
3. Click **Find Now**. The found items and corresponding locations are listed in the **items found** section of the **Search Tool** dialog box. The results are also highlighted in the Model Tree and graphics window.
4. Click  to transfer items from **items found** to the **items selected** section of the **Search Tool** dialog box.
5. Click **Close**. The **Search Tool** dialog box closes.
6. Right-click in the graphics window. The shortcut menu appears.
7. Click **Edit Value**. The text box appears.
8. Type the new value.

To Search for Intent Objects

1. Click  or click **Edit > Find**. The **Search Tool:1** dialog box opens.
2. Click the **Attributes** tab if it is not selected by default.
3. To search for intent objects, use one of the following combinations for **Look For** and **Look By**. The items that satisfy the specified rule and criteria are listed under **items found** in the **Search Tool:1** dialog box only after you click **Find Now**.

Look For	Look By	Items found
Intent Datum Plane	Intent Datum Plane	Lists all intent datum planes in the model
Intent Datum Axis	Intent Datum Axis	Lists all intent datum axes in the model
Intent Datum point	Intent Datum point	Lists all intent datum Points in the model
Intent Datum Csys	Intent Datum Csys	Lists all intent datum csys in the model
Intent Chain	Intent Chain	Lists all the intent chains in the model
Intent Chain	Feature	Lists all the intent chains that belong to a feature or features
Intent Chain	Edge	Lists all intent chains, that contain all the edges
Intent Surface	Intent Surface	Lists all the intent surfaces in the model
Intent Surface	Feature	Lists all the intent surfaces that belong to a feature or features


4. Under **Rule**, set a rule to search for the intent objects or build a query.

Note: You can use the Name rule to search for the intent objects, that is, search by intent name. The intent name is unique and is used to define the intent object and how Pro/ENGINEER features reference the intent object. The intent name is not the same as the name of the datum reference feature that defines it.


5. Under **Criteria**, specify the criteria to search for the required intent object.

6. If required, click **Options > Build Query** to build a query to search for intent objects and perform the following steps under **Query Builder**:

- a. Specify the rule and criteria for a query.
- b. Click **Add New**, the query appears under **Query Builder**.
- c. If required, specify the rule and criteria for another query and click **Add New** to build another query. You can build multiple such rules for a single search.

7. Click **Find Now**. The items and their corresponding references are listed under **items found** in the **Search Tool:1** dialog box. The results are also highlighted on the Model Tree and in the graphics window.
8. If required, click  to transfer items from **items found** to **items selected**.
9. Click **Close**.


To Search for Objects Referenced by an Intent Object

1. Click  or click **Edit > Find**. The **Search Tool:1** dialog box opens.
2. Click the **Attributes** tab if it is not selected by default.
3. To search for the objects referenced by an intent object, use one of the following combinations for **Look For** and **Look By**. The items that satisfy the specified rule and criteria are listed under **items found** in the **Search Tool:1** dialog box only after you click **Find Now**.

Look For	Look By	Items found
Datum	Intent Datum	Lists all the datums referenced by the intent datums in the model
Datum plane	Intent Datum Plane	Lists all the planes referenced by the intent datum Planes in the model
Axis	Intent Datum Axis	Lists all the axes referenced by the intent datum axes satisfying the specified rule and criteria.
Point	Intent Datum Point	Lists all the points referenced by the intent datum points in the model
Coordinate System	Intent Datum Csys	Lists all the coordinate systems referenced by the intent datum csys' in the model
Edge	Intent Chain	Lists all the edges in the model referenced by intent chains
Surface	Intent Surface	Lists all the surfaces in the model referenced by intent surfaces
Quilt	Intent Surface	Lists all the quilts in the model referenced by intent surfaces
Quilt	Intent Chain	Lists all the quilts in the model referenced by intent chains

4. Under **Rule**, set a rule to search for objects referenced by the intent objects or build a query.

Note: You can use the Name rule to search for the intent objects, that is, search by intent name. The intent name is unique and is used to define the intent object and how Pro/ENGINEER features reference the intent object. The intent name is not the same as the name of the datum reference feature that defines it.

5. Under **Criteria**, specify the criteria to search for the required intent object.
6. Click **Find Now**. The items are listed under **items found** in the **Search Tool:1** dialog box. The results are also highlighted in the graphics window.
7. If required, click  to transfer items from **items found** to **items selected**.
8. Click **Close**.


Annotation Search

To Search for Annotation Feature by Classification

You can search for annotation features by their classification which is a feature level enumerated parameter. Create the first and second rule to search for annotations by classification.

1. Click **Edit > Find**. The **Search Tool** dialog box opens.
2. Click **Attribute**.
3. Select **Feature** from the **Look for** list.
4. In the **Rule** section, click **Type**.
5. Select **Annotation Feature** from the **Value** list.
6. Select the criteria for the annotation element type if necessary.
7. Click **Options > Build Query**. The **Query Builder** dialog box opens.
8. Click **Add New**. The query string is added to the **Rule Description** column.


To create the second rule:

1. In the **Rule** section, click **Expression**.
2. Type the parameter name in the **Symbol** box.
3. Type the desired classification parameter value in the **Value** box.
4. Click **Add New**. The second rule is added to the **Rule Description** column.
5. Click **Operator** and select `and` from the operator list.
6. Click **Find Now**. The found items and corresponding locations are listed in the **items found** section of the **Search Tool** dialog box. The results are also highlighted in the Model Tree and graphics window.
7. To select items, click  to transfer items from **items found** to the **items selected** section of the **Search Tool** dialog box.

8. Click **Close**.

Note: If only the second rule is used, then the features found are not limited to annotation features.

To Search for Annotation Elements with Missing References

1. Click **Edit > Find**. The **Search Tool** dialog box opens.
2. Click **Status**.
3. Select **Annotation Element** in the **Look for** list.
4. Select **Annotation Element** in the **Look by** list.
5. In the **Rule** section, click **Regeneration**.
6. Select **Missing References** in the **Value** list.
7. Click **Find Now**. The found items and corresponding locations are listed in the **items found** section of the **Search Tool** dialog box. The results are also highlighted on the Model Tree and in the graphics window.
8. To select items, click  to move items from the **items found** section to the **items selected** section of the **Search Tool** dialog box.
9. Click **Close**.


Data Sharing Features

To Search for Data Sharing Features with Annotation Element

1. Click **Edit > Find**. The **Search Tool** dialog box opens.
2. Click **Attribute**.
3. Select **Feature** from the **Look for** list.
4. In the **Rule** section, click **Type**.
5. Select the type of data sharing feature to search for from the **Value** list.
6. Click **Options > Build Query**. The **Query Builder** dialog box opens.
7. Click **Add New**. The query string is added to the **Rule Description** column.

To create the second rule:


1. Click **History**.
2. Select **Feature** from the **Look for** list.
3. Select **Annotation Elements** from the **Look by** list.
4. Select **All**.
5. Click **Add New**. The query string is added to the **Rule Description** column.

6. Click **Operator** and select `and` from the operator list.
7. Click **Find Now**. The found items and corresponding locations are listed in the **items found** section of the **Search Tool** dialog box. The results are also highlighted in the Model Tree and graphics window.
8. To select items, click  to transfer items from **items found** to the **items selected** section of the **Search Tool** dialog box.
9. Click **Close**.

Regenerating the Model

About Regenerating Models

When Pro/ENGINEER regenerates a model, it recreates the model feature by feature, in the order in which each feature was created, and according to the hierarchy of the parent-child relationship between features. In an assembly, component features are regenerated in the order in which they were created, and then in the order in which each component was added to the assembly. Pro/ENGINEER regenerates a model automatically in many cases, including when you open, save, or close a part or assembly or one of its instances and when you open an instance from within the Family Table. You can also use the Regenerate command to manually regenerate the model.

The Regenerate command, located on the **Edit** menu or , allows you recalculate the model geometry, incorporating any changes made since the last time the model was saved. If no changes have been made, the system informs you that the model has not changed since the last regeneration. In general, it is a good idea to regenerate the model every time you make a change, so that you can see the effects of each change in the Graphics window as you build the model. Regenerating often helps you stay on course with your original design intent by helping you to resolve failures as they happen.

You can use Regenerate to find bad geometry, broken parent-child relationships, or any other problem with a part feature or assembly component.

When Pro/ENGINEER regenerates a model, the Regeneration status bar appears and indicates the status of the regeneration process. The status or regeneration result of a model is indicated by the following colors:

- Green—Model is up to date - no regeneration needed
- Yellow—Model has changed - regeneration may be needed
- Red—Regeneration has failed or has warnings.

About the Regeneration Footer

The Regeneration Footer is a section of the Model Tree that lists certain types of features. You can place certain features in the Regeneration Folder that are always created and evaluated at the end of the feature list. In addition, some features are

automatically placed in the Regeneration Footer. Placing features in the Regeneration Footer helps you to better understand that certain features are always regenerated at the end of the list

The option of placing certain features in the footer allows you to see different behavior for the same feature, depending on where it is placed. There are two classes of features, Active and Declaration. Only Declaration class features can exist in the Regeneration Footer:

- **Active**—Features that modify the model geometry or new geometry. These features are rolled back during part modification.
- **Declaration**—The features that do not affect the model topology and potentially are not rolled back. These features reference the final geometry and are accessible during any feature operation in the model.

You can place the following types of Declaration features in the Regeneration Folder:

- Zones
- Publish Geometry
- Annotation Features
- Datum Reference Features
- Interfaces

To Move Features to the Regeneration Footer

You can move the features that are created by default in the active list (in the Model Tree above the Insert Here node) to the Regeneration Footer. These features include the Analysis features, Publish Geometry features, Reference feature, Annotation features.

Note: An Interface Feature is created in the footer by default.

1. Right-click the feature that you want to move. The shortcut menu appears.
2. Click **Move to footer**.

You can move the feature types mentioned above from the active list to the Regeneration Footer or from the Regeneration Footer feature list back to the active list.

1. Right-click the feature that you want to move. The shortcut menu appears.
2. Click **Move from footer**.

Note: By default, zones are created in the Regeneration Footer feature list. But, for legacy parts that were created in versions before Pro/ENGINEER Wildfire 2.0, you must use **Move to footer** to move the zones to the Regeneration Footer feature list.

You cannot move zones from the Regeneration Footer feature list to the active list.

About Resolving Failures in Regeneration


There are several reasons why model regeneration can fail, including:

- Bad geometry in one or more features.
- A component or feature is missing from the model; that is, Pro/ENGINEER cannot find it in the working directory. For assemblies, component part files may have been stored in a different directory than that in which the assembly file resides.

When model regeneration fails, all subsequent features (features to be regenerated after the failed feature) remain unregenerated. The current model shows only the regenerated features in the Graphics window (the failed feature and all subsequent features are not displayed).

Pro/ENGINEER enters Resolve mode. The Failure Diagnostics window opens and provides information about which features have failed and why. The window provides helpful links to additional information about the regeneration failure (Overview and Feature Info), including general information about resolving failures.

To Regenerate a Model




1. Click  or **Edit > Regenerate**.
2. Pro/ENGINEER rebuilds your change in the part or assembly model.
3. If a feature fails regeneration, Pro/ENGINEER enters the **Resolve** environment.

To View Model Creation History

1. Click **Tools > Model Player**. The **Model Player** dialog box opens.
2. Select one of the following options for controlling feature regeneration and display:

Regenerate features

Display each feature

3. Using the control buttons or the slider, select the point in the model creation history (that is, select the feature) at which you want to begin the regeneration process. To select a starting feature from the Graphics window or the Model Tree, click . You can also choose **Feat #** and type a feature number to specify a starting feature. Use  and  to choose a starting feature by stepping backward or forward through the playback operation one feature at a time.

The feature you specify becomes the starting feature. The system suppresses all features that occur after it.

4. To stop playback, click **Stop**.
5. Click a command at the bottom of the Model Player to get information about the current feature (the feature that was currently regenerated when you stopped



the model playback process). For a description of each command, click *See Also* below.



If a feature is one that intersects any other feature on the part, the system displays the geometry of the feature in red wireframe before intersection.


Using the Model Player


To open the **Model Player** dialog box, click **Tools > Model Player**.

- You can select one of the following commands for controlling feature regeneration and display:
 - **Regenerate features**—Regenerates each feature in sequence, starting from the specified feature, as the model moves forward. If this box is cleared, features will be rolled forward without regenerating. If you have made changes to the model without regenerating, the command is checked and unavailable.
 - **Display each feature**—Displays each feature in the Graphics window as it is being regenerated or rolled forward. Future features are not displayed until they are regenerated. If this box is cleared, the entire display is updated only when the desired feature is reached and the model playback is complete, or when you stop model playback.
 - **Compute CL**—(Available in Manufacturing mode only) When selected, the CL data is recalculated for each NC sequence during regeneration. If cleared, no CL data is calculated during regeneration.
- Select one of the following commands to select the place (feature) in the model creation history at which to start the regeneration process:


—Moves immediately to the beginning of the model (suppresses all features). Use  to step through the features one at a time.



—Moves immediately to the end of the model (resume all features). Use  to step through the features one at a time.

—Steps backward through the model one feature at a time and regenerates the preceding feature in the model.

—Steps forward through the model one feature at a time and regenerates the next feature in the model.

Slider Bar—Drags the slider handle to the first feature at which you want model playback to begin. The features are highlighted in the Graphics window as you move through their position with the slider handle. The feature number and type are displayed in the selection panel (such as #16 (CHAMFER)), and the feature number is displayed in the **Feat #** box. When you release the slider, the model immediately rolls or regenerates to that feature.

—Lets you select a starting feature from the graphics window or the Model Tree. Opens the **SELECT** dialog box and **SELECT FEAT** menu. After you select a starting feature, its number and ID are displayed in the selection panel, and the feature number is displayed in the **Feat #** box. When you select a feature, the model immediately rolls or regenerates to that feature.

Feat #—Lets you specify a starting feature by typing the feature number in the box. After you enter the feature number, the model immediately rolls or regenerates to that feature. Use  and  to step backward or forward through the playback one feature at a time.

The feature you specify becomes the starting feature, and all features that occur after it are suppressed.

- To stop playback, click the **Stop** button.
- You can use the following commands to obtain information about the current feature (the feature that was currently regenerated when you stopped the model playback process):
 - **Show Dims**—Displays the dimensions of the current feature.
 - **Feat Info**—Provides regular feature information about the current feature in an Information window.
 - **Geom Check**—Investigates the geometry error for the current feature. This command is accessible only when the system encounters a geometry error.
 - **Fix Model**—Activates Resolve mode by forcing the current feature to abort regeneration. When you exit Resolve mode, Pro/ENGINEER returns to the Model Player at the current feature. You can then continue to move backward or forward through the model.
 - **Close**—Closes the Model Player and enters Insert mode at the current feature (the last regenerated feature). You can choose this command anytime during the model playback.
 - **Finish**—Closes the Model Player and returns to the last feature in the model. Pro/ENGINEER restores all features. You can choose this command anytime during model playback.

If a feature is one that intersects any other feature on the part, the system displays the geometry of the feature in red wireframe before intersection.

Integrating Changes using Pro/INTRALINK

About Integration

Integration is a process that lets you merge conflicting changes made by multiple users to the same revision of an object fetched from a Pro/INTRALINK product database.

Integration facilitates concurrent engineering by allowing multiple engineers to modify an object simultaneously without fear of overwriting each other's changes or making irreconcilable modifications. Preparation for change integration requires steps to be performed in Pro/INTRALINK and Pro/ENGINEER.

Causes of Revision Conflict

It may occur that multiple users fetch an object from a Pro/INTRALINK product database and make different changes to the object independently of each other. When the modified objects are resubmitted to the product database, Pro/INTRALINK checks each submitted object for conflicts with objects already present in the product database.

For example, suppose that you and another user fetch the latest revision of a part from a product database. You both modify the object, but the other user resubmits the object to Pro/INTRALINK and receives approval for the changes before you do. The other user's modified part has become the latest revision in the product database.

When you submit your modified part, you are informed that your changes are not based on the latest revision of the part, and that the revision you are attempting to create already exists in the product database. To resolve this conflict, you must merge the changes in your version, the source, with the latest version in the database, the target by using **File > Integrate**.

Similarly, conflicts can arise when you submit an object, such as a drawing or assembly, which depends upon an object which itself has a conflict. If a drawing uses a changed part, requiring you to switch references for the drawing, then the drawing also must be included in the list of objects to be integrated.

Based on information that you have already provided, Pro/ENGINEER determines which references in the drawing may be switched automatically, then prompts you to map any remaining references manually.

To Use the INTEGRATE Menu

1. After you have retrieved the model to be integrated, click **File > Integrate**. The difference report, source object, and target object are displayed.
2. The MERGE_VIEW and the DIFF_VIEW Windows appear. The DIFF_VIEW Window displays the difference report for the object revisions. The **INTEGRATE** menu appears, listing commands for performing the integration:
 - **Next**—Step to the next item (difference) in the report.
 - **Previous**—Step to the previous item (difference) in the report.
 - **Show**—In the active window, highlight the entity (such as a feature or component) 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**—Bring up the Information window, containing information about the current item in the difference report such as the model name, component number, internal ID number, part name, and parents and/or children.
- **Action**—Bring up the **ACTION** menu. When you specify an action for one occurrence of an item (such as a feature or a component) in the report, the specified action is set automatically for that item every time it appears in the report.
- **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, children must be deleted before parents).
- **Diff View**—(Default) Display the difference report sorted by object and, within object, by item type. This gives a quick indication of where and what the differences are.
- **Done/Return**—Dismiss the DIFF_VIEW Window and quit the **INTEGRATE** menu.

Accessing the Integrate Mode to Resolve Conflicts

To resolve revision conflicts, each mode in Pro/ENGINEER includes the **Integrate** command.

The **Integrate** command accesses **Integrate** mode for retrieving integration project files (created in Pro/INTRALINK) generating difference reports.

The Difference Report

A difference report, displayed in the DIFF_VIEW Window by default, lists the differences between the source object and the target object, including all of their modified dependent objects, and serves as the basis of the integration session.

The MERGE_VIEW Window may be brought up, listing the differences between the source object and the target object in the order in which they must be resolved. Changes in a simplified representation of an object are shown in a difference report.

A difference report shows differences for the object in the active window and its dependents, if any. For example, if an assembly is the active object, differences in the assembly and in its components display in the difference report.

Using a difference report, you can specify actions that you want Pro/ENGINEER to perform automatically on the target object to resolve the differences with the source object. This includes merging changes in the source object into the target object,

deleting from the target those items that exist only in itself, or intentionally ignoring an item. (In other words, take no merge or delete action on it even though a difference is noted.)

A difference report shows the differences for all of the objects included in an integration project. You may specify actions only for an object that is modifiable in the current mode. For instance, an integration project file for an assembly includes the differences of all of its components.

If the active object is a part that is a member of the assembly, the differences for the parent assembly still display in the difference report, but in Part mode, you may specify actions only for the part. To specify actions for the parent assembly, you must retrieve it into Assembly mode.

Sample Difference Report Before Specifying Actions shows an example of a difference report before commands in the **ACTION** menu are used to specify how a difference must be resolved.

Example: A Sample Difference Report before Specifying Actions

Model Comparison :

BASE MODEL: FOR_ANALYSIS.PRT.2
 BASE VERSION:
 COMPARISON MODEL: SURFACE_FOR_SHADOW.PRT.1
 COMPARISON VERSION:

Change List

Item Type	Item Name	Item Id	Change Type	Change Description
SURF	RIGHT	1	Metadata	DIFFERENT NAMES/MODELS
PARAMETER	PARAMETER_1		Metadata	DESIGNATED ONLY IN BASE MODEL
PARAMETERS	PARAMETER_1		Metadata	FOUND ONLY IN BASE MODEL
PARAMETERS	MODELED_BY		Metadata	FOUND ONLY IN BASE MODEL
PARAMETERS	DESCRIPTION		Metadata	FOUND ONLY IN BASE MODEL
DTM_PL	TOP	3	Geometry	FOUND ONLY IN BASE MODEL
DTM_PL	FRONT	5	Geometry	FOUND ONLY IN BASE MODEL
CRDSYS	PRT_CSYS_DEF	7	Geometry	FOUND ONLY IN BASE MODEL
CRDSYS	CS0_2	197	Geometry	FOUND ONLY IN BASE MODEL
DTM_AX	A_2	203	Geometry	FOUND ONLY IN BASE MODEL
DTM_PL	DTM1	164	Geometry	FOUND ONLY IN BASE MODEL
Style		39	Geometry	FOUND ONLY IN BASE MODEL
Style		51	Geometry	FOUND ONLY IN BASE MODEL
PRTRSN		75	Geometry	FOUND ONLY IN BASE MODEL
ROUND		93	Geometry	FOUND ONLY IN BASE MODEL
CURVE		134	Geometry	FOUND ONLY IN BASE MODEL
DTM_PT	PNT0	171	Geometry	FOUND ONLY IN BASE MODEL
DTM_PT	PNT1	173	Geometry	FOUND ONLY IN BASE MODEL
ANNOTFT	Annotation 1	228	Geometry	FOUND ONLY IN BASE MODEL
RELATIONS			Geometry	DIFFERENT
NOTE		0	Geometry	NEW IN BASE MODEL
LAYER	06__PRT_ALL_SURFS		Cosmetic	IS ABSENT IN COMPARISON MODEL
LAYER	05__PRT_DEF_DTM_CSYS		Cosmetic	IS ABSENT IN COMPARISON MODEL
LAYER	05__PRT_ALL_DTM_CSYS		Cosmetic	IS ABSENT IN COMPARISON MODEL
LAYER	04__PRT_ALL_DTM_PNT		Cosmetic	IS ABSENT IN COMPARISON MODEL
COLORS			Cosmetic	DIFFERENT
VIEW	DEFAULT		Cosmetic	DIFFERENT
VIEW	LEFT		Cosmetic	NEW IN COMPARISON MODEL

If you set the `info_output_format` configuration option to `html`, the difference between features of the selected parts is listed in a report in the Pro/ENGINEER

browser window. Alternatively, you can also view the model information in an INFORMATION WINDOW, if the `info_output_format` configuration option is set to `text`. All the differences between the objects is itemized, listing a type of change (feature, layer, relation, and so forth), source and target name, item ID, description, and space for specifying an action.

Pro/ENGINEER provides a standard set of descriptions. Note that not all descriptions apply to all change categories:

- New
- Deleted
- Different
- Suppressed
- Redefined (Attributes, Section, Scheme, and so on)
- Moved

Each description is appended, if applicable, with one of the following notations, indicating where the change is located:

- In Target
- In Source

The column with the ID indicates the location of the change. If the ID appears in both columns, then the modified item exists in both objects.

Note: Differences in patterns are reported only for the pattern leaders.

Comparing Family Tables in Assemblies

When two assemblies are compared so that they can be integrated, their Family Tables are compared only if the tables contain the same number of items. In addition, their Family Table instances are not compared if any of the following conditions exist:

- Differences are found in the table-driven items (columns) of the Family Tables.
- The numbers of instances in the Family Tables differ.
- The names of the instances in the Family Tables differ.

To Prepare the Objects for Integration

In Pro/INTRALINK, select the objects that you want to integrate changes for and click **Object > Integrate > Start** in the Workspace browser.

You are now ready to enter a Pro/ENGINEER session and merge the changes.

To Integrate Changes in Pro/ENGINEER

1. Start a Pro/ENGINEER session and retrieve the target object to be integrated.

Note: Integration must be started in Pro/INTRALINK Workspace prior to beginning the integration process in Pro/ENGINEER. The target object is checked out from the Commonsense browser and appears in the Workspace browser. The source object is then no longer listed in the Workspace browser.

2. Click **File > Integrate** on the main menu for the mode.
 3. The MERGE_VIEW Window appears, displaying the corresponding target or source object (whichever is not yet active in the session), and the DIFF_VIEW Window appears, displaying the difference report for the object revisions. The **INTEGRATE** menu and the **ACTION** menu appear.
4. Specify an action to be taken (**Take From Source, To Delete, Do Manual**, and so on) for items in the difference report. For example, if you choose to merge different dimensions for an item, the source dimension value is used in the updated target object. If you specify an action for a child item, Pro/ENGINEER warns you that parent items are marked with the same action; you can then confirm or cancel the assignment.
5. If a feature needs to be integrated manually from the source object to the target object, exit Integrate mode, then use commands in the **FEATURE** menu (such as **Delete** or **Copy**) to do so. If you use **Copy**, specify **FromDifVers** and **Independent** to indicate how and from where the feature is being copied.
6. For items that you have integrated manually, use the **Merged** and **Deleted** commands in the **ACTION** menu to indicate the updated status of each item.
7. For items which you marked to be integrated automatically (using **Take From Source, To Delete**, and so on), click the **Execute** command from the **INTEGRATE** menu.
8. Upon choosing **Execute**, all items are updated according to the specified action (for example, if you selected **To Delete**, the corresponding items are marked **DELETED**). If you choose to delete a feature that is new in the target object, this feature is removed in the updated target when **Execute** is chosen. If a merge is not possible because of missing external references, or if the merge regeneration fails, the standard regeneration failure interface appears.

If the **RegenBackup** option in the **ENVIRONMENT** menu was set, then the system first backs up all the targets that would be affected by the **Execute** command before actually processing the command. After the execution has completed successfully and provided that you have not already restored all the affected objects from within regeneration, you can then confirm or cancel the changes. If you cancel them, all objects that have not been restored are restored from their backups. All items that are undone also become unmarked, so that you can execute them again.

9. Click **Done/Return** from the **INTEGRATE** menu when the integration session is finished. Updated versions of target objects are saved automatically. You may continue to integrate changes at a later time repeating the process explained above.

After the integration process is complete and the modified target object is saved, you must enter a Pro/INTRALINK session to submit the modified target object back to the product database.

Resolution in Pro/INTRALINK

Submit the modified target object back to the product database for approval. Upon approval, it becomes the latest revision of the object in the Pro/INTRALINK product database.

About Switching References in a Drawing

If a source object being integrated is displayed in a drawing, any drawing references to merged items have to be switched to corresponding references in the target model.

For example, if a hole was added to the source object and the diameter of the hole was displayed on a drawing, the dimension on the drawing has to be switched to reference the hole which has been merged into the target object.

Many references can be switched automatically when items are merged between the source and target parts or assemblies. In some cases, however, references must be switched manually.

For example, in the case of a hole added to the source object, after the hole is merged into the target object, the system must be able to automatically switch the diameter dimension shown on a drawing.

If, however, you redefine the section of a feature in the source object so that it looks completely different than that in the target object, the references might not be readily compatible, and you must switch them manually.

To Switch References in a Drawing Manually

1. Retrieve the drawing that references the source object or objects.
2. Click **File > Integrate**. The **Open** dialog box opens.
3. Select the integration project file (*.ipf).
4. Click **Open**. If the drawing contains more than one model, the **DWG SW MDL** menu appears, enabling you to select a model with which to work:
 - **Pick Model**—Select a model in the current object. If that model needs to be switched, the target for that model is retrieved in a sub-window, and the **DW MDL MAP** menu appears.
 - **Next Model**—You can step through models in the current object one by one. Models that do not need to be switched are automatically skipped.
 - **Prev Model**—Same as **Next Model**, but stepping in the opposite direction.

- **Done**—Finish switching references. Pro/ENGINEER warns you if any references are left unmapped. If you disregard the warning, any items that use the unmapped references are lost.
- **Quit**—Abort the process of mapping references.

If the drawing contains only one model that needs to be switched, that model is selected automatically. In either case, the selected target model is retrieved into a sub-window.

5. The **DW MDL MAP** menu enables you to switch references from the source model to the target model. The **DW MDL MAP** commands are:
 - **Next Ref**—Step to the next reference that needs to be mapped.
 - **Prev Ref**—Return to the previous reference.
 - **Pick Ref**—Pick a reference to map.
 - **Unmap Ref**—Undo the mapping for a particular reference.
 - **Show Und**—Display unmapped references.
 - **Show Done**—Display references which have been mapped.
 - **Done**—Finish mapping references. Pro/ENGINEER warns you if any references are left unmapped. If you disregard the warning, any items that use the unmapped references are lost.
 - **Quit**—Abort the process of switching references.
6. Click **Next Ref**. A referenced entity (for example, surface, edge, and dimension) highlights on the drawing. Select the corresponding entity on the target model in the sub-window.
7. Continue this process until all references have been switched.

Reference mappings can be changed by toggling to the correct reference (using **Next Ref** and **Prev Ref**), choosing **Unmap Ref** to undo the mapping, and then re-selecting the target entity.

Example: Integration Process Conflict

The procedure that follows is based on revision conflicts produced by two users. The integration process can merge changes between only two versions at a time; this process may be repeated to integrate the conflicts of multiple users.

The Revision Conflict

Imagine that you and another user have retrieved the same revision of a part from a Pro/INTRALINK product database and made different modifications to it. The other user has already submitted the object back to Pro/INTRALINK and received approval for the changes that he or she made, and the modified object is now the latest revision in the product database.

When you attempt to submit your modified version of the object, Pro/INTRALINK informs you that this revision of the object already exists in the product database. Pro/INTRALINK provides you with two solutions.

If your design replaces the previously submitted version, you may continue to submit using the Pro/INTRALINK command **Override Errors**. However, if you want to incorporate the changes made in the latest revision in the database, Pro/INTRALINK provides you with tools to integrate these models.

Source and Target Objects

When you decide to integrate the changes between two revisions of the same object, you must use those revisions to prepare two versions of the object, a source object and a target object.

The source object is the revision of the object that contains unapproved changes that you want to have incorporated into the target object, which is the latest revision of the object in the product database.

Pro/INTRALINK provides you with a process for creating a renamed version of the source object and fetching a copy of the target object, so that you can integrate the changes and create a new latest revision of the object without overwriting the original source and target objects.

Annotation Features

About Annotation Features

Annotation features are data features that you can use to manage the model annotation and propagate model information to other models, or to manufacturing processes.

An Annotation feature consists of one or more Annotation Elements. Each Annotation Element can contain one annotation item, along with associated references and parameters. You can include the following types of annotations in an Annotation Element:

- Note
- Symbol
- Surface finish
- Geometric tolerance
- Set Datum Tag
- Ordinate baseline dimension
- Driven dimension
- Ordinate driven dimension
- Chamfer dimension

- Reference dimension
- Ordinate reference dimension
- Existing annotation

You can do the following:

- Include Annotation features in User Defined Features (UDFs). This way, you can define a library of company-specific annotations, and then place it in various models, as appropriate.
- Include the necessary manufacturing information in the model annotation, and use this information when defining the manufacturing process.
- Search for annotation options and values, or for Annotation Element parameters, and based on the search results, simultaneously modify multiple instances.
- Propagate Annotation Elements from Annotation features to Data Sharing features, with options for dependency.
- Duplicate annotations by placing User Defined Features, creating reference patterns, creating patterns of groups that have annotation features, or copying Annotation features. The duplicated annotations do not overlap except for free placed annotations. In the duplicated annotations, the relationship between the annotation leader attachment point and the annotation placement point is maintained and is the same as that of the original annotation. Similarly, in case of angle and length dimensions, the relationship between the annotation attachment point, that is, the reference point and the dimension text placement point is maintained and is the same as that of the original annotation.

Note: In copy-and-paste operations where alternative attachment references are specified, the annotations do not overlap.

Annotation Feature Basics

About Annotation Feature Structure

An Annotation feature consists of a name and one or more Annotation Elements. The system provides a default name for each Annotation feature as you create it, such as `Annotation 1`, `Annotation 2`, and so on. You can specify a different name.

An Annotation Element is composed of:

- Name—The system provides a default name for each Annotation Element as you create it. The default name includes the annotation type and a sequentially incremented number, such as `AE_NOTE0`. You can specify a different name.
- Annotation—An Annotation Element may include not more than one annotation, such as note, symbol, and so on. The type of annotation determines the Annotation Element type. If no annotation is included, then the Annotation Element is Non-graphical.

- **References**—If an annotation is included in the Annotation Element, its geometric references become the references of the Annotation Element. You can also specify additional geometric references.
- **Parameters**—You can add parameters to an Annotation Element. All the regular types of parameters, such as String, Integer, and so on, are available. You can also specify that certain parameters are automatically added to each Annotation Element at the time of its creation.

About Annotation Types

The following types of annotation can be included in Annotation features:

- Note
- Symbol
- Surface finish
- Geometric tolerance
- Set Datum Tag
- Ordinate baseline dimension
- Driven dimension
- Ordinate driven dimension
- Chamfer dimension
- Reference dimension
- Ordinate reference dimension
- Existing annotation

The type of annotation defines the type of the Annotation Element. You can also create Non-Graphical Annotation Elements, which do not include annotation of any type.

Another item that can be included in an Annotation Element is a Manufacturing template. For more information on defining Manufacturing templates and using them in Annotation features, see the Process Manager documentation in NC Manufacturing.

About Annotation Plane

Annotations that can be included in Annotation features have to reference an Annotation plane. The Annotation plane is a plane that defines the orientation of the annotation item with respect to the 3D model. The annotation item is not actually placed on the Annotation plane, but on an imaginary plane parallel to the Annotation plane, and facing in the direction of the front of the Annotation plane. The actual location of this imaginary plane with respect to the model is determined by the first selected attachment for the annotation.

When you reorient the model, the annotation is also reoriented accordingly. As you look at the model from different views, various annotations will be viewable (and some will not).

About Annotation Orientation

An annotation orientation refers to the plane or the parallel plane in which the annotation lies, the viewing direction, and the right direction or text rotation.

An active annotation orientation refers to the annotation orientation that will be used for creating the next annotation or annotation element. A current annotation orientation refers to the annotation orientation of an annotation or Annotation Element that is placed.

Using the **ACTIVE ANNOTATION ORIENTATION** or the **CURRENT ANNOTATION ORIENTATION** dialog boxes, you can define an annotation orientation using a datum plane, flat surface, or named view. You can also define annotation orientations such that they are placed flat to screen and also switch between these different types of annotation orientations.

The **ACTIVE ANNOTATION ORIENTATION** dialog box opens when you click **View > Annotation Orientation** while the **CURRENT ANNOTATION ORIENTATION** dialog box opens when you right-click an annotation and click **Orientation** on the shortcut menu.

Note the following points about annotation orientations:

- Define the active annotation orientation, if required, before creating the first annotation in any session. This annotation orientation is retained per session per model. Thus, when you define the active annotation orientation and create a new annotation or Annotation Element, Pro/ENGINEER retains that annotation orientation and uses it to create the next annotation. All subsequently created annotations retain the same orientation unless the Annotation plane information is changed. This holds true for new Annotation Elements created in new Annotation Features as well.
- The default annotation orientation is a frozen orientation in the Z-direction of the default coordinate system. This default annotation orientation corresponds to the FRONT datum plane or FRONT orientation in the PTC template for parts. However, this default orientation does not reference the FRONT datum or FRONT view. In the **ACTIVE ANNOTATION ORIENTATION** dialog box, the default orientation is defined by the view. However, view name is not specified.
- When redefining an existing annotation, if you change its annotation orientation, the changed definition of the annotation orientation is not retained for the next created annotation.
- If you delete or suppress a datum plane or named orientation that defines the annotation plane in an annotation, the next annotation orientation is frozen.
- If an annotation cannot be placed on the current annotation orientation, then Pro/ENGINEER displays a message stating that the annotation plane is invalid and that a new annotation plane must be defined.

- If the last created Annotation Element created a datum plane for its annotation plane, thus creating an embedded datum, then the next created Annotation Element in the same Annotation Feature uses the same orientation as the Annotation Element with the embedded datum plane. If the next created Annotation Element is in another Annotation feature, then the Annotation Element does not directly reference the embedded datum plane.
- When you open the **ACTIVE ANNOTATION ORIENTATION** or **CURRENT ANNOTATION ORIENTATION** dialog box, the current or active annotation plane is visible as a grid plane. The active annotation grid plane is displayed in green, that is, the Highlight-Edge system color while the annotation orientation of an existing annotation is displayed in orange, that is, Secondary Selected system color. Both grid planes show viewing direction and text direction.

Annotation Orientation by Plane

Annotation Orientation by Named Orientation

Annotation Orientation by View Screen

Annotation Plane

About Annotation Orientation by Plane

You can define the orientation for annotations by selecting a datum plane or flat surface which is parallel to the annotation. The annotation orientation defined by a reference plane can be driven by plane or frozen.

Driven By Plane

When an annotation orientation is driven by plane, the datum plane or flat surface is displayed as a reference of the Annotation Element in the **References** collector in the **ANNOTATION FEATURE** dialog box. If you delete the reference, Pro/ENGINEER replaces the reference with an alternative reference. If an alternative reference is not found, then the orientation becomes frozen. Similarly, in a model annotation if the reference is deleted, the annotation is not displayed.

If you change the datum plane or flat surface orientation, the annotation orientation changes accordingly.

Frozen

When an annotation orientation is frozen, the annotation orientation is stored in the annotation and the annotation plane reference is not required. Therefore, even if you delete the plane reference, the annotation is not affected.

Pro/ENGINEER does not display references in the **References** collector in the **ANNOTATION FEATURE** dialog box for an Annotation Element with a frozen annotation orientation.

You can switch the annotation orientation between the driven-by-plane state and frozen state by clicking or clearing the **Freeze Annotation Plane reference** checkbox. However, you can revert the annotation orientation of an annotation to the driven-by-plane state only if the annotation plane still exists.

It is not necessary to place or copy datum plane of an annotation with frozen annotation orientation to UDF or DSF. You can change the annotation orientation of the target Annotation Element independent of the source Annotation Element.

To Define Annotation Orientation by Plane

1. Click **View > Annotation Orientation**. The **ACTIVE ANNOTATION ORIENTATION** dialog box opens.
2. Click **Reference Plane** and select a datum plane or a flat surface to which you want the annotation to be parallel. The active annotation grid plane is displayed in light green and shows the default viewing and text directions. The text direction is the direction in which you read the text.
3. Click **Flip** next to **Viewing Direction** to flip the viewing direction of the annotation.
4. In the **Text Rotation** box, type a value for the angle of text rotation or select from the list of most recently used values. The direction of the arrow in the graphics area is updated accordingly.

Note: The default rotation angle of the annotation plane is 0.

5. If required, click **Freeze Annotation Plane reference** to freeze the annotation plane reference. The annotations that use this annotation do not reference the datum plane or named view.
6. Click **OK**.

About Annotation Orientation by Named Orientation

You can define an annotation plane using a named orientation. If you delete the named orientation, the annotation orientation remains unchanged.

Note the following points about an annotation orientation by named orientation.

- Annotation Elements with a named orientation do not display any reference for the annotation plane in the **References** collector in the **ANNOTATION FEATURE** dialog box.
- Pro/ENGINEER does not copy a named orientation in the UDF or DSF. The Annotation is frozen relative to the UDF geometry.
- All Annotations oriented by named orientation placed with independent UDFs are frozen. You can change the Annotation Element orientation in the target model independent of the source Annotation Element.
- If you create the annotation in a part in an assembly, then you can select only the named views of the part for an annotation plane.

To Define Annotation Orientation by Named Orientation

1. Click **View > Annotation Orientation**. The **ACTIVE ANNOTATION ORIENTATION** dialog box opens.

2. Click **Named orientation** and select a named orientation from the list. The active annotation grid plane is displayed in green and shows the default viewing and text directions. The text direction is the direction in which you read the text.
3. Click **Flip** next to **Viewing Direction** to flip the viewing direction and the text direction of the annotation.
4. In the **Text Rotation** box, type a value for the angle of text rotation or select from the list of most recently used values.
5. Click **OK**.

About Flat-to-screen Annotation Orientation

You can define an annotation plane that remains flat to the screen for an annotation. Pro/ENGINEER places such annotations flat to the screen.

Note the following points about an annotation orientation flat to the screen.

- If you place flat-to-screen annotations in the graphics window and these annotations are not attached to the model, then the annotations do not spin, pan, or zoom with the model. Such annotations remain at a fixed location in the graphics window and have the location point placed in a relative screen position relative to lower left corner of the screen.
- If you place flat-to-screen annotations that are attached to a point on an entity of the model, and specify the text height in standard screen units by clearing the **Text height in model units** checkbox in the **ACTIVE ANNOTATION ORIENTATION** dialog box, then such annotations pan with the model but do not zoom and rotate with the model.
- If you place flat-to-screen annotations that are attached to a point on an entity of the model, and specify the text height in model units by checking the **Text height in model units** checkbox in the **ACTIVE ANNOTATION ORIENTATION** dialog box, then such annotations pan and zoom with the model but cannot rotate.

Model units use the same size definition as the annotations placed in annotation planes. You can switch between model units and standard screen units (mm). The text height of a flat to screen annotation placed in graphics window is defined in standard screen units and the default text height is set to 3.5 mm.

- You cannot attach flat-to-screen annotations to silhouette edges, or on points defined by the intersection of entities or for any other type of attachment which is dependent on a projection in a particular view direction.

Note: Flat to screen annotations are not the same as old style flat to screen notes.

To Define Flat-to-screen Annotation Orientation

1. Click **View > Annotation Orientation**. The **ACTIVE ANNOTATION ORIENTATION** dialog box opens.

2. Click **Flat to screen** to define an annotation plane that remains flat to the screen.
3. Click **Screen Location** to select a point in the graphics window but unattached to the model when defining the placement of the annotation.

Note: Flat-to-screen annotations placed in graphics window but unattached to the model, do not spin, pan, or zoom with the model.

Alternatively, click **Geometry Location** to use a point on the model to define placement of annotation. On clicking **Geometry Location**, the **Text height in model units** checkbox becomes available for selection.

4. Select **Text height in model units** checkbox to specify text height in model units. To specify the text height in screen units clear the checkbox.

Note: Flat-to-screen annotations with height specified in screen units, only spin and zoom with the model. However, flat-to-screen annotations with height specified in model units, only zoom and pan with the model.

5. Click **OK**.

About Reference Types

Annotation Element references include all the geometric references of its annotation item. For example, if an Annotation Element includes a note, and the note is attached to two edges, then these edges and the note's Annotation plane are automatically included in the References list of the Annotation Element.

Optionally, you can explicitly select additional Annotation Element references of the following types:

- Datum points and geometry vertices
- Datum curves, geometry edges, and surface edges
- Solid or quilt surfaces
- Datum axes
- Coordinate systems
- Datum planes
- Intent objects
- Components (only available for Annotation features within an assembly)
- Sub-assemblies (only available for Annotation features within an assembly)
- Entities on layers (combinations of any of the entity types above, but not rules)
- Entities through selection, for entities such as chains of edges or sets of surfaces.

These references can be Weak or Strong. A loss of Strong reference causes the failure of the Annotation feature. A loss of a Weak reference does not cause the failure of the Annotation feature, but it may make the annotation inactive.

By default, all the additional Annotation Element references that you select are Weak. You can change a reference to Strong by selecting the checkbox in the **Strong** column next to the reference name in the appropriate reference collector in the **ANNOTATION FEATURE** dialog box.

You can remove a previously selected reference by right-clicking its name in the collector and selecting **Remove** on the shortcut menu.

You cannot remove the automatically populated references directly from the **References** collector. To change these references, redefine the properties of the annotation item. You can, however, change the status of these references (Strong or Weak) by selecting or clearing the checkbox next to the reference name in the **References** collector.

Strong and Weak References

The Annotation Element references can be Strong or Weak:

- Strong references are checked during regeneration of the Annotation feature. If a Strong reference is lost, the Annotation feature fails regeneration.
- Weak references are checked at the end of the model regeneration. If a Weak reference is lost, the Annotation feature does not fail regeneration. If the missing Weak reference is used by an annotation leader, the leader is lost (not displayed). If the last remaining leader is lost, the annotation becomes inactive. Inactive annotation is not displayed.

For example, if an Annotation Element includes a note, and the note is attached to two edges, then the two edges are Weak references. If one of the edges is not found at the end of regeneration (for example, if it was chamfered somewhere later in the regeneration sequence), the leader that used to be attached to this edge is not displayed. The note has only one leader, attached to the second edge. However, if the second edge is also lost, the note would lose its last remaining leader and become inactive.


When the references are automatically populated from an annotation item, they are designated Strong or Weak according to the following rules:

- Annotation plane is a Strong reference.
- Leader attachments are Weak references.
- The reference entity of a Geometric tolerance (the entity to which the GTOL applies) is a Strong reference.
- In Driven and Reference dimensions, all their references are Strong, because without these references, the dimension cannot exist.

By default, all the additional Annotation Element references that you select are Weak. You can change a reference to Strong by selecting the checkbox in the **Strong** column next to the reference name in the appropriate reference collector in the **ANNOTATION FEATURE** dialog box.

Creating Annotation Features

About Creating Annotation Features


You can create Annotation features at any time by clicking the  icon. When you create Annotation Elements, you can either select existing annotation, or create annotation on the fly. As you select the Annotation Element type, the system invokes the regular user interface for creating this specific annotation type (a note, a symbol, and so on). Once the annotation is successfully created, you can specify parameters or select additional references for the Annotation Element.

To Create an Annotation Feature

This topic describes the general procedure of creating an Annotation feature.

1. Define the active annotation orientation before creating the first annotation in any session using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.

Note: If you skip step 1, then Pro/ENGINEER uses the default active annotation orientation. However, if you have previously created annotations then Pro/ENGINEER uses the same annotation orientation as the previously created annotation.

2. Click  on the Annotation toolbar (by default, located on the right side of the Pro/ENGINEER window). The **ANNOTATION FEATURE** and **Add Annotation** dialog boxes open simultaneously. The **Add Annotation** dialog box opens with **Note** or the last created annotation in the current session selected by default.
3. Select one of the following annotation types:
 - **Note**—Create a note to include in the Annotation Element.
 - **Symbol**—Create a symbol to include in the Annotation Element.
 - **Surface Finish**—Create a surface finish to include in the Annotation Element.
 - **Geometric Tolerance**—Create a geometric tolerance to include in the Annotation Element.
 - **Set Datum Tag**—Create a set datum tag to include in the Annotation Element.
 - **Ord Baseline**—Create an ordinate baseline dimension.
 - **Driven Dimension**—Create a driven dimension with tolerance to include in the Annotation Element.
 - **Ord Driven Dimension**—Create one or more ordinate driven dimensions to include in the Annotation Element.
 - **Chamfer Dimension**—Create a 45 degree chamfer dimension to include in the Annotation Element.

- **Reference Dimension**—Create a reference dimension to include in the Annotation Element.
 - **Ord Reference Dimension**—Create one or more ordinate reference dimensions to include in the Annotation Element.
 - **Manufacturing Template**—Select a previously saved manufacturing template to include in the Annotation Element. For more information, refer to documentation on NC Manufacturing.
 - **Non-Graphical**—The Annotation Element does not contain an annotation.
 - **Existing Annotation**—Select an existing annotation item (such as a note, symbol, and so on) to include in the Annotation Element. You can also select an existing Annotation Element to include a copy of its annotation item in the current Annotation Element.
4. Click **OK** or middle-click. Pro/ENGINEER invokes the user interface for creating or selecting the annotation item. For more information on creating an annotation of a particular type, follow the *See Also* links.

Note:

- If you want to change the active annotation orientation for the annotation you are about to create then click **View > Annotation Orientation** or click **Active Orientation** on the shortcut menu that opens when you right-click in the user interface for creating the annotation item.
- After you place the annotation you can change the current annotation orientation by clicking **Current Orientation** on the shortcut menu that opens when you right-click in the user interface for creating the annotation item.

Once you create or select the annotation item, Pro/ENGINEER creates the Annotation Element, adds its name, type, orientation, and copy flag to the elements list in the **ANNOTATION FEATURE** dialog box, and adds the references, along with the reference type and default reference description, to the **References** collector.

5. Click **Parameters** to specify the Annotation Element parameters.
6. You can click **Edit** to edit the definition of the annotation item included in the selected Annotation Element, or click **Remove** to delete the selected Annotation Element from the Annotation feature.
7. You can click **Columns** to customize the columns in the Annotation Elements list and the **References** list. In the **Annotation Columns** dialog box that opens when you click **Columns**, select the list that you want to customize from the **Type** list and move the names of the columns that you want to view in the list to the **Displayed** box. You can also specify the width of the column.
8. You can change the orientation of the annotation by clicking **Current Orientation** on the shortcut menu that appears when you right-click the annotation in the Annotation Elements list.

Note: The **Current Orientation** command is not available for non-graphical Annotation Elements.

9. To add additional references to the Annotation Element, click **References** and then click **Annotation References**, **Surface Collector**, or **Chain Collector** to activate the respective collector (**References**, **Chains** or **Surfaces**) and select the valid entities for the specified collector.
10. In the **Strong** column, if the check box next to the reference name is not selected, then the reference type is Weak. If you want to change the reference type to Strong, select the check box.

Note: If you have selected datum points as references, then a check box in the **Auto-Propag** column is also available for the datum points in the **References** collector. Select the check box to enable automatic propagation of datum points. If you propagate the Annotation Element to data sharing features, then those datum point references for which you have selected the check box in the **Auto-Propag** column are also automatically propagated.

11. In the **Reference Description** column, if you want to edit the default description of the reference or add a new description, select the description corresponding to the reference name and type the new description.

Note: The description should not exceed 32 characters.

12. You can replace an annotation reference by clicking **Replace** on the shortcut menu that appears when you right-click the annotation reference in the **References** list and selecting a new reference entity of a similar type.

Note: You cannot replace references that are added after the annotation is created. However, you can remove the unwanted references and add new references.

13. To add more Annotation Elements to the Annotation feature, click **Add** in the **ANNOTATION FEATURE** dialog box and repeat Steps 3 through 11.
14. Click **OK** or click middle-mouse button to close the **ANNOTATION FEATURE** dialog box and create the Annotation feature.

style="x-condition: include_for_toyota;">

Using the ANNOTATION FEATURE Dialog Box

The **ANNOTATION FEATURE** dialog box contains two tabs, **Definition** and **Properties**.

The top portion of the **Definition** tabbed page contains **Active Annotation Orientation** box and **Edit Orientation** button and the elements list.

The **Active Annotation Orientation** box displays the active annotation orientation. The **Edit Orientation** button allows you to change the annotation orientation to be used for creating the next Annotation Element.

For each Annotation Element included in the Annotation feature, the elements list lists:

- **Element Name**—The name of the Annotation Element. When you create the element, Pro/ENGINEER gives it a default name, such as `AE NOTE0`. You can type a different name. The name has to be unique within the model.
- **Type**—The type of the Annotation Element, such as Note, Symbol, and so on.
- **Backup Refs**—The attribute that defines whether the backup references of the Annotation Element are to be created. This column is visible only if you set the `af_copy_references_flag` configuration option to `yes`.

When you select a name of an Annotation Element in the elements list, it becomes active. You can manipulate its references, specify its parameters, edit the annotation item included in it, or remove it from the Annotation feature.

If you right-click on an Annotation Element name in the elements list, the shortcut menu contains the following commands:

- **Edit**—Redefines the properties of the annotation in the selected Annotation Element. Similar to the **Edit** command described below.
- **Repeat**—Creates a new Annotation Element of the same type and the corresponding UI of the selected Annotation Element type appears.
- **Duplicate**—Creates a new Annotation Element that is an exact copy of the selected Annotation Element.
- **Parameters**—Specifies the parameters of the selected Annotation Element. Similar to the **Parameters** command described below, but here you can select multiple elements. In this case, only parameters that are common to all the selected elements appear in the **Parameters** dialog box. If you add parameters, they are added to all selected elements.
- **Current Orientation**—Opens the **CURRENT ANNOTATION ORIENTATION** dialog box and allows you to redefine the current annotation orientation for the Annotation Element.
Note: The **Current Orientation** command is not available for non-graphical Annotation Elements.
- **Text Style**—Modifies the text style of the underlying annotation. This command is not available for non-graphical elements.
- **Remove**—Removes the selected element or elements from the Annotation feature. Deleting an Annotation Element also deletes the underlying annotation. If you want to delete the Annotation Element but keep the annotation, you have to first make the element non-graphical, to disassociate the underlying annotation.
- **Make Non-Graphical**—Change the type of Annotation Element to Non-Graphical and make the underlying annotation independent.

Below the elements list is the **References** collector. It contains the names of the geometric references and the reference description associated with the active Annotation Element. The checkbox next to each reference name indicates whether the reference is Strong or Weak.

The **Definition** tabbed page also contains the following buttons:

- **Add**—Adds an Annotation Element to the Annotation feature. Once you add a new Annotation Element, it becomes active.
- **Edit**—Invokes the user interface that lets you redefine the properties of the annotation item included in the active Annotation Element. For example, if the Annotation Element type is Note, clicking the **Edit** button opens the **Note** dialog box, which lets you edit the note text, modify its placement, attachment, or location, add a hyperlink, and so on. If the Annotation Element type is Non-Graphical, clicking the **Edit** button opens the **Edit Annotation** dialog box, which contains the same options as the **Add Annotation** dialog box and lets you change the annotation type.
- **Parameters**—Lets you specify parameters of the active Annotation Element. Clicking this button opens the **Parameters** dialog box, which lists all the parameters currently associated with the Annotation Element, and lets you create additional parameters and manipulate existing ones.
- **Remove**—Removes the active Annotation Element from the Annotation feature.
- **Columns**—Invokes the **Annotation Columns** dialog box that allows you to select the columns of the Annotation Element list and the **References** list to be displayed. You can select the list that you want to customize from the **Type** list and move the names of the columns that you want to view in the list to the **Displayed** box. You can also specify the width of the displayed columns.
- **References**—Displays a list of Annotation Element references collectors. Click **Annotation References**, **Surface Collector**, and **Chain Collector** to expand or hide the respective collector. These collectors let you select additional references of the appropriate type to be associated with the active Annotation Element. Each of these references can also be designated as Strong or Weak.
 - Clicking the **Details** button next to the **Chains** collector opens the **Chain** dialog box, which provides you with a variety of tools for defining chains of edges or curves.
 - Clicking the **Details** button next to the **Surfaces** collector opens the **Surface Sets** dialog box, which provides you with a variety of tools for defining surface sets.

When references need to be addressed Pro/ENGINEER displays the collector names in red.

If you right-click in any of the collectors, the shortcut menu contains the following commands:

- **Single References Collector**—Activates the **References** collector, which lets you select single geometric references of any type.
- **Curve Collector**—Activates the **Chains** collector, which lets you select chains of edges or curves.
- **Surface Collector**—Activates the **Surfaces** collector, which lets you select sets of surfaces.

The **Properties** tabbed page contains the feature name and an icon to access feature information.

At the bottom of the **ANNOTATION FEATURE** dialog box, there are two buttons:

- **OK**—Create the Annotation feature, or accept its new definition, and close the dialog box.
- **Cancel**—Cancel creation or redefinition of the Annotation feature and close the dialog box.

Manipulating the References of an Annotation Element

The **ANNOTATION FEATURE** dialog box contains three collectors for the geometric references associated with the active Annotation Element:

- The **References** collector is automatically populated by the geometric references of the annotation item included in the Annotation Element, such as the Annotation plane, attachment edges, and so on. These references are grayed out. To change these references, redefine the properties of the annotation item. You can select additional references, such as datum points, coordinate systems, edges, surfaces, and so on.
- The **Chains** collector lets you select additional curves and edges to be associated with the active Annotation Element. The **Details** button next to the **Chains** collector opens the **Chain** dialog box, which provides you with a variety of tools for defining chains of edges or curves.
- The **Surfaces** collector lets you select additional surfaces to be associated with the active Annotation Element. The **Details** button next to the **Surfaces** collector opens the **Surface Sets** dialog box, which provides you with a variety of tools for defining surface sets.

You can perform the following operations on the references:

- To select additional references, click the appropriate collector to activate it, then select the geometric entities on the screen, following the usual rules of selection and collection. Pro/ENGINEER lists the name of each reference in the collector.
- To change the orientation of the annotation, click **Current Orientation** on the shortcut menu that appears when you right-click the annotation in the Annotation Elements list.

Note: The **Current Orientation** command is not available for non-graphical Annotation Elements.

- To edit the default description of the reference or add a new description, in the **Reference Description** column, select the description corresponding to the reference name and type the new description. The description has a limit of 32 characters.
- To remove a reference, right-click its name in the collector, then click **Remove** on the shortcut menu. You can not remove references that were automatically included in the **References** collector.

- To replace an annotation reference, right-click its name in the **References** collector, and click **Replace** on the shortcut menu, and select a new reference entity of a similar type.

Note: You cannot replace references that are added after the annotation is created. However, you can remove the unwanted references and add new references.

- The checkbox next to each reference name indicates whether the reference is Strong or Weak. By default, all references are added as Weak. To change a reference type to Strong, select the checkbox next to the reference name.
- To enable automatic propagation of datum points that you have selected as references, select the checkbox in the **Auto-Propag** column. If you propagate the Annotation Element to data sharing features, then those datum point references are also automatically propagated for which you have selected the checkbox in the **Auto-Propag** column.

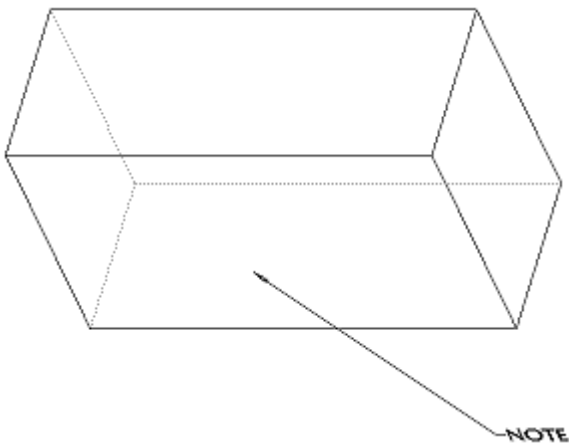
Creating Backup References of an Annotation Element

The **Backup Refs** checkbox next to the Annotation Element name defines whether the backup references of the Annotation Element are to be created within the Annotation Element.

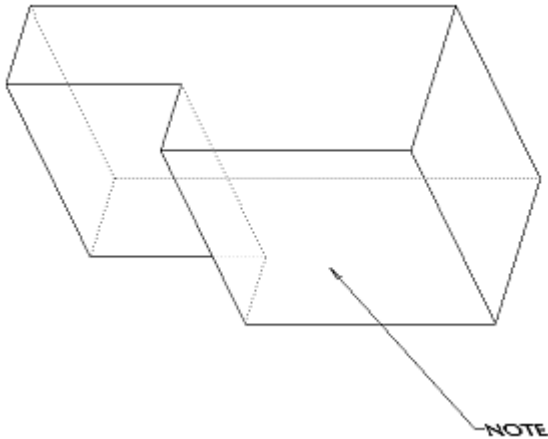
Note: The **Backup Refs** column is visible in the **ANNOTATION FEATURE** dialog box only if you set the `af_copy_references_flag` configuration option to `yes`. Once you created an Annotation feature with this column visible, redefinition will show the **Backup Refs** column even if the configuration option is set to `no`.

Creating the backup references of an Annotation Element ensures that the reference does not change during regeneration of consecutive features.

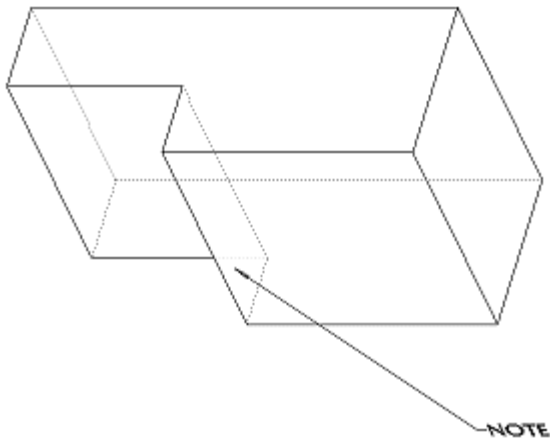
In the following example, an Annotation Element of type Note is created and attached to a middle point of the front surface of the part.



Later, a cut is created. The cut trims the front surface. If the Annotation Element was created without the backup references, the note attachment is recalculated and updated to reference the middle point of the new surface.



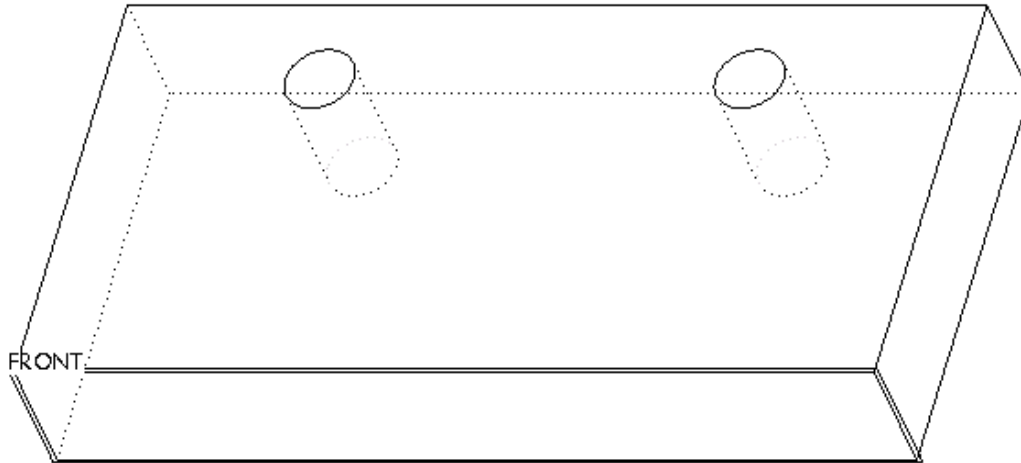
If, at the time of creating or redefining the Annotation Element, the **Backup Refs** checkbox is selected, the system creates an invisible backup surface, and both the Annotation Element and the note reference this backup surface instead of the new surface. In this case, the note attachment is not displaced.




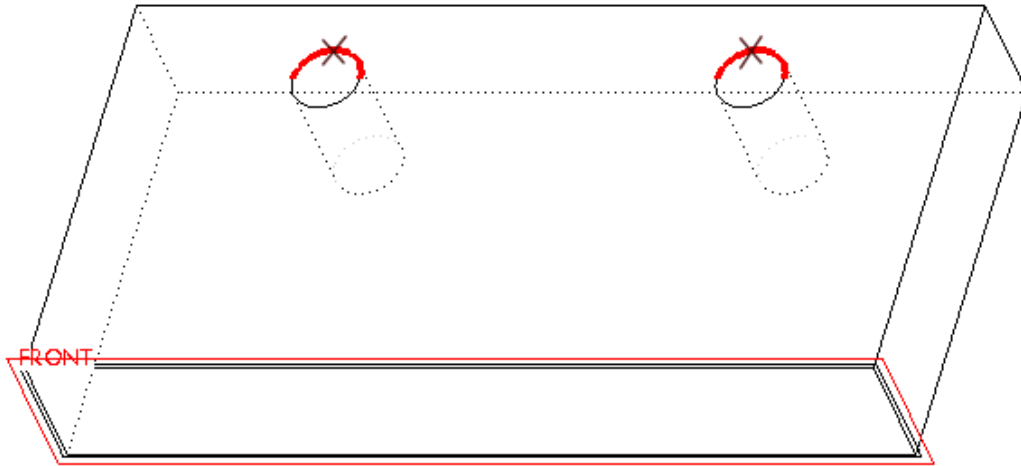
Note: The Backup Refs attribute status is not passed to Data Sharing Features.

Example: Creating an Annotation Feature

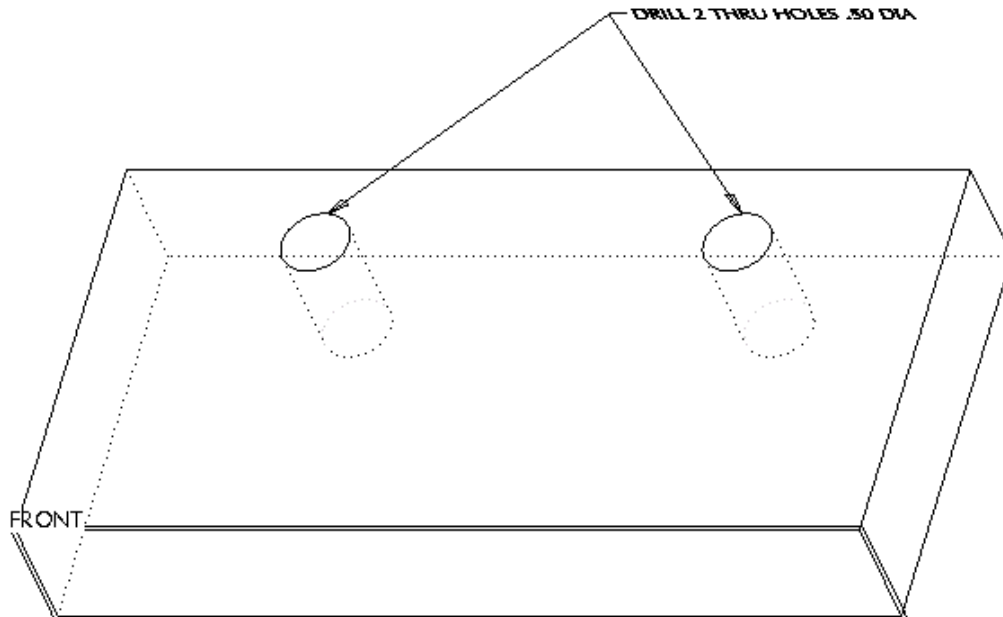
This example describes creating an Annotation feature with one Annotation Element of type Note in the model shown in the following illustration.



1. Define the datum plane named `FRONT` as an Annotation plane using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click  on the Annotation toolbar. The **ANNOTATION FEATURE** dialog box and **Add Annotation** dialog box open simultaneously.
3. If the **Add Annotation** dialog box opens with **Note** selected by default then click middle-mouse button. If **Note** is not selected by default, select **Note** and click **OK**. The **Note** dialog box opens.
4. In the **Text** box, type `DRILL 2 THRU HOLES .50 DIA.`
5. Click the **Place** button in the **Placement** box.
6. On the **NOTE TYPES** menu, click **With Leader**, **Standard**, and **Done**.
7. Select edges of both holes as attachment points, as shown in the next illustration, then click **Done** on the **ATTACH TYPE** menu.



8. Click on the screen to select where the note should be placed. The system places the note, as shown in the following illustration.




9. Click **OK** to return to the **ANNOTATION FEATURE** dialog box. The system creates the Annotation Element with the `FRONT` datum plane as a Strong reference and two edges of the holes as Weak references.
10. Click **OK** to create the Annotation feature.

About Set Datum Tag Annotations in Annotation Elements

You can create a set datum tag as an annotation placed on an annotation plane, and referencing a datum feature or a surface associated with a datum feature. The set datum tag Annotation Elements appear with the datum name in a box that is attached to a solid triangle. Set datum tag Annotation Elements are displayed in the Model Tree in the Annotation Feature as `AE_SET_DATUM_TAG_<Datum_Name>` while the set datum tag annotations are displayed as `Datum_Tag_<Datum_Name>`. Note the following points about set datum tag Annotation Elements.

- When creating a set datum tag Annotation Element, you can reference an existing datum plane or datum axis. Alternatively, you can reference a planar surface, cylindrical surface, circular edge or curve. When you reference a planar surface, cylindrical surface, circular edge or curve, a datum feature is created. For example, if you reference a planar surface then a datum plane is created and if you reference a cylindrical surface, curve, or edge, then a datum axis is created. This datum feature that is created is used as a reference for a set datum tag Annotation Element. After the set datum tag Annotation Element is created, the relationship between the set datum tag Annotation Element and the reference datum feature is a parent-child relationship.
- You can place set datum tag Annotation Elements on a datum, or attach to a dimension or GTOL and on geometry. You can place a set datum tag on geometry, such as, on a coincident surface, edge or curve, coplanar surfaces with set datum planes, and cylindrical surfaces with a set datum axis.
- You can include the set datum tag annotation elements in a User Defined Feature (UDF), only if the UDF contains the reference datum feature, and also the geometry references, if any, of the set datum tag Annotation Elements.
- Similarly, you can propagate set datum tag Annotation Elements to a Data Sharing Feature, only if the Data Sharing Feature contains the references of the set datum tag annotation element. You cannot propagate set datum tag annotation elements in the same model.

To Create a Set Datum Tag Annotation Element

1. Define the active annotation orientation before creating the first annotation in any session using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click  on the Annotation toolbar that is located on the right side of the Pro/ENGINEER window, by default. The **ANNOTATION FEATURE** dialog box and the **Add Annotation** dialog box open simultaneously. The **Add Annotation** dialog box opens with **Note** or the last created annotation in the current session selected by default.
3. Click **Set Datum Tag**. Click **OK**. The **Set Datum Tag** dialog box opens. By default, **Datum** appears selected under **Reference**.
4. To specify a reference for the set datum tag, under **Reference** click one of the following.

Datum—Click **Datum** to specify a datum plane or axis as the reference and then select a datum plane or axis from the model. Pro/ENGINEER uses the selected datum plane to create a set datum tag Annotation Element.

- The name for the selected datum plane or axis is displayed in the **Name** box.
- The **On Datum** option under **Placement** is selected by default.

Geometry—Click **Geometry** to specify a geometry item as the reference and then select a planar surface, cylindrical surface, circular edge, or circular datum curve from the model appropriately. As soon as the entity is selected,

- Depending upon the type of the selected geometry, a new datum plane or datum axis is automatically created. The datum feature is embedded inside the Annotation Feature on the Model Tree. Pro/ENGINEER uses this datum feature to create a Set Datum Tag Annotation Element.
- The **Name** box displays the name of the datum feature.
- The **On Geometry** option under **Placement** is selected by default.

5. If required, type a new name in the **Name** box. This name must not be an existing one.
6. To specify the placement type, under **Placement** click one of the following.
 - **On Datum**—Click to place the set datum tag on the selected datum or the automatically created datum.
 - **In Dim**—Click to place the set datum tag attached to the dimension.
 - **In Gtol**—Click to place the set datum tag attached to the GTOL.
 - **On Geometry**—Click to place the set datum tag on a geometry.

Note: Select a location on a geometry which is coplanar with the datum plane, or coaxial with the datum axis. The set datum tag is placed on the geometry at the selected location.

7. Depending on the type of placement that you selected, perform the following to place the set datum tag.
 - For **On Datum**, the set datum tag is placed on the datum.
 - For **In Dim**, select a dimension in the graphics window. The set datum tag is placed in the dimension.
 - For **In Gtol**, select a GTOL in the graphics window. The set datum tag is placed on the GTOL.
 - For **On Geometry**, select a location on a geometry which is coplanar with the datum plane, or coaxial with the datum axis. The set datum tag is placed on the geometry at the selected location.
8. If required, specify a new location for the set datum tag annotation.

- To specify a new location on the datum axis click **Pick on Datum**. Select a point on the axis. The set datum tag is placed at the selected point.

Note: Pick on Datum is not available if you are creating a set datum tag Annotation Element for a datum plane.
 - Similarly, to specify a new location for the set datum tag annotation on the geometry use **Pick Geom**.
 - To specify a new dimension or GTOL use **Pick Dim** or **Pick Gtol**, respectively.
9. Click **OK** in the **Set Datum Tag** dialog box. The set datum tag is added to the **ANNOTATION FEATURE** dialog box.

Note:

- After you have created the set datum tag annotation, you can change the direction of the solid triangle by clicking **Flip** on the shortcut menu that appears when you right-click the set datum tag annotation element on the Model Tree or in the **ANNOTATION FEATURE** dialog box. Flipping does not affect the text rotation.
- If you remove the set datum tag Annotation Element using the **ANNOTATION FEATURE** dialog box, then the old datum feature continues to exist in the model. The old feature displays the datum name in a box that is attached to the datum feature.

About Editing a Set Datum Tag Annotation Element

When editing the properties of the set datum tag Annotation Element, the **Set Datum Tag** dialog box that opens does not contain the **References** area. Therefore, you cannot change the reference of the set datum tag annotation when editing the properties of the set datum tag annotation element. You can change the placement type and specify a new location for the set datum tag annotation.

Note: When editing the set datum tag in the Drawing mode, you can change the placement type to **In Dim** or **In Gtol**.

You can redefine the reference datum feature to a new placement reference by right-clicking the datum feature on the Model Tree and clicking **Edit Definition** on the shortcut menu. On redefining the reference datum feature to a new placement reference, the set datum tag Annotation Element fails regeneration. To regenerate the set datum tag Annotation Element, redefine the attachment reference of the set datum tag appropriately.

Placement Behavior of Set Datum Tag Annotations

The following points describe the behavior of the set datum tag annotations when you place them on a datum plane or datum axis using the placement options in the **Set Datum Tag** dialog box.

- If you place an annotation using **On Datum**, then the set datum tag annotation appears attached to the datum. If you create a set datum tag annotation for a

datum plane, then the location of the set datum tag annotation on the plane is fixed. However, you can specify its viewing direction and Right direction using its Annotation plane. If you create a set datum tag Annotation for a datum axis, then you can redefine the location of the set datum tag annotation along the axis using **Pick on Datum**. The Annotation plane should be parallel to the axis that the set datum tag references.

- If you place an annotation using **In Dim**, then the set datum tag annotation is placed on the selected dimension. If the dimension is a driving dimension, then the set datum tag annotation is visible only when that dimension is displayed. If the dimension is a driven dimension, then the set datum tag uses the annotation plane of the driven dimension.
- If you place an annotation using **In Gtol**, then the set datum tag uses the annotation plane of the GTOL to which the set datum tag annotation is attached.
- If you place an annotation using **On Geometry**, then you can place a set datum tag annotation created for a datum plane only on a coplanar surface and a set datum tag annotation created for an axis only on a cylindrical surface that is coaxial with the datum axis referenced by set datum tag.

The solid triangle attached to a set datum tag annotation that is placed on a surface is normal to the surface regardless of the direction of text.

You can also place a set datum tag annotation created for an axis on a circular edge or curve if the selected annotation plane contains the selected reference. The set datum tag annotation is normal to the selected edge or curve and the solid triangle appears attached to the edge or curve.


About Ordinate Baselines

Ordinate baseline annotation elements are reference baselines that you can use to create ordinate dimensions. All dimensions that reference the same baseline must share a common plane. You can use an existing ordinate baseline Annotation Element or create a new one to create ordinate dimension Annotation Elements.

Baseline Annotation Elements are created inside an Annotation feature and reference model geometry or datums. You cannot make ordinate baseline Annotation Elements nongraphical as they do not have an equivalent 3D model annotation.

To Create an Ordinate Baseline

1. Define the active annotation orientation before creating the first annotation in any session using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.

2. Click  on the Annotation toolbar located to the right of the Pro/ENGINEER window, by default. The **ANNOTATION FEATURE** and **Add Annotation** dialog boxes open simultaneously. The **Add Annotation** dialog box opens with **Note** or the last created annotation in the current session selected by default.

3. Click **Ord Baseline**. The **ATTACH TYPE** menu appears enabling you to select an entity or a surface as a baseline reference.
4. Click **On Entity** or **On Surface** and select an edge or surface. Select an attachment point and middle-click to select the dimension text placement point. If Pro/ENGINEER cannot determine the direction of the witness line of the ordinate baseline, Pro/ENGINEER prompts you to select an additional reference to define the direction of the witness line of ordinate baseline Annotation Element. The ordinate baseline Annotation Element is added to the **ANNOTATION FEATURE** dialog box.

Working with Annotation Features

About Working with Annotation Features

Annotation features do not create any geometry, therefore, you can not select them on the screen, like other features. The only way to select the Annotation features is from the Model Tree. Therefore, it is very important that the Annotation features are displayed in the Model Tree.

In Part mode, the Annotation features are listed in the Model Tree by default, like any other type of features. In Assembly mode, they are not listed by default. Use the configuration option `mdl_tree_cfg_file`, or follow the procedure for displaying annotation in the Model Tree.

When you work with Annotation features, it is helpful to display the Annotation Elements and individual annotation in the Model tree as well. When you display annotation in the Model Tree:

- You can expand an Annotation feature name, to see all the Annotation Elements listed underneath. This makes it easy to see the structure of an Annotation feature, and to modify the Annotation Elements.
- All the independent annotation items (such as symbols, reference dimensions, and so on), that are not included in Annotation features, are listed at the top of the tree, immediately below the part or assembly name. Notes are listed below its parent (assembly, part, or feature). This makes it easy to see which annotation items are independent. Once an annotation item is included in an Annotation feature, it is consumed by the Annotation Element and is not listed separately in the Model Tree.

Once you set up the proper display of Annotation features in the Model Tree, you can work with them like with other Pro/ENGINEER features: edit definition or references, suppress, resume, delete, and so on. The topics in this section are discussing the specific aspects of working with Annotation features.

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To Display Annotation in the Model Tree

In Part mode, the Annotation features are listed in the Model Tree by default, like any other type of features. In Assembly mode, they are not listed by default. The

Annotation Elements and individual annotation items are not listed by default in either mode. Displaying annotation in the model tree helps working with Annotation features.

To display annotation in the Model Tree:

1. In the Model Tree, click **Settings > Tree Filters**. The **Model Tree Items** dialog box opens.
2. Under **Display**, select **Annotations**. This option controls the display of Annotation Elements and individual annotation items.
3. If the **Features** checkbox is empty, select it. This option controls the display of all features, including Annotation features.
4. Click **OK**.

To Display Annotations in the Graphics Window

You can hide or display the standalone annotations and annotations in Annotation Elements in the graphics window in 3D models using any of the following methods:

- Click  on the datum display toolbar.
- Click **3D Annotations** in the **Datum Display** dialog box that opens when you click **View > Display Settings > Datum Display**.
- Set the configuration option `display_annotations`.

To Select an Annotation Feature

Because Annotation features are only visible in the Model Tree, you can only select them in the Model Tree.

When you select an Annotation feature, the following occurs:

- All the annotations included in the Annotation Elements of the selected Annotation feature are highlighted in the graphics area.
- All the geometric references of all the Annotation Elements of the selected Annotation feature are highlighted in the graphics area.

Once you have selected an Annotation feature, you can rename, reorder, hide, suppress, or delete it, as well as edit its definition or references.

To Redefine an Annotation Feature

Editing the definition of an Annotation feature lets you add, remove, or redefine its Annotation Elements.

To redefine an Annotation feature:

1. Select the Annotation feature in the Model Tree.
2. Click **Edit > Definition**, or right-click on the feature name and select **Edit Definition** from the shortcut menu.

3. Use the **ANNOTATION FEATURE** dialog box to add or remove the Annotation Elements, or edit their properties or parameters.
4. Click **OK** when finished.

To Edit References of an Annotation Feature

Editing references of an Annotation feature lets you specify alternate references for its Annotation Elements.

To edit references of an Annotation feature:

1. Select the Annotation feature in the Model Tree.
2. Click **Edit > References**, or right-click on the feature name and select **Edit References** from the shortcut menu.
3. The system prompts whether you want to roll back the model, that is, temporarily suppress all the features after the Annotation feature being redefined. Click **Yes** or **No**.
4. Use the following commands:
 - **Reroute Feat**—The system highlights each reference in turn. Select a geometric entity of the appropriate type to replace it or click **Same Ref** to keep the existing reference.
 - **Replace Ref**—Select a geometric entity, such as a surface or an edge, then select another entity of the same type to replace it. Click **Sel Feat** and select the Annotation feature in the Model Tree to replace, for all the elements of the Annotation feature, all the references to the first entity by the second entity, or click **All Children** to replace these references in all the features.
5. Click **Done/Return** when finished.

To Replace References of an Annotation Feature

1. Select the Annotation Feature in the Model Tree.
2. Click **Edit > Definition** or right-click the feature name and click **Edit Definition** on the shortcut menu. The **Annotation Feature** dialog box opens.
3. Select the annotation reference that you want to replace from the **References** list, right-click, and click **Replace** on the shortcut menu.

Note: You cannot replace references that are added after the annotation is created. However, you can remove the unwanted references and add new references.

4. Select a new reference entity of a similar type.
 - The new entity replaces the old entity in the **References** list.
 - Depending on the type of reference entity replaced, the display of the Annotation Element is updated in the graphics window. For example, if the

Annotation plane is replaced, then the new Annotation plane is used to display the Annotation Element with the default plane as the Right direction, that is the direction in which the text is read.

To Edit References

To Rename an Annotation Feature

When you create an Annotation feature, it is given a default name, such as Annotation 1.

To change this name:

1. Double-click on the Annotation feature name in the Model Tree, or select the Annotation feature in the Model Tree, right-click on its name and select **Rename** from the shortcut menu.
2. Type the new name in the Model Tree.

To Reorder an Annotation Feature

If any of the Strong references of an Annotation feature are missing, the Annotation feature fails regeneration. If you do not want it to fail, you can either change the reference type to Weak, or reorder the Annotation feature to a different position in the regeneration sequence.

You can reorder Annotation features in the same way as you reorder other types of features:

1. Select the Annotation feature in the Model Tree.
2. Drag it to the desired location in the Model Tree.

You can not reorder an Annotation feature to come before any of its references. For example, if the Annotation feature contains a note attached to an edge of a round, you can not reorder the Annotation feature to come before the Round.

For more information on reordering features, see the Part Modeling documentation.

To Remove an Annotation Feature from Display

You can remove the annotation associated with an Annotation feature from the display in the graphics area, while leaving the Annotation feature itself in the regeneration sequence of the model. Such an Annotation feature behaves as any other Annotation feature: it fails regeneration if any of its Strong references are missing, you can rename it, redefine, reorder, and so on. When you select this Annotation feature in the Model Tree, its geometric references are highlighted in the graphics area.

To turn off the display of annotation associated with an Annotation feature:

1. Select the Annotation feature in the Model Tree.
2. Right-click on the feature name and select **Hide** from the shortcut menu.

All the annotation associated with the Annotation feature is removed from the display. The Annotation feature stays in the Model Tree, but the color of its icon changes to gray, to show that its annotation is hidden.

To turn on the display of annotation associated with a hidden Annotation feature:

1. Select the Annotation feature in the Model Tree.
2. Right-click on the feature name and select **Unhide** from the shortcut menu.

To temporarily remove an Annotation feature both from display and from the regeneration sequence, use the **Suppress** command.

To Suppress or Resume an Annotation Feature

Suppressing an Annotation feature temporarily removes it from the display and from the regeneration sequence.

To suppress an Annotation feature:

1. Select the Annotation feature in the Model Tree.
2. Right-click on the feature name and select **Suppress** from the shortcut menu.
3. The system prompts you for a confirmation. Click **OK** in the **Suppress** dialog box.

All the annotation associated with the Annotation feature is removed from the display and the Annotation feature name is removed from the Model Tree. However, geometric references of the Annotation feature, even though they were highlighted when the Annotation feature was selected, are not suppressed.

Resuming an Annotation feature restores it in the display and the regeneration sequence.

To resume a suppressed Annotation feature:

1. Click **Edit > Resume**.
2. Select one of the options:
 - **Last**—Resumes only the last set of suppressed features.
 - **All**—Resumes all the suppressed features in the model.

To Delete an Annotation Feature

Deleting an Annotation feature permanently removes it from the display and from the regeneration sequence.

To delete an Annotation feature:

1. Select the Annotation feature in the Model Tree.
2. Right-click on the feature name and select **Delete** from the shortcut menu.
3. The system prompts you for a confirmation. Click **OK** in the **Delete** dialog box.

All the annotation associated with the Annotation feature is removed from the display and the Annotation feature name is removed from the Model Tree. However, geometric references of the Annotation feature, even though they were highlighted when the Annotation feature was selected, are not deleted.


Note:

- Annotation that was created as stand-alone and then included in an Annotation feature using the **Existing Annotation** option is not deleted when you delete the Annotation feature. Instead, it reverts to its stand-alone status.
- Deleting an Annotation feature deletes all the annotation associated with it (unless it was added by using the **Existing Annotation** option). If you want to delete the Annotation feature but keep the annotation, you have to first redefine the Annotation feature and make the appropriate elements Non-Graphical, to disassociate the annotation from the Annotation feature.

To Pattern an Annotation Feature

You can pattern an Annotation feature only if some of its references belong to a patterned feature.

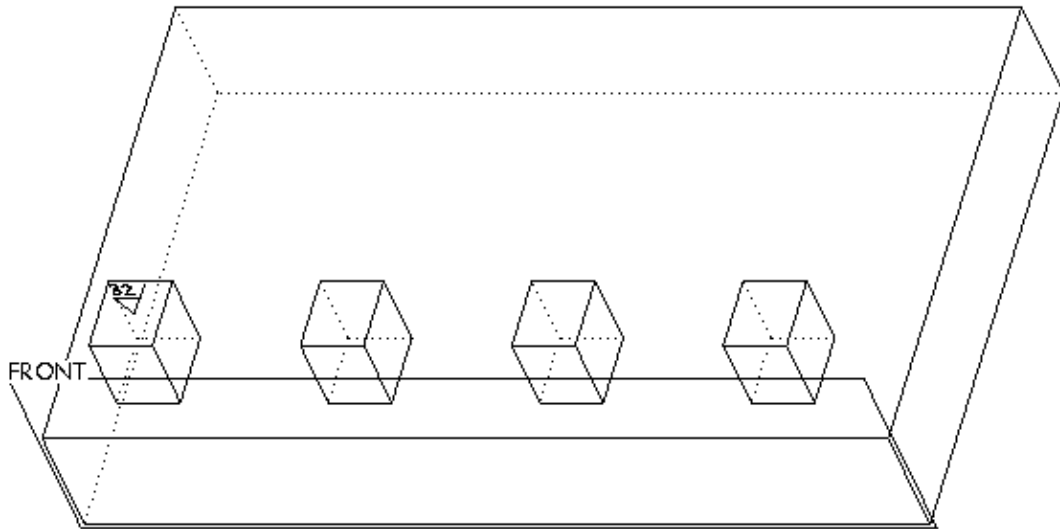
1. Select the Annotation feature in the Model Tree.


2. Click . This icon is available only if the selected Annotation feature has an Annotation Element with at least one reference that belongs to a patterned feature. When you click this icon, the system creates a By Reference pattern of the Annotation feature. This pattern follows the pattern of the feature referenced by the Annotation Element.

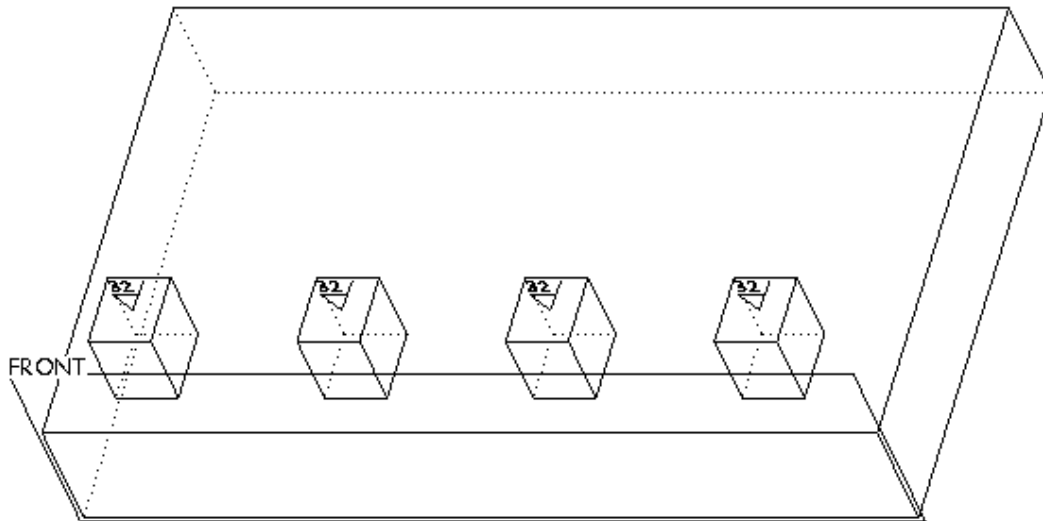
Note: If an Annotation feature has multiple Annotation Elements, some of which reference a patterned entity and some of which reference non-patterned entities, then the Annotation Elements that do not reference the patterned entities will be duplicated but will still reference the same entities as the original Annotation Elements.

Example: Patterning an Annotation Feature

1. Create an Annotation feature with an Annotation Element of type Surface Finish. Place the Surface Finish symbol on the top plane of the patterned protrusion, as shown in the following illustration.




2. Select the Annotation feature in the Model Tree and click . The system creates a By Reference pattern of the Annotation feature, as shown in the next illustration.



About Propagating Annotations in the Same Model

When you create certain feature types such as surface copy, surface offset, or surface mirror, Pro/ENGINEER builds new geometry based on the original geometry, and assigns new IDs to this new geometry. As a result, Annotation Elements of previously created Annotation features do not reference this new geometry, which is

the design intent. Instead of reordering and rerouting the Annotation features to use the new geometry, you can use  to create an Annotation Element propagate feature that contains the propagated Annotation Elements with certain features within the same model. You can propagate Annotation Elements with the following features:

- Surface offset
- Surface copy
- Surface mirror
- Feature created using **Apply Move/Rotate transformations to copies** in the **Paste Special** dialog box.

When you select one of these features and issue a command to propagate its annotation, Pro/ENGINEER does the following:

1. Finds the original geometry based on a new geometry selection.
2. Determines if the original geometry has any Annotation Elements referencing it, and whether all of those Annotation Element references belong to the selected feature. If yes, Pro/ENGINEER propagates the Annotation Elements, that is, creates a single new Annotation Element propagate feature, that contains copies of all the Annotation Elements found in the original geometry, even if they originally belong to different Annotation features. The geometry references in the copies of the Annotation Elements are replaced with the new geometric entities.

These copies are dependent on the original. Any changes to the original Annotation Elements or its references after the Annotation Elements have been propagated are carried over to the Annotation Element propagate feature and the new Annotation Elements in the Annotation Element propagate feature upon regeneration.

You can also propagate Annotation Elements within the same model automatically by setting the value of the `auto_propagate_ae` configuration option to `yes`. The Annotation Element propagate feature is automatically created.

Note: When you create features such as Draft and Expand that replace current geometry that was referenced by Annotation Elements, propagation is not required as Pro/ENGINEER automatically searches for alternate references for existing Annotation Elements.

About the Annotation Element Propagate Feature

The Annotation Element propagate feature contains the propagated Annotation Elements and the references of one or more Annotation Elements. Note the following points about the Annotation propagate feature.

- Upon regeneration of the model after you update the references, Pro/ENGINEER automatically updates the Annotation Element propagate feature and the propagated Annotation Elements in the Annotation Element propagate feature.
- The propagation feature is visible in the Model Tree and you can select it only in the Model Tree. You can reorder the Annotation Element propagate feature by


dragging it in the Model Tree as long as its references precede it in the Model Tree.

- If you remove the references of the Annotation Element propagate feature, the feature cannot be regenerated.
- If you delete the Annotation Element propagate feature, then the Annotation Elements in the feature are also deleted. You can delete an Annotation Element from the Annotation Element propagate feature when redefining the Annotation Element propagate feature.
- If you want to work with Annotation Elements propagated before Pro/ENGINEER Wildfire 3.0, then delete the propagated Annotation Feature and create a new Annotation Element propagate feature in Pro/ENGINEER Wildfire 3.0. Similarly, if you want to use Draft or Solid Offset features created before Pro/ENGINEER Wildfire 3.0, then you must create these features again in Pro/ENGINEER Wildfire 3.0.

Note: The `af_copy_reference_flag` configuration option that is used to back up references of Annotation Elements does not support Annotation Elements in the Annotation Element propagate feature.

Controlling the Propagate Status and Dependency

You can control the propagate status and dependency of the newly propagated Annotation Elements in an Annotation Element propagate feature using the **Copy All** and **Dependent** check boxes on the Annotation Element propagate feature



dashboard that appears when you click .

The status of the **Copy All** and **Dependent** check boxes does not affect the propagate status or dependency status of Annotation Elements that are already in the list of Annotation Elements in the slide up panel. This panel appears when you click **Annotation Elements** on the Annotation Element propagate feature dashboard.


If you do not want any new Annotation Elements to propagate to the Annotation Element propagate feature after you create the Annotation Element propagate feature, then clear the **Copy All** check box before you complete the creation of the Annotation Element propagate feature. Subsequently, even if you add new Annotation Elements in the original Annotation Feature and even if these Annotation Elements reference the correct geometry and qualify for propagation, they do not propagate. If you want new Annotation Elements to propagate, you must select the **Copy All** check box before the completion of the Annotation.


On regeneration, if you want Pro/ENGINEER to automatically make the newly propagated Annotation Elements in the Annotation Element propagate feature to be independent, then clear the **Dependent** check box before completing the creation of Annotation Element propagate feature.

To Propagate Annotations in the Same Model

1. Select the feature or geometry, for example, surface offset or surface mirror, that was built on geometry referenced by Annotation Elements.
2. Click  or **Insert > Model Datum > AE Propagate**. The dashboard for Annotation Element Propagate Feature appears and the propagated annotations are visible in the graphics window.
3. Click **References** on the dashboard.
 - A slide up panel appears with the **Features, Surface Sets, and Chains** collectors. These collectors display references that were selected before you clicked .
 - The **Copy All** and **Dependent** check boxes are selected by default.
4. You can use the **Features, Surface Sets, and Chains** collectors to add or remove features and references.
5. Click **Annotation Elements** on the dashboard. The slide up panel displays the list of Annotation Elements that reference the specified features. The check boxes in the **Propagate** and **Dependency** columns are selected by default.


Note:

- Clearing the check boxes in the **Copy All** and **Dependency** column does not affect the propagation or dependency status of the Annotation Elements that are already present in the list of Annotation Elements in the slide up panel.
 - If you have specified a feature reference after clicking  and clearing the **Copy All** and **Dependent** check boxes, then in the slide up panel for the Annotation Elements, the check boxes in the **Propagate** and **Dependency** columns are not selected.
6. In the **Propagate** column, if the check box next to the Annotation Element name is selected, then the annotation is propagated. If you do not want to propagate an annotation, clear the check box in the **Propagate** column corresponding to the Annotation Element.
 7. In the **Dependency** column, if the check box next to the Annotation Element name is selected, then the propagated Annotation Element is dependent on the original. You can change the status of the check box in the **Dependency** column only when the corresponding check box in the **Propagate** column is selected.
 8. On regeneration, if you do not want Pro/ENGINEER to update the propagated Annotation Elements in the Annotation Element propagate feature, then clear the **Copy All** check box before completing the creation of the Annotation Element propagate feature.

9. On regeneration, if you want Pro/ENGINEER to automatically make the newly propagated Annotation Elements in the Annotation Element propagate feature to be independent, then clear the **Dependent** check box before completing the creation of Annotation Element propagate feature.
10. Click . The new Annotation Element propagate feature appears in the Model Tree.

About Dependent Copies of Annotations

Annotation Elements in an Annotation Element propagate feature are dependent on the original Annotation Feature. An Annotation Element propagate feature is created when Annotation Elements are propagated with features such as surface offset,

surface copy, or surface mirror within the same model automatically or using .

In these propagated Annotation Elements, the position of the annotation in the graphics window is independent of the original. All other properties are dependent on the original. However, after you convert an Annotation Element propagate feature to an Annotation Feature, the propagated Annotation Elements in the new Annotation Feature become detail dependent.

Copies of Annotation Elements created by the following methods too are detail dependent on the original.

- Annotation features that are copied using the **Dimensions and Annotation Element Details Only** option in the **Paste Special** dialog box that opens when you click **Edit > Copy**.
- Annotation features that are copied using the **Copy > Dependent** command on the **FEAT** menu that appears when you click **Edit > Feature Operations**.

In a detail dependent copy, the Annotation Element references and the position of the annotation in the graphics window are independent of the original. All other properties, such as note text, surface finish values, tolerance values of driven dimensions, tolerance values for geometric tolerances, text style, color and parameters are dependent.

If you make changes in the original Annotation Element, these changes are carried over to the dependent copy of the Annotation Element upon regeneration.

Note the following points about dependent copies.

- In the dependent copy, the position of the annotation in the graphics window is independent of the original. All other properties are dependent.
- Text properties of a dependent copy are dependent on the text properties of the original Annotation Element. However, when you attach an Annotation Element to an entity such as a dimension or another GTOL, the text properties of the Annotation Element become the same as that of the entity that the Annotation Element is attached to.

- You cannot modify the parameters created by the original Annotation Element in the dependent copy. However, you can add new parameters with full access to the dependent Annotation Element.
- You cannot modify the dependent properties of copies using tools such as family tables or variable items in UDFs, flexible components, and inheritance features. These tools allow you to specify a variation for the dependent properties. However, on regeneration, Pro/ENGINEER gives a warning that invalid variations have been specified and that these dependent Annotation Elements will remain unchanged.
- If you change the type of the original geometric tolerance, for example, from Straightness to Flatness, the dependency of the copy is removed automatically.

The dependency between Annotation Elements is sequential. For example, consider three Annotation Elements, A, B, and C. B is dependent on A and C is dependent on B. If you change the color of A, then the color of B changes automatically and subsequently the color of C also changes automatically. If you make B independent, C is still dependent on B and any changes to the properties of B are reflected in C.

You can restore or remove the dependency of Annotation Elements.

You can view the parent-child information of Annotation Elements by clicking **Info > Parent/Child** on the shortcut menu that appears when you select an Annotation feature or Annotation Element in the Model Tree or the graphics window and right-click. Alternatively, you can view parent-child information by clicking **Info > Global Reference Viewer** on the menu bar.

Copies of Annotation Features that you create by clicking the **Fully Dependent with options to vary** option and the **Dependent copy** check box in the **Paste Special** dialog box that opens when you click **Edit > Copy** are fully dependent copies. However, in such copied features you can change the copy status of annotations, Annotation Element references and other properties using the **Varied Items** dialog box that opens when you click **Copied feature > Varied items** on the shortcut menu. The properties that you can change are annotation values for surface finish and geometric tolerances, visibility status for note, text style, color, and parameters.

To Remove or Restore the Dependency of Annotation Elements

You can remove or restore the dependency of Annotation Elements in an Annotation Element propagate feature as follows.

- To remove the dependency of an Annotation Element, select the dependent Annotation Element, right-click, and click **Make Independent** on the shortcut menu.
- To restore the dependency of the Annotation Elements, select the dependent Annotation Element, right-click, and click **Make Dependent** on the shortcut menu.

If you have converted an Annotation Element propagate feature to an Annotation Feature, then you can remove or restore the dependency as follows.

- To remove the dependency of an Annotation Element, select the dependent Annotation Element, right-click, and click **Detail Independent** on the shortcut menu.
- To restore the dependency of the Annotation Elements, select the dependent Annotation Element, right-click, and click **Detail Dependent** on the shortcut menu.
- If you change the type of the original geometric tolerance, for example, from Straightness to Flatness, the dependency of the copy is removed automatically. To restore the dependency, change the type of geometric tolerance back to Straightness and click **Detail Dependent** on the shortcut menu.

Note: You can restore dependency only if the original Annotation Element still exists, that is, the original Annotation Element has not been deleted, suppressed, or is not in the rollback state.

To Convert an Annotation Element Propagate Feature to an Annotation Feature

To convert an Annotation Element propagate feature to Annotation Feature, select the Annotation Element propagate feature on the Model Tree, right-click, and click **Convert to AF** on the shortcut menu.

Note: The conversion of an Annotation Element propagate feature to an Annotation Feature is irreversible.

To Select the Original of a Dependent Annotation Element

1. Select the dependent Annotation Element and right-click. The shortcut menu appears.
2. Click **Select Driving AE** on the shortcut menu. The original Annotation Element is selected.

Note: If the dependency between Annotation Elements is sequential, the driving Annotation Element is selected. For example, consider three Annotation Elements, A, B, and C. C is dependent on B and B is dependent on A. If you right-click on C and click **Select Driving AE** on the shortcut menu, the driving Annotation Element A is selected.

About Annotations in Difference Reports

You can generate a report of differences between Annotation features and Annotation Elements in two versions of the same model or drawing. This report describes the differences between the features of the two compared models or drawings, including the differences between Annotation features and differences between Annotation Elements. Models or drawings being compared can have the same or different names.

For User Defined Features and features that are a part of a group, the report displays the group name or the User Defined Feature name with the actual feature in parentheses in the **Item Name** column. This enables you to sort the report for similar grouped features by group name and UDF name. For example, `Group_ABD(AE_1)` where `Group_ABD` is the group name and the `AE_1` is the actual feature.

Pro/ENGINEER generates a difference report by comparing the two versions in the following manner (only the differences are reported).

Annotation Features in the Difference Report

- Annotation feature name
- Number of Annotation Elements in Annotation feature

Annotation Elements in the Difference Report

- Annotation Element type
- Annotation Element name and Annotation name
- Annotation Element dependent and independent options
- Annotation Element references
 - Number of References
 - Reference IDs
 - Type of references (Weak or Strong)
- Annotation Element Parameters
 - Number of parameters
 - Parameter names
 - Parameter values

Annotations in the Difference Report

- Annotation Display Status
- Annotation Active or Inactive status
- Note Properties
 - Text field value of a note
 - Uniform Resource Locator (URL) value of a note
 - Attachment references
- Symbol Properties
 - Change information

- Attachment references
- Geometric Tolerance Properties
 - Geometric Tolerance type
 - Tolerance value
 - Attachment references
 - Text value
- Surface Finish Properties
 - Surface finish values
 - Attachment references
- Driven Dimension and Tolerance
 - References
 - Tolerance value

To Create a Difference Report

1. Open the first of the two parts that you want to compare.
2. Click **Analysis > Compare Part > By Feature**. The **File Open** dialog box opens.

Note: You can compare two assemblies using **Analysis > Compare Assembly by Feature** in the Assembly mode. Similarly, you can compare two drawings using **Analysis > Compare Drawing** in the Drawing mode.
3. Select the second part. The difference report is displayed in the Pro/ENGINEER browser.

Note: If you attempt to open a part with the same name as the first part, you are prompted to rename one of the parts.
4. If required, 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.
5. If required, export the difference report by clicking **Export Report** on 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 exported file in the rich text format as `<model name>_CMPRO.txt` and `<model name>_CMPRO.txt.1`. You can open these saved files in Microsoft Excel.

Note: 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>_CMPRO0.csv and <model name>_CMPRO.txt files are replaced by latest exported file.

Example: Difference Report Containing Annotations

Model Comparison :

BASE MODEL: CRANK1.PRT.1
 BASE VERSION:
 COMPARISON MODEL: CRANK_OLD.PRT.1
 COMPARISON VERSION:

Change List				
Item Type	Item Name	Item Id	Change Type	Change Description
CURVE		72	Geometry	FOUND ONLY IN COMPARISON MODEL
DIM PRTRSN		89	Geometry	DIMENSION VALUE: from 140.1000 to 150.0000
PRTRSN		89	Geometry	REFERENCE(S) DIFFERENT
DIM CUT		210	Geometry	DIMENSION VALUE: from 48.4300 to 46.2400
CUT		210	Geometry	REFERENCE(S) DIFFERENT
RELATIONS			Geometry	DIFFERENT
NOTE		0	Geometry	DIFFERENT
NOTE		2	Geometry	DELETED IN COMPARISON MODEL
NOTE		3	Geometry	NEW IN COMPARISON MODEL
VIEW	(Current_view)		Cosmetic	DIFFERENT
LAYER	03__PRT_ALL_CURVES		Cosmetic	MODIFIED ITEMS
GROUP HEAD	ANNOT	272	Geometry	FOUND ONLY IN COMPARISON MODEL
ANNOTFT	ANNOT(Annotation 1)	267	Geometry	FOUND ONLY IN COMPARISON MODEL
ANNOTFT	ANNOT(Annotation 2)	270	Geometry	FOUND ONLY IN COMPARISON MODEL
GROUP HEAD	CURVE	266	Geometry	FOUND ONLY IN COMPARISON MODEL
CURVE	CURVE(CURVE_1)	39	Geometry	FOUND ONLY IN COMPARISON MODEL
GROUP HEAD	DATUM	273	Geometry	FOUND ONLY IN COMPARISON MODEL
PARAMETERS	D_LARGE		Metadata	FOUND ONLY IN BASE MODEL
PARAMETERS	D_SMALL		Metadata	FOUND ONLY IN BASE MODEL
DTM_PL	M_A	254	Geometry	NOT GROUPED IN BASE MODEL
DTM_PL	M_B	262	Geometry	PLACEMENT DIFFERENT
DTM_AX	M_X	257	Geometry	PLACEMENT DIFFERENT
DTM_AX	M_X	257	Geometry	NOT GROUPED IN BASE MODEL
DIM	d36	36	Drawing	DIMENSION COSMETICS
DIM	d50	50	Drawing	DIMENSION TOLERANCE VALUE: from 0.0100 to 0.0010
GEOM TOL	g10	10	Drawing	NEW IN COMPARISON MODEL
GEOM TOL	g9	9	Drawing	DELETED IN COMPARISON MODEL
SURFACE	sf0	0	Geometry	NEW IN COMPARISON MODEL
FINISH				

Working with Annotation Elements

About Working with Annotation Elements

You create, delete, and redefine the type and references of Annotation Elements while creating or redefining Annotation features.

When you select an Annotation Element on the screen or in the Model Tree, you can:

- Edit the Annotation Element parameters by right-clicking and selecting **Edit Parameters** from the shortcut menu. The **Parameters** dialog box opens. You can add or delete parameters, or modify their values or other properties. You can simultaneously modify parameters of multiple Annotation Elements.
- Modify properties of the annotation included in the Annotation Element by right-clicking and selecting **Properties** from the shortcut menu. Depending on the type of the Annotation Element, the appropriate dialog box (such as **Dimension Properties**, **Note**, and so on) opens to let you modify the properties of the underlying annotation.

- Modify the text style of the annotation included in the Annotation Element by right-clicking and selecting **Text Style** from the shortcut menu. The **Text Style** dialog box opens to let you modify the text style of the underlying annotation.
- For certain types of Annotation Elements, such as Surface Finish or Geometric Tolerance, you can modify the tolerance value of the underlying annotation by right-clicking and selecting **Value** from the shortcut menu, then typing the new tolerance value at the prompt. You can select multiple Annotation Elements of the same type and simultaneously modify value for all the underlying annotations.

To Select an Annotation Element

To select an Annotation Element, you can:

- Select the Annotation Element in the Model Tree.
- Select **Annotation** from the filter list on the Status Bar and select the appropriate annotation on the screen. When you select an annotation on the screen, and the annotation is part of an Annotation Element, the Annotation Element, and not the annotation itself, is selected. All the annotation operations, except **Delete**, can be performed through the Annotation Element.

When you select an Annotation Element, the following occurs:

- The Annotation Element that contains the annotation is highlighted in the Model Tree.
- The annotation is highlighted in primary color in the graphics area.
- All the references of the Annotation Element that contains the selected annotation are highlighted in secondary color in the graphics area.
- If you right-click on the selected Annotation Element, the **Select References** command is available in the shortcut menu. Using this command lets you select all the references of the Annotation Element.

To Modify Parameters of Multiple Annotation Elements

You can simultaneously modify parameters of multiple Annotation Elements:

1. Select multiple Annotation Elements, either by holding down the CTRL key and selecting their names in the Model Tree, or as a result of a search.
2. Right-click on one of the selected names and select **Edit Parameters** from the shortcut menu.
3. The **Parameters** dialog box opens. It lists only the parameters that are common to all the selected Annotation Elements. A parameter is considered common if its name, type, and units are the same for all Annotation Elements. The parameter values (as well as sources, descriptions, designate statuses, restrictions) that are the same for all the selected Annotation Elements are displayed. If a parameter has different values (or sources, descriptions, designate statuses, restrictions) for

different Annotation Elements, the appropriate field is blank. You can add or delete parameters, or modify their values or other properties.

4. Click **OK** when finished. All the changes you have made are applied to all the selected Annotation Elements.

About Moving Annotations in Annotation Elements

You can move annotations in Annotation Elements and standalone annotations by clicking **Move** on the shortcut menu that appears when you select the Annotation Element in the Model Tree or the graphics window and right-click. Moving the pointer to a new location and clicking places the annotation at the new location. Middle-clicking cancels the move operation. Similarly, you can move Annotation Elements that are propagated into data sharing features and Annotation Elements propagated within a model. The position of the propagated annotation is independent of the position of the original Annotation Element.

You can move standalone annotations using **Properties** on the shortcut menu or the menu manager. Similarly, you can also move annotations in Annotation Elements.

You cannot move either annotations in Annotation Elements or standalone annotations if they have been placed directly on an entity.

Driving Dimension Annotation Elements

About Driving Dimension Annotation Elements

You can include model driving dimensions created and owned by features in Annotation Elements by selecting the dimensions and placing them on annotation planes. Such Annotation Elements are called Driving Dimension Annotation Elements (DDAEs) and are created in the feature that owns the dimension. In DDAEs you can include all driving dimensions that you can modify. DDAEs do not have user-defined or system references in the annotation element. Note the following points about DDAEs:

- You cannot make DDAEs nongraphical. You can delete them from the feature that contains them.
- DDAEs can only exist within the feature in which you create them.
- One DDAE can include only one dimension. After a DDAE includes a dimension, another DDAE cannot include the same dimension again.
- If you modify the properties or value of the dimensions in a feature the changes are propagated to the DDAE. Similarly, if you modify the properties or value of DDAEs the changes are propagated to the dimension.
- If you modify the feature such that the DDAE still exists but the Annotation Element's annotation plane is no longer compatible with its current annotation plane, then the Annotation Element becomes inactive.
- If a dimension no longer exists in the feature, Pro/ENGINEER deletes the corresponding DDAE from the feature.


- You can create DDAEs for dimensions that convey the increment and number of instances in a pattern.
- You cannot create a DDAE for the selected dimension of a pattern instance in the Edit mode. However, if one or more dimensions shown in the Edit mode for that instance belong to the leader, then you can create DDAEs for such dimensions, but they will be located under the leader feature and not under the instance.
- You cannot create DDAEs for instances in a pattern. However, you can create DDAEs for the instances in a pattern table, as a pattern table creates new dimensions for the instances when patterning a feature.
- When creating a family table or a relation you can select DDAEs on the Model Tree.
- You can select a DDAE as an existing annotation when defining an annotation orientation. However, when creating a new Annotation Element you cannot select a DDAE as an existing annotation.
- You can change the value of a DDAE by clicking **Value** on the shortcut menu that opens when you right-click the driving dimension annotation on the Model Tree or in the graphics window.
- You can delete a DDAE by clicking **Delete** on the shortcut menu that opens when you right-click the driving dimension annotation on the Model Tree or in the graphics window.
- Component placement dimensions are shown on the Model Tree with the placement constraints of the component. If you create a DDAE with a placement constraint in an assembly, and hide the placement folder using the Model Tree filters, then Pro/ENGINEER does not display the DDAE on the Model Tree.
- Pro/ENGINEER displays DDAEs in the Data Sharing Features (DSFs) and merge features. When switching between merge and inheritance features, the placement constraints remain the same. Therefore, there is no change in the DDAEs.
- You cannot create DDAEs for ordinate dimensions, dimensions in Auto Round feature, and for some dimensions in anatomic features.
- The Undo and Redo functionality for DDAEs is similar to other annotation elements. In addition, the **Undo** and **Redo** commands are also available for actions such as creating and deleting DDAEs and changing the orientation of DDAEs.

About Display of Driving Dimension Annotation Elements

Pro/ENGINEER displays Driving Dimension Annotation Elements (DDAEs) whenever the driving dimensions are displayed. For example, when you edit a feature, create relations, or define family tables. Note that Pro/ENGINEER displays DDAEs in the following cases too:

- DDAEs are located on hidden layers.

- DDAEs are manually hidden using **View > Hide**.
- DDAEs are located in a clipped zone.

If you hide all annotations using , then Pro/ENGINEER does not display DDAEs.

Note: Pro/ENGINEER displays inactive DDAEs in the Edit mode in the same state as before the conversion of the dimensions to DDAEs.

Naming Convention for Driving Dimension Annotation Elements

The format for the name of a DDAE is `DRV_DIM_DX`, where `DX` is the driving dimension identification number, for example, `DRV_DIM_D4`.

The format for independent copies and fully dependent copies of DDAEs is `DRV_DIM_DY`, where `DY` is the new identification number for the independent copy.

All DDAEs have a unique name. If you change the name of a DDAE such that Pro/ENGINEER cannot use the default naming convention to create a unique name, then Pro/ENGINEER increments the default name to make the Annotation Element name unique. For example, `DRV_DIM_DX_#`, where `X` is the identification of the dimension and `#` is an increment that Pro/ENGINEER uses to make the name unique.

About Creating Copies of Driving Dimension Annotation Elements

Independent Copies

When you copy and paste a feature, and if Driving Dimension Annotation Elements (DDAEs) exist in the source feature and its orientation is appropriate, Pro/ENGINEER copies the DDAEs to the target feature. Pro/ENGINEER displays independent copies of the DDAEs only if the orientation is appropriate. If the orientation is not appropriate, then the copies of DDAEs are inactive and are not displayed in the graphics window, but are visible on the Model Tree. In an independent copy, all DDAEs are unique and are not driven by the source dimensions.

Fully Dependent Copies

You can create fully dependent copies by clicking **Fully dependent with options to vary** in the **Paste Special** dialog box. In the fully dependent copy, all dimensions are driven by the dimensions in the source feature. The automatically created copies of DDAEs are therefore, not visible in the copy. However, if you add a driven dimension in the copy to the varied items table such that the primary value or tolerance values are varied, then the dimension becomes a driving dimension. If the source dimension is a DDAE, the varied target DDAE becomes visible.

If you apply transformations to the copy, Pro/ENGINEER creates a unique dimension at the top level to drive the transformations of the copy. This dimension does not exist in the source feature. You can convert the transformation dimensions to DDAEs.

Dimension and Annotation Element Dependent Copies

You can create dependent copies by clicking **Dimensions and annotation details only** in the **Paste Special** dialog box. The automatically created copies of DDAEs are not visible in the copy.

If you apply transformations to the copy, Pro/ENGINEER creates transformation dimensions in the feature that contains them to drive the placement of the copy. These dimensions do not exist in the source feature. You can convert these transformation dimensions to DDAEs.


If DDAEs are present in the source feature you are copying and if you make the dimensions corresponding to these DDAEs in the copy independent then the DDAEs in the copy become visible.

If you copy a source feature without DDAEs and if you make the dimensions in the copy independent, the dimensions become visible driving dimensions and are not driven by the source dimensions. You can convert independent driving dimensions in the dependent copies to DDAEs. If you make changes to the properties of the source dimensions after the dimensions are made independent, then these changes are not propagated to the DDAEs. You can delete DDAEs of independent dimensions in dependent copies.

To Create Driving Dimension Annotation Elements

1. Define the active annotation orientation before creating the first annotation in any session using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.

Note:

- You can use either a Named orientation or a frozen Reference plane to create Driving Dimensions Annotation Elements (DDAEs). Flat-to-screen annotation orientations are not supported for DDAEs.
 - If you skip step 1, then Pro/ENGINEER uses the default active annotation orientation.
2. Select a one or more valid features or components.
 3. Perform one of the following actions:
 - Click **Insert > Annotations > Driving Dimension AE**.
 - Right-click the feature on the Model Tree or in the graphics window and click **Create Driving Dimension AE** on the shortcut menu.
 - Click  on the Model Detailing toolbar. To view the Model Detailing toolbar click **Model Detailing** in the **Toolbars** tabbed page in the **Customize** dialog box. To open the **Customize** dialog box, click **Tools > Customize Screen**.

Note that the menu option, command, and toolbar are not available when all dimensions are converted to DDAEs.

Pro/ENGINEER creates DDAEs in the annotation plane in the selected features or components. If any dimensions are not converted, Pro/ENGINEER displays a message in the message area that one or more dimensions were not converted to DDAEs.

Note: You can also create a driving dimension in the Edit mode. In the Edit mode select the dimensions you want to convert to DDAEs and perform step 3. Pro/ENGINEER creates DDAEs using the active annotation orientation in the features or components you are editing. If the annotation orientation is Driven by plane, Pro/ENGINEER freezes the annotation plane.

Creating Various Annotation Types

Working with Annotation

About Annotation

An annotation is a note-type entity created to store model information without geometry. Annotations consist of:

- Notes
- Symbols
- Driven dimensions
- Reference dimensions
- Ordinate driven dimensions
- Ordinate reference dimensions
- Chamfer dimensions
- Geometric tolerances
- Surface finish symbols

Annotations can exist independently within the model, or be included in Annotation features. If an annotation is part of an Annotation feature, it becomes the graphic representation of an Annotation Element.

Different annotation types are created differently and have different properties. The following help topics describe techniques common to working with all annotation types.

Active and Inactive Status of Annotation

If an annotation is part of an Annotation Element, it can become inactive if all of its Weak references are lost. For example, if an Annotation Element includes a note, and the note is attached to two edges, then the two edges are Weak references. If one of the edges was chamfered somewhere later in the regeneration sequence, this reference is lost and the leader that used to be attached to this edge is not

displayed. The note has only one leader, attached to the second edge. However, if the second edge is also lost, the note would lose its last remaining leader and become inactive.

Inactive annotation exists in the Model Tree, but it is not displayed on the screen.

To Select an Annotation

If an annotation is part of an Annotation Element, selecting it on the screen results in selecting the Annotation Element. You can not directly select an annotation that is consumed by an Annotation Element. All the annotation operations, except **Delete**, can be performed through the Annotation Element.

To select a stand-alone annotation item, you can:

- Select **Annotation** from the filter list on the Status Bar and select the annotation on the screen.
- Select the annotation in the Model Tree.

When you select an annotation on the screen, the following occurs:

- The annotation is highlighted in the Model Tree.
- The annotation is highlighted in primary color in the graphics area.
- All the annotation references are highlighted in secondary color in the graphics area.

To Edit Properties of an Annotation

If an annotation is part of an Annotation Element, you modify its properties by modifying properties of the Annotation Element.

To modify properties of a stand-alone annotation:

1. Select **Annotation** from the filter list on the Status Bar and select the annotation on the screen, or select the annotation in the Model Tree.
2. Right-click on the annotation and select **Properties** from the shortcut menu.
3. Depending on the type of annotation, the appropriate dialog box (such as **Dimension Properties**, **Note**, and so on) opens to let you modify the properties of the annotation.
4. Click **OK** when finished.

To Modify Multiple Annotations

You can simultaneously modify multiple annotations of the same type by using the **Properties** or the **Value** shortcut command.

The following table describes which values in which Annotations can be modified at once:

Type of annotation	Value that can be modified	Shortcut Command
Driven dimension	upper and lower tolerance values	Properties
Geometric tolerance	primary tolerance value	Value
Surface finish	surface finish value	Value

To simultaneously modify multiple annotations:

1. Select multiple annotations of the same type on the screen or in the Model Tree.
2. Right-click on one of the selected names and select the appropriate command from the shortcut menu.
3. Type the new value. The system updates the values for all the selected annotations.

Model Notes

About Model Notes

Model notes are text strings that you can attach to objects. You can attach any number of notes to any object in your model.

You can use model notes to:

- Tell other members of your workgroup how to review or use a model that you have created.
- Explain how you approached or solved a design problem when defining the features of a model.
- Explain changes that you have made to the features of a model over time.

You can include model notes in Annotation features. This way, you can define a library of company-specific annotation, and then place it in various models, as appropriate.


To Create a Model Note

Use the following procedure to create a model note:

1. Define the active annotation orientation before creating the first annotation in any session by using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Notes**. The **MDL NOTES** menu appears.
3. Click **New**. The **Note** dialog box opens.

4. If you want to change the default name of the note, type a name in the **Name** box.
5. Select a parent from the **Parent** list, if necessary. By default, the note belongs to the top-level model, part, or assembly.
6. Type the text for the note in the **Text** box.
7. Use the available options (**Insert**, **Style**, and **Symbols**) to further create your note. For more information, see the topic To Modify a Model Note.
8. Click **Place** under **Placement**. The **NOTE TYPES** menu appears.

Note: You can click the **Place note flat to screen in model space** check box instead of selecting an Annotation plane. In this case, the note does not change its orientation with the model. You cannot include such a note in Annotation features.

9. On the **NOTE TYPES** menu, select a leader type and define the note placement.
10. If you are creating a note with any type of leader, or an On Item note, select the attachment point. You can select multiple points for certain types of leader attachment, then click **Done** on the **ATTACH TYPE** menu.
11. Once you defined the attachment type, the pointer changes to . Click on the screen to select where the note should be placed. Pro/ENGINEER places the note.
12. Click **Hyperlink** in the **Note** dialog box to create a URL and ScreenTip on your note.
13. Click **OK** in the **Note** dialog box.

To Define the Note Placement

When you insert a note, you have a series of options to set up the note properties, such as attachment type and location, leader style, and text position.

1. While creating a note, when you click **Place** in the **Note** dialog box, specify the Annotation plane (or place note flat to screen), the **NOTE TYPES** menu appears with the following commands:
 - **No Leader**—No leader. You bypass any leader setup options and are prompted only for the note location.
 - **With Leader**—Leader attached to a specified point. You have to define the attachment style, select the attachment point or points, then select the note location. If you select multiple attachment points, the note will have multiple leaders.
 - **ISO Leader**—ISO style leader. The same procedure as **With Leader**, creates a note with ISO style leader.

- **On Item**—Note is directly attached to selected item. You bypass any leader setup options and are prompted only for an item to which to attach the note directly.

The following commands define the leader orientation, if a leader is used:


- **Standard**—Uses the default leader type. The note can have multiple leaders.
- **Normal Ldr**—Makes the leader normal to the entity; in this case, the note can have only one leader.
- **Tangent Ldr**—Makes the leader tangent to the entity; in this case, the note can have only one leader.

2. If you are placing a note with a leader, then, once you click **Done** on the **NOTE TYPES** menu, the **ATTACH TYPE** menu appears with the following commands:

- **On Entity**—Creates a leader that is attached to a model edge or curve. 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, the system updates the leader point accordingly.
- **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.

Note: The various leader style options appear only for notes created flat to screen. If a note is placed on an Annotation plane, the only leader style available is Arrow Head.

3. Select the commands that define the attachment type and style, then select the attachment point on the screen. If the note can have multiple leaders, you can select multiple attachment points. Click **Done** on the **ATTACH TYPE** menu when finished.

4. Finally, as the pointer changes to , select the location for the note.

To Create a Hyperlink in a Note

You can associate a hyperlink and screen tip with a model note during note creation, or attach them to an existing note in your model.

1. When creating or modifying a note, click **Hyperlink** in the **Note** dialog box.

At any other time, select the note on the screen or in the Model Tree, right-click on it, and select **Add Link** from the shortcut menu.

The **Edit Hyperlink** dialog box opens.

2. In the text box, type a URL or internal link that you wish to associate with the Note.
3. (Optional) To create a screen tip:

Click **Screen Tip**. The **Set Hyperlink ScreenTip** dialog box opens.

Type your text in the text box. Click **Ok**.

4. Click **Ok** in the **Edit Hyperlink** dialog box. The note color changes to indicate there is a hyperlink associated with it. By default, it turns blue and underlined. The actual color and style depend on your **Browser Options**, available under **Tools** in the top menu bar.

If a note has a hyperlink associated with it, you can edit or remove the hyperlink by selecting the note on the screen or in the Model Tree, right-clicking on it, and selecting **Edit Link** or **Remove Link**, respectively, from the shortcut menu.

To Open a Hyperlink Defined in a Model Note

1. Select **Annotation** from the filter list on the Status Bar.
2. Move the cursor over the note with a hyperlink. It highlights and a Screen Tip appears.
3. Press **CTRL**. The pointer changes to a hand icon.
4. Click the mouse. The associated URL opens in the Pro/ENGINEER embedded browser.

You can also open the URL by using the **Open URL** command from the shortcut menu.

1. In the graphics window or the Model Tree, click the note to select it. Right-click the mouse. The shortcut menu displays.
2. Select **Open URL**. The associated URL opens in the Pro/ENGINEER embedded browser.

To Delete or Erase a Model Note

Deleting a note removes the note entirely. Erasing a note removes the note from the screen, but the note still exists. Therefore, it is visible as a sub-level to its parents in the Model Tree, and is available for selection and other operations.

To delete selected notes:

1. Click **Insert > Annotations > Notes**. The **MDL NOTES** menu appears.
2. Click **Delete**.
3. Select the notes you want to delete and click **Ok**. A **Confirm** dialog box opens.
4. Click **Yes** to confirm the deletion and return to the **MDL NOTES** menu.

To erase selected notes or all notes:

1. Click **Insert > Annotations > Notes**. The **MDL NOTES** menu appears.
2. Click **Erase**. The **NOTE DISP** menu opens.
3. Select the notes to erase and click **Ok**, or click **All** to erase all notes for the model.

Another way to delete or erase a note is to select the note on the screen or in the Model Tree, right-click on it, and select **Delete** or **Erase**, respectively, from the shortcut menu.

To Display a Model Note

To display an existing model note:

1. Click **Insert > Annotations > Notes**. The **MDL NOTES** menu appears.
2. Click **Show**. The **NOTE DISP** menu appears. All the notes present in the model are temporarily displayed.
3. Select notes to display, or click **All** to display all notes in the model.

To turn the display of model notes on or off:

1. Click **Insert > Annotations > Notes**. The **MDL NOTES** menu appears.
2. Click **Toggle**. The **NOTE DISP** menu appears. All the notes present in the model are temporarily displayed.
3. Select notes to switch their display, or click **All** to switch display of all notes.

To Modify a Model Note

1. Select **Annotation** from the filter list on the Status Bar and select the note on the screen, or select the note in the Model Tree.
2. Right-click on the note and select **Properties** from the shortcut menu. The **Note** dialog box opens.
3. Use the following options of the **Note** dialog box to change your note:
 - **Insert**—Allows you to insert text from a file or from a note.

- **Style**—Displays the **Text Style** dialog box to change the following, if applicable:

Under **Character**, you can change width factor and slant angle of selected text characters, and you can also set **Underline** on or off. The text height and thickness of a model note cannot be changed in Part or Assembly mode.

Under **Note/Dimension**, you can adjust line spacing, angle, set horizontal and vertical justification, change your note to a mirror image of itself by selecting the **Mirror** check box, and use the **Color** dialog box to change system and user-defined colors.

Click **Reset** to revert to the default settings, **Apply** to make the changes to the current settings to the selected note, and **OK** to make the changes and return to the **Note** dialog box.
 - **Symbols**—Displays the **Symbol Palette** dialog box from which you select a symbol to accompany your note.
 - **Move Text**—Highlights the text for the selected note and allows you to move it. Use the left mouse button to move the text to the new location and the middle mouse button to cancel the move.
 - **Mod Attach**—Displays the **MOD OPTIONS** menu, with the following commands for changing references for the selected note:

Same Ref—Allows you to move the reference to a different point on the same entity.

Change Ref—Displays the **ATTACH TYPE** menu, which allows you to change the reference type and characteristics of a selected reference for the selected note.

Add Ref—Displays the **ATTACH TYPE** menu, which allows you to select a reference point on the model and add a new reference from the selected note.

Delete Ref—Allows you to select and delete a leader for the selected note.
 - **Move**—Highlights the current selected note and allows you to use the mouse to move the entire note and its references. Click the left mouse button to accept the change or the middle mouse button to cancel it.
 - **Unplace**—Removes the selected note. The **Place** button then appears and you can place the note again.
 - **Hyperlink**—Allows you to add or edit a hyperlink and a screen tip for the note, or remove an existing link.
2. Click **OK**.

To Modify Note Text Style

1. Click **Insert > Annotations > Notes**. The **MDL NOTES** menu appears.

2. Click **Text Style** and select the note or notes that you want to change and then click **Ok**. The **Text Style** dialog box opens.
3. Under **Copy From**, you can click **Select Text** to select existing text to modify.
4. Under **Character**, you can change width factor and slant angle of selected text characters, and you can also set **Underline** on or off. The height or thickness of the text in a model note cannot be changed in Part or Assembly mode.
5. Under **Note/Dimension**, you can adjust line spacing and angle, set horizontal and vertical justification, change your note to a mirror image of itself by selecting the **Mirror** checkbox, and use the **Color** dialog box to change system and user-defined colors.
6. Click **Reset** to revert to the default settings, **Apply** to make the changes to the current settings to the selected note, and **OK** to make the changes and return to the **MDL NOTES** menu.

To Move a Model Note

You can move a note using the menu manager or start the process directly on the Model Tree or graphics window. To use the menu manager, see the second procedure in this topic.

Note: To see notes in the model tree click **Settings > Tree Filters > Annotations**. To select notes in the graphics window, select **Annotation** from the Filter list on the Status Bar.

1. Select the note in the Model Tree or graphics window and right click. A shortcut menu appears.
2. Click **Move**. The note and leader line turn gray. A box outline of the note along with the leader line moves with the pointer.

Note: You can also click **Edit > Properties** and then click **Move Text** in the **Note** dialog box.

3. Move the outline of the note to the new location.
4. Click to place the note in the new location or middle-click to cancel the move.

To Move a Note Using the Menu Manager

1. Click **Insert > Annotations > Notes**. The **MDL NOTES** menu appears.
2. Click **Move**.
3. From the Model Tree or graphics window, select the note you want to move and click **OK**. The note and leader line turn gray. A box outline of the note along with the leader line moves with the pointer.
4. With the pointer, move the outline of the note to the new location.
5. Click to place the note in the new location or middle-click to cancel the move.

To Save a Model Note as a Text File

Use the following procedure to save an existing model note in a text file:

1. Click **Insert > Annotations > Notes**. The **MDL NOTES** menu appears.
2. Click **Save Note**.
3. Choose one of the following commands on the **NOTE SAVE** menu:
 - **As Symbols**—Saves the text with the parametric information as symbols. The information in a text file saved using this command contains the note text as well as information required by Pro/ENGINEER to parametrically control the note text. If you include a dimension in the note text, saving the note text As Symbols saves the information required by Pro/ENGINEER to find the dimension in the model and output its value in a note.
 - **As Numeric**—Saves the text with the parametric information as numerical values. The information in the text file saved using this option contains only the note text. If you include a dimension in the note text, saving the note text, saving the note As Numeric saves only the current value of the dimension.
4. Select the model note that you want to save as a .txt file. The **Save Note** dialog box opens.
5. In the **Save Note** dialog box, enter a name for the text file or use the default name. Click **OK** to finish.

Symbols

About Adding Symbols to a Model

Pro/ENGINEER lets you place drafting symbols on a 3D model. A symbol can be attached directly to a surface or other entity, or attached by leaders. It will be graphically presented as if on a plane parallel to the referenced Annotation plane, which you select when you place the symbol. The symbol orientation stays constant with the plane as the plane rotates with the model.

To Add Symbols to a 3D Model

To add symbols to a 3D model, in Part or Assembly mode:

1. Define the active annotation orientation before creating the first annotation in any session by using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Symbol**. The **3D SYMBOL** menu opens with the following commands:
 - **Custom**—Lets you create custom instances from generic definitions.
 - **From Palette**—Lets you create symbol instances using the **Symbol Instance Palette**, which is a .dwg file dedicated to storing frequently used

symbol instances. These symbols are usually simple symbols, without graphic groups or variable text.

- **Move**—Moves an existing symbol. Available only after symbols have been added to the model.
 - **Symbol Gallery**—Opens the **SYM GALLERY** menu, which lets you define new symbols, change symbol definitions, and perform other tasks working with symbols.
3. The **Custom Drawing Symbol** dialog box appears.
 4. Use the **3D SYMBOL** menu commands to place a symbol from the Symbol Palette, or from the **Custom Drawing Symbol** dialog box. The symbol can be attached directly to a surface or other entity, or attached by leaders. It will be placed parallel to the selected Annotation plane. For more instructions on defining symbols, or using the Symbol Palette or the **Custom Drawing Symbol** dialog box, see the documentation for Detailed Drawings.

To Move a Symbol

You can move a symbol using the shortcut menu or the menu manager.

To Move a Symbol Using the Shortcut Menu

1. Select the symbol in the Model Tree or graphics window and right-click. The shortcut menu appears.

Note:

- To view the symbol in the Model Tree, click **Settings > Tree Filters > Annotations**.
 - To select the symbol in the graphics window, select **Annotation** from the Filter list on the status bar.
2. Click **Move** on the shortcut menu.
 3. Move the pointer to the new location. An outline of the symbol along with the leader line moves with the pointer.
 4. Click to place the symbol in the new location or middle-click to cancel the move operation.

To Move a Symbol Using the Menu Manager

1. Click **Insert > Annotations > Symbol**. The **3D SYMBOL** menu opens.
2. Click **Move**.
3. Select the symbol in the Model Tree or graphics window.
4. Move the pointer to the new location. An outline of the symbol along with the leader line moves with the pointer.

- Click to place the symbol in the new location or middle-click to cancel the move operation.

Dimensions

Working with Dimensions

About Z-extension Lines

Z-extension lines are lines that Pro/ENGINEER automatically creates when you position a dimension such that the dimension does not intersect its reference in the Z-direction.

Pro/ENGINEER creates a Z-extension line from the end of the witness line and perpendicular to the annotation plane. The Z-extension line extends to the dimension's reference, making the reference easier to locate. If the Z-extension line does not intersect the dimension's reference, Pro/ENGINEER does not create it.

The Z-extension line is attached to the edge of the surface at a distance closest to the dimension witness line. The dimension witness line is aligned with the edge of the surface closest to the point where you place the dimension text in the direction of the witness line. Pro/ENGINEER displays the extension lines as dashed lines in hidden line color.

Note:

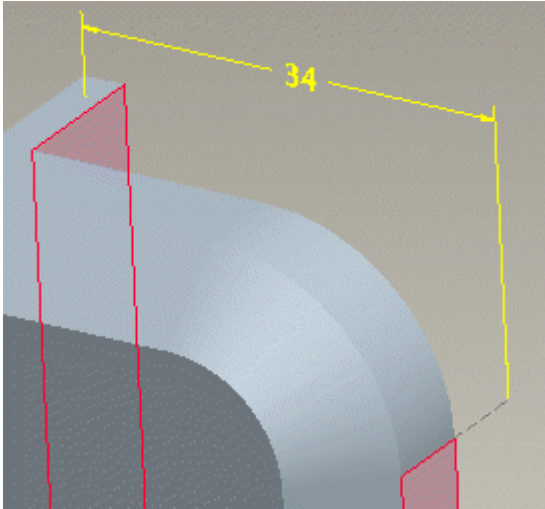
- When you create a dimension using the **Intersect** command on the **ATTACH TYPE** menu, Pro/ENGINEER creates a Z-extension line only if both the entities are located at the point of the intersection.
- When you create a dimension using the **Make Line** command, Pro/ENGINEER does not display the line and the Z-extension line.
- When Pro/ENGINEER creates a Z-extension line, it removes the default gap between the witness line and the dimension's reference.

Example: Z-extension Lines

The following examples are of Z-extension lines in linear, angular, diameter dimensions, and radius dimensions.

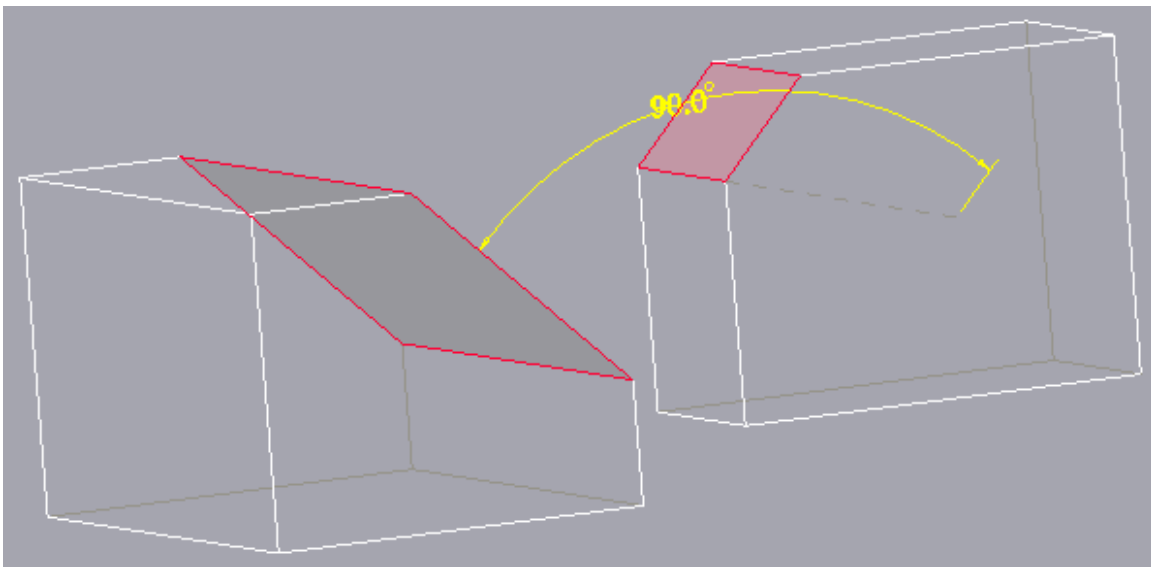
Z-extension Lines in a Linear Dimension

In the following figure, the Z-extension line is attached to the edge of the surface at a distance closest to the dimension witness line.



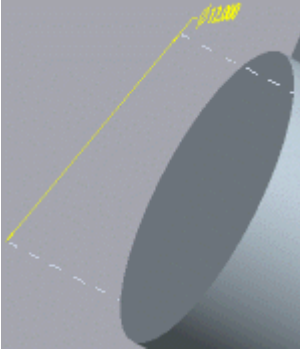
Z-extension Lines in an Angular Dimension

The following figure shows Z-extension lines for angular dimensions.



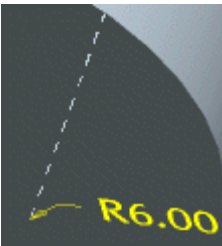
Z-extension Lines in a Diameter Dimension

For diameter dimensions with two arrowheads, the Z-extension line extends from each arrowhead as shown in the following figure.

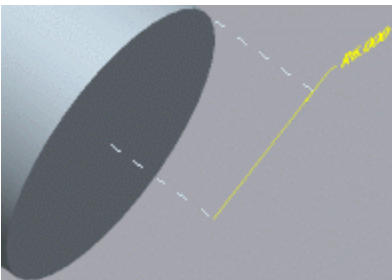


Z-extension Lines in a Radius Dimension

For a radius dimension with a single arrowhead (no extensions), a single Z-extension line extends from the reference to the dimension arrowhead as shown in the following figure.



For a radius dimension shown with flipped arrows with a leader line extension to the center, there are two Z-extension lines. One line extends to the reference, while the other extends to the center of the reference.



About Modifying Attachment References for Dimensions

You can modify the attachment references of all 3D dimensions such as driven or reference dimension Annotation Elements in 3D mode using the **ATTACH TYPE** menu that appears in one of the following ways:


- When you click **Edit Attach** in the **Dimension Properties** dialog box. The **Dimension Properties** dialog box opens when you click **Edit** in the **ANNOTATION FEATURE** dialog box or when you right-click the dimension on the Model Tree or in the graphics window and click **Properties** on the shortcut menu.
- When you click **Edit Attachment** on the shortcut menu that opens when you right-click the annotation on the Model Tree or in the graphics window.

When modifying the attachment references, you cannot create or select a different baseline reference for the dimension. You can modify the attachment reference of an ordinate baseline Annotation Element in 3D mode by replacing the reference from the **ANNOTATION FEATURE** dialog box.

When you edit the attachments for a driven or reference dimension Annotation Element, the new references replace the automatically generated Annotation Element references.

Note: You cannot undo or redo the changes to the dimension attachment references for ordinate baseline Annotation Elements. Therefore, you must be careful when modifying attachment references for ordinate baseline Annotation Elements as changing the references could cause the dependent ordinate dimensions to fail.

To Modify Arrow Head Style of Leader Lines

1. Select a dimension.
2. Move the pointer close to the arrow head that you want to modify till the pointer changes to .
3. Right-click and click **Modify Arrow Style** on the shortcut menu. The **ARROW STYLE** menu appears.
4. Click the required arrow head style on the menu. Pro/ENGINEER updates the style in the graphics window.
5. Click **Done/Return**.

Tip: Using Configuration File Options to Modify the Number of Decimal Places

Using the **Number of decimal places** box in the **Dimension Properties** dialog box, you can modify the number of digits for individual tolerances. This resets a tolerance to a default value corresponding to the specified number of decimal places. You can then change the value of individual tolerances.

The value in the **Number of decimal places** box depends on the setting of the `default_dim_num_digits_changes` configuration option:

- If you set it to `yes` (the default), the system sets the number of digits to the last value you entered. Also, when you create dimensions, it defaults to the number of digits associated with the Num Digits setting.
- If you set it to `no`, it bases the value on the configuration file option `default_dec_places`. Also, it defaults to the value in `default_dec_places` for created dimensions.

When a dimension appears in dual units, the system calculates the tolerance value of secondary units so that it is always within the limits established by the tolerance values in primary units. Thus, truncating, rather than rounding, to the secondary tolerances may occur.

To Modify the Number of Decimal Places in Dimensions

1. Select a dimension and click **Edit > Properties**. The **Dimension Properties** dialog box opens.
2. Type a new value in the **Number of decimal places** box and click **OK**.

To Set Dimensions as Inspection Dimensions

1. Select a dimension and click **Edit > Properties**. The **Dimension Properties** dialog box opens.
2. Click **Inspection** and then click **OK**. The system encloses the value of each selected dimension in an oval box, along with any associated text.

Note: To return the dimension to its original status, click **Neither** in the **Dimension Properties** dialog box.

To Transform Existing Dimensions into Basic Dimensions

1. Select a dimension and click **Edit > Properties**. The **Dimension Properties** dialog box opens.
2. Click **Basic** and then click **OK**. The system encloses each dimension in a feature control frame.


Note: To return the dimension to its original status, click **Neither** in the **Dimension Properties** dialog box.

Working with Witness Lines**About Modifying Witness Lines**

Witness lines are lines that indicate the start and end of a dimension. You can modify the witness lines of valid 3D dimensions as follows:

- Erase witness lines
- Restore erased witness lines
- Insert jogs in witness lines
- Remove jogs from witness lines
- Insert breaks in witness lines
- Remove breaks from witness lines
- Clip witness lines
- Align ends of witness lines
- Skew witness lines

To Erase Witness Lines

- To erase a single witness line:
 - a. Select a dimension.
 - b. Move the pointer close to the witness line that you want to erase till the pointer changes to .
 - c. Right-click and click **Erase Witness Line** on the shortcut menu. The witness line is erased.
- To erase one or more witness lines:
 - a. Select one or more dimensions.
 - b. Right-click outside the model in the graphics window.
 - c. Click **Erase Witness Lines** on the shortcut menu. The valid witness lines are highlighted as you move the pointer over them.
 - d. Move the pointer over a witness line. When it is highlighted, click to select it. If you want to erase more witness lines select additional witness lines by holding down the CTRL key.
 - e. Middle-click. The selected witness lines are erased.
- To erase one or more witness lines using the **Dimension Properties** dialog box:
 - a. Select one or more dimensions.
 - b. Right-click on the Model Tree or in the graphics window.
 - c. Click **Properties** on the shortcut menu and go to the **Display** tabbed page.
 - d. Click **Erase**. The valid witness lines are highlighted as you move the pointer over them.
 - e. Select one or more witness lines and middle-click. The selected witness lines are erased.

Note:


- When you erase a witness line, the arrow next to it automatically changes to the default clipped arrow style, that is, a double arrow. You can modify the clipped arrow style using the **Modify Arrow Style** command.
- You can erase only one witness line of a dimension at any given time. If you attempt to erase the second witness line of the same dimension, the first is automatically restored.
- You cannot erase witness lines of ordinate dimensions.

To Restore Erased Witness Lines

- To restore erased witness lines:
 - a. Select one or more dimensions.

- b. Right-click outside the model in the graphics window.
 - c. Click **Show Witness Lines** on the shortcut menu. All witness lines that are currently erased in the selected dimensions become visible.
- To restore erased witness lines using the **Dimension Properties** dialog box:
 - a. Select one or more dimensions.
 - b. Right-click on the Model Tree or in the graphics window.
 - c. Click **Properties** on the shortcut menu and go to the **Display** tabbed page.
 - d. Click **Show**. All witness lines that are currently erased in the selected dimensions become visible.

To Insert Jogs

- To insert a jog:
 - a. Select a dimension.
 - b. Move the pointer close to the witness line to which you want to add a jog till the pointer changes to .
 - c. Right-click and click **Insert Jog** on the shortcut menu. The witness line on which you want to insert a jog is highlighted as you move the pointer over it.
 - d. Select a start point and an end point of the jog. The jog is added.

Note: The start point must lie on the witness line.
- To insert multiple jogs on one or more witness lines:
 - a. Select one or more dimensions.
 - b. Right-click outside the model in the graphics window.
 - c. Click **Insert Jogs** on the shortcut menu. The witness lines on which you can insert jogs are highlighted as you move the pointer over them.
 - d. Move the pointer over a witness line. When it is highlighted, select a start and an end point for the jog. The jog is added.

Note: The start point must lie on the witness line.


 - e. To add multiple jogs in the selected dimensions repeat step d.
 - f. Middle-click when finished.

Note:

- When you insert a jog in a witness line Pro/ENGINEER adds a handle at the start and end points of the jog. These handles are called jog draggers.
- If you attempt to place the second point of a jog within a break, the break is reduced or eliminated if the second point is beyond the break.

- You can reposition jogs by selecting the dimension and dragging the jog dragger to the required point. The dimension witness line changes shape as you move the dragger. You can move the dragger only in the XY-direction of the current annotation plane.
- To position the jog end points with ease, set the orientation of the model same as the annotation plane of the dimension you are modifying.
- The movement of the jog dragger closest to the attachment point of the dimension is restricted to movement that is parallel to the witness line. You can move additional jogs in the XY-direction.


To Remove Jogs

- To remove a jog from a witness line:
 - a. Select one or more dimensions.
 - b. Place the pointer over a jog dragger and right-click.
 - c. Click **Remove Jog** on the shortcut menu. The jog dragger and its associated jog are removed. Jog dragger are handles at the start and end points of the jog.
- To remove all jogs from a single witness line:
 - a. Select a dimension.
 - b. Move the pointer close to a witness line from which you want to remove a jog till the pointer changes to .
 - c. Right-click and click **Remove Jogs** on the shortcut menu. All jogs on the witness line are removed.
- To remove all jogs from one or more witness lines:
 - a. Select one or more dimensions.
 - b. Right-click outside the model in the graphics window.
 - c. Click **Remove All Jogs** on the shortcut menu. All jogs on the witness lines of all selected dimensions are removed.
- To remove jogs dynamically, drag the jog draggers at the jog points to overlap each other.

Note: You cannot move the jog dragger beyond the clip witness line dragger or the dimension leader line.


To Insert Breaks

- To insert a break in a witness line:
 - d. Select a dimension.

- b. Move the pointer close to the witness line on which you want to insert a break till the pointer changes to .
 - c. Right-click and click **Insert Break** on the shortcut menu.
 - d. Select a start point and an end point for the break on the witness line. A break is added.
- To insert multiple breaks in one or more witness lines:
 - e. Select one or more dimensions.
 - f. Right-click outside the model in the graphics window.
 - g. Click **Insert Breaks** on the shortcut menu. The witness lines in which you can insert breaks are highlighted when you move the pointer over them.
 - h. Move the pointer over a valid witness line. When the witness line is highlighted, select a start and an end point for the break on the witness line. The break is added.
 - i. To add multiple breaks in the selected dimensions repeat steps d.
 - e. Middle-click when finished.

Note: When you insert a break in a witness line, Pro/ENGINEER adds a handle at the start and end points of the break. These handles are called break draggers.

To Remove Breaks

- To remove a break from a witness line:
 - a. Select one or more dimensions.
 - b. Place the pointer over a break dragger and right-click.
 - c. Click **Remove Break** on the shortcut menu. The break is removed. Break draggers are handles at the start and end points of a break.
- To remove all breaks from a single witness line:
 - a. Select a dimension.
 - b. Move the pointer close to a witness line from which you want to remove breaks till the pointer changes to .
 - c. Right-click and click **Remove Breaks** on the shortcut menu. All breaks from the witness line are removed.
- To remove all breaks from one or more witness lines:
 - a. Select one or more dimensions.
 - b. Right-click outside the model in the graphics window.

- c. Click **Remove All Breaks** on the shortcut menu. All breaks from the witness lines of all selected dimensions are removed.
- To remove breaks dynamically, drag the break draggers at the break points to overlap each other.

About Clipping Witness Lines

You can clip witness lines by using the **Clip Witness Lines** or **Clip All** commands on the shortcut menu. During the clipping operation, the end points of the selected witness lines are not aligned.

When you use the **Clip Witness Lines** command:

- The length of all selected witness lines of the selected dimensions is automatically adjusted based on the movement of the handle that you dragged and the current position of individual clipping draggers.
- Selected witness lines of the angular and linear dimensions are clipped at the same time if the selected witness lines are parallel.
- Pro/ENGINEER does not align the ends of witness lines automatically in the 3D mode.

When you use the **Clip All** command:

- The length of each witness line of the selected dimensions changes based on the movement of the handle that you dragged and the current position of individual clipping draggers.
- For linear dimensions, you can clip the witness line only in a direction parallel to the witness line of the selected dragger. If you select multiple dimensions, then witness lines are clipped only for those dimensions whose witness lines are parallel to a dimension's witness line whose dragger you selected and clicked **Clip All**.
- For angular dimensions, only those selected dimensions that share a common vertex with the dimension whose dragger you selected, are clipped. If you select multiple angular dimensions that do not share the same vertex, only those witness lines are clipped whose dimensions share a common vertex with the witness line dragger you selected to initiate the command.
- You cannot clip linear and angular dimension witness lines at the same time.

To Clip Witness Lines

- To clip one or more witness lines:
 - a. Select one or more dimensions.
 - b. Right-click outside the model in the graphics window.
 - c. Click **Clip Witness Lines** on the shortcut menu. The witness lines that you can clip are highlighted when you move the pointer over them.

- d. Move the pointer over a witness line. When it is highlighted, click to select it. If you want to clip more witness lines select additional witness lines by holding down the CTRL key and selecting them.
 - e. Middle-click. The selected witness lines display clipping draggers.
 - f. Adjust the length of a witness line by dragging the handle. The length of each selected witness line is automatically adjusted based on the movement of the handle that you dragged and the current position of individual clipping draggers.
 - g. Click outside the model in the graphics window when finished.
- To clip all witness lines in the selected dimensions:
 - a. Select one or more dimensions.
 - b. Place the pointer over a witness line dragger and right-click.
 - c. Click **Clip All** on the shortcut menu.
 - d. Adjust the length of a witness line by dragging the handle. The length of each witness line in the selected dimensions is automatically adjusted based on the current position of individual clipping draggers.
 - e. Click outside the model in the graphics window when finished.

To Align Ends of Witness Lines

1. Select one or more dimensions.
2. Place the pointer over a clipping dragger of a witness line and right-click.
3. Click **Align Ends** on the shortcut menu. Pro/ENGINEER aligns all the selected ends of witness lines that are parallel to the witness line that you selected and clicked **Align Ends**.

Note:

- For linear dimensions, Pro/ENGINEER aligns the witness lines of all the selected dimensions to points on a plane that intersects the clipping dragger over which you placed the pointer.
- For angular dimensions, Pro/ENGINEER aligns the witness lines of all the selected dimensions to points at the same distance from the vertex as the clipping dragger over which you placed the pointer.

About Skewing Witness Lines

You can skew the witness lines of only one dimension at a time. When you skew the witness lines of a dimension, the witness lines are inclined according to the movement of the skew dragger. You can skew witness lines in a direction parallel to the dimension's leader line. You can skew witness lines of dimensions with Z-extension lines.

You cannot skew the witness lines in the following cases:

- If dimensions have a single witness line.
- If dimensions have witness lines that are on opposite sides of the dimension leader line.

The witness lines snap to their default positions when you move the pointer near the default location of the witness lines.

Note: You can skew only the witness lines of linear dimensions.

To Skew Witness Lines of a Dimension

1. Select one or more dimensions.
2. Right-click outside the model in the graphics window.
3. Click **Skew Dimension** on the shortcut menu. The skew draggers become available for all selected dimensions. The skew dragger is located at the midpoint of the dimension's witness lines at some height above the dimension's text.
4. Drag the skew dragger of the dimension whose witness lines you want to skew. Pro/ENGINEER places the witness lines with an incline according to the movement of the skew dragger. You can skew only one dimension at a time. The skew dragger for that dimension disappears after you finish dragging it. The skew draggers of the other selected dimensions also disappear.

Driven Dimensions

About Driven Dimensions

Driven dimensions are used by Pro/ENGINEER to measure the size and shape of features within a model. The value of a driven dimension changes when the size and shape of the features are modified. Driven dimensions can have tolerances, to which manufactured components can be accepted or rejected.

Manipulation of the tolerances for a driven dimension is similar to that of the tolerances of regular dimensions. You can change the tolerance values and modify their format. A difference is that the driven dimension cannot be set to an upper or lower bound because the nominal value of the driven dimension is not changeable. Its value is derived from the measurement between the two reference entities.

Driven dimensions can be included in Annotation features.

About Annotation Plane Requirements for Dimensions with Surface References

The Annotation plane requirements when creating a driven dimension that includes surface references are as follows:

- Planar surface reference—The annotation plane must be perpendicular to the planar surface reference.

Note:

- If the planar surface reference is perpendicular to the annotation plane and parallel to the other dimension reference, Pro/ENGINEER creates a linear dimension.
- If the planar surface reference is perpendicular to the annotation plane but not parallel to the other dimension reference, then Pro/ENGINEER creates an angular dimension.
- Cylindrical surface reference—The annotation plane must be perpendicular or parallel to the axis of the cylindrical surface.

Note:

- If you select a cylindrical surface reference by clicking it and the annotation plane is perpendicular to the axis of the surface, Pro/ENGINEER creates a radius dimension.
- If you select a cylindrical surface reference by double-clicking it and the annotation plane is perpendicular to the axis of the surface, Pro/ENGINEER creates a diameter dimension.
- If you select a cylindrical surface reference by clicking it or double-clicking it and the annotation plane is parallel to the axis of the surface, no dimension is created and an error message is displayed.
- Conic surface reference—The annotation plane must be perpendicular or parallel to the axis of the cone for a dimension to be created.
- Spherical surface reference—There are no requirements for the annotation plane except those imposed by the other dimension reference.

Note: If you select the spherical surface reference by clicking it or by double-clicking it, Pro/ENGINEER creates a spherical radius dimension or diameter dimension, respectively.

If the Annotation plane requirements are not fulfilled, then Pro/ENGINEER displays a message stating that it is unable to create a dimension.

To Create a Driven Dimension in a Part or Assembly

1. Define the active annotation orientation before creating the first annotation in any session by using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Dimension**. The **REFDIM** menu appears with **Create** selected by default. If not then, click **Create**.
3. Select entities, such as model edges or curves, to create the dimension between them. Use the **ATTACH TYPE** menu commands, if necessary, to attach the dimension to a surface, vertex, a midpoint of an entity, or a center of a circle or arc.
4. Click the middle mouse button to place the dimension.

5. If the dimension is between two points, use one of the **DIM ORIENT** menu commands to specify the dimension orientation. If the dimension is attached to a spherical or cylindrical surface, use one of the **ARC PNT TYPE** menu commands to specify whether the dimension should be created from the center or tangent of the surface.
6. Repeat Steps 2 through 5 to create more driven dimensions using the same Annotation plane.
7. When finished, click **Return > Done**.

Placing Dimensions Using the ATTACH TYPE Menu

The **ATTACH TYPE** menu lets you select the point on the model to which you want to attach a driven or reference dimension. It contains the following commands:

- **On Entity**—Attach the dimension to the entity at the pick point, according to the rules of creating regular dimensions.
- **On Surface**—Attach the dimension to the surface.
- **Midpoint**—Attach the dimension to the midpoint (on the screen) of the selected entity.
- **Center**—Attach the dimension to the center of the circle or arc.
- **Intersect**—Attach the dimension to the closest intersection point of two selected entities.
- **Make Line**—Make a line to reference.

Note: You can change the attachment type in the middle of making a dimension (for example, you can select one point as **On Entity**, then change to **Center** and select another point).

Placing Dimensions Using the DIM ORIENT Menu

When you create a driven or reference dimension between two reference entities, such as, circle or arc centers, datum points, or edge intersections, the **DIM ORIENT** menu lets you specify the dimension placement orientation. The **DIM ORIENT** menu contains the following commands:

- **Horizontal**—Indicates the horizontal distance.
- **Vertical**—Indicates the vertical distance.
- **Slanted**—Indicates the shortest distance between two attachment points. This command is available only when the dimension is attached to points.
- **Parallel**—Indicates distance in the direction parallel to the selected entity. Select an entity to determine the direction.
- **Normal**—Indicates distance in the direction normal to the selected entity. Select an entity to determine the direction.

After you create the Annotation Element you can redefine the dimension orientation using the **DIM ORIENT** menu that appears when you click **Orient** in the **Dimension Properties** dialog box. The **Dimension Properties** dialog box opens when you perform one of the following actions:

- Click **Edit** in the **ANNOTATION FEATURE** dialog box that opens when you right-click the dimension in the model tree and click **Edit Definition** on the shortcut menu.
- Right-click the dimension in the model tree or graphics window and click **Properties** on the shortcut menu.

You can change the orientation of the selected dimension without changing the references or annotation planes. However, the changes to the orientation are retained even if you click **Cancel** in the **Dimension Properties** dialog box after the changes are made. Changing the orientation changes the driven value of the dimension.

Placing Dimensions Using the ARC PNT TYPE Menu

When you create a driven dimension attached to a spherical or cylindrical surface, the **ARC PNT TYPE** menu allows you to specify whether the dimension should be created from the center of the surface or tangent to it. This menu contains the following commands:

Center—Creates a dimension from the axis or a point on the selected surface.

Tangent—Creates a dimension from the tangent of the surface closest to the point selected on the surface with respect to the annotation plane.

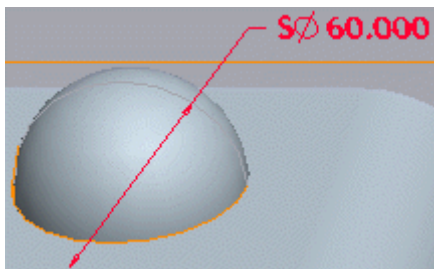
Concentric—Creates a linear dimension between the selected surfaces. This command is available only when both the selected surfaces are concentric.

Note: The **ARC PNT TYPE** menu does not appear for driven dimensions attached to conical surfaces as Pro/ENGINEER automatically creates the dimension with the center as the reference.

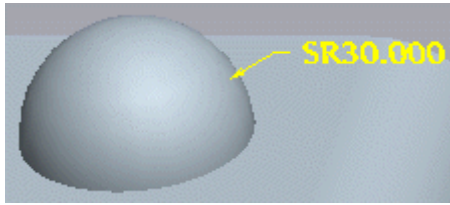
Example: Driven Dimensions With a Spherical Surface Reference

When creating a driven dimension, if you select a spherical surface by clicking it or double-clicking it, Pro/ENGINEER creates a spherical radius or diameter dimension. The following figures display spherical dimensions and the prefix for the numerical dimension values.

Spherical Diameter Dimension



Spherical Radius Dimension



To Modify Tolerances of a Driven Dimension

Manipulation of the tolerances for a driven dimension is similar to that of the tolerances of regular dimensions. You can change the tolerance values and modify their format. A difference is that you can not set an upper or lower limit of the driven dimension, because the nominal value of the driven dimension is not changeable. The system calculates the limits based on the measurement between the two reference entities and the tolerance values.

Note: Make sure your environment is set to display dimension tolerances. Click **Tools > Environment**, select the **Dimension Tolerances** checkbox and click **OK**.

1. Select a driven dimension in the Model Tree or on the screen.
2. Right-click on the dimension name and select **Properties** from the shortcut menu. The **Dimension Properties** dialog box opens.
3. To change the tolerance format, select one of the options from the **Tolerance mode** list:
 - **Nominal**—Only the nominal dimension value is displayed on the screen, but the upper and lower tolerance values are stored with the dimension and available for modification.
 - **Limits**—The dimension is displayed as two numbers, upper and lower limit. Upper limit is the nominal value plus upper tolerance. Lower limit is the nominal value minus lower tolerance. You can not modify the upper and lower limit values of a driven dimension, because its nominal value is not changeable. To modify the tolerance values, set the tolerance format to **(As Is)**.
 - **Plus-Minus**—The dimension is displayed as nominal with plus-minus tolerances. The upper and lower tolerance values are independent.
 - **+ - Symmetric**—The dimension is displayed as nominal with a plus-minus tolerance. The upper and lower tolerance values are the same. If at the time you switch to this format the values are different, the system sets the tolerance to the upper tolerance value.
 - **+ - Symmetric (Superscript)**—Same as **+ - Symmetric**. However, displays the plus-minus tolerance value at the half-way point of the dimension text and extending upwards.
 - **(As Is)**—Preserves the tolerance format and lets you modify the tolerance values.

4. Change the tolerance values as needed by typing the new values in the **Upper tolerance** and **Lower tolerance** boxes (or in the **Tolerance** box if the tolerance format is set to **+ - Symmetric**).
5. Click **OK** when finished.

To Move a Driven Dimension

1. Select the driven dimension in the Model Tree or graphics window and right-click. The shortcut menu appears.

Note:

- To view the driven dimension in the Model Tree, click **Settings > Tree Filters > Annotations**.
 - To select the driven dimension in the graphics window, select **Annotation** from the Filter list on the status bar.
2. Click **Move** on the shortcut menu. Alternatively, click **Move** in the **Dimension Properties** dialog box that opens when you click **Properties** on the shortcut menu.
 3. Move the pointer to the new location.
 4. Click to place the driven dimension in the new location or middle-click to cancel the move operation.

Reference Dimensions

About Reference Dimensions

Reference dimensions appear on models or in drawings for information only. Therefore, they are read-only and cannot be used to modify the model; however, they are automatically updated during regeneration if changes are made to the model. You can include reference dimension created between edges in Annotation features.

You can use reference dimensions in place of some relations. For example, instead of defining a side of a triangle with the relation

$$\text{DIAGONAL} = \text{SQRT} ((\text{D3}/2)^2 + \text{D1}^2)$$

- you can use

$$\text{DIAGONAL} = \text{rd7}$$

Reference dimensions can be created in Part, Assembly, and Sketcher modes. They can also be created in Drawing mode, but only if you have a Detailed Drawings license.

Reference dimensions are distinguished from standard dimensions in either of the following ways:

- They are followed by the notation REF

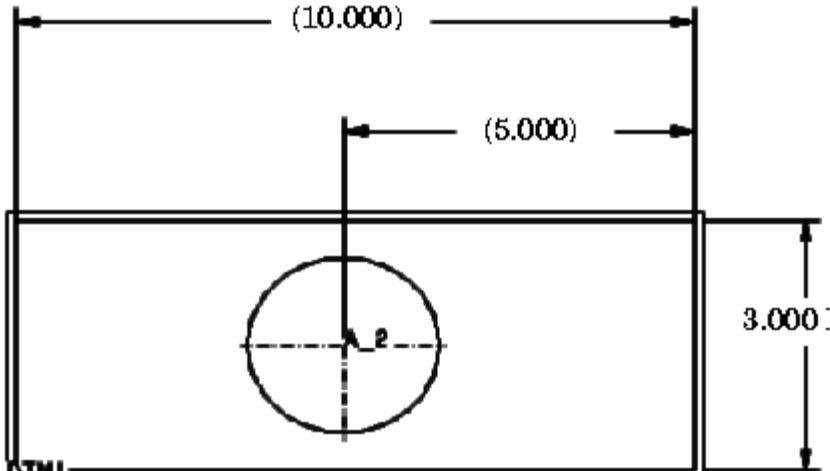
- To have them appear in parentheses as the default, set the configuration file option `parenthesize_ref_dim` to `yes`.

The parameter symbol for reference dimensions is `rd#` (or `rsd#` in the case of Sketcher).

To Create a Reference Dimension

1. Define the active annotation orientation before creating the first annotation in any session by using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Reference Dimension**. The **REFDIM** menu appears with **Create** selected by default. If not then, click **Create**.
3. Click **Create**. The **ATTACH TYPE** menu appears.
4. Use the **ATTACH TYPE** menu commands, if necessary, to attach the dimension to an entity, a surface, a vertex, a midpoint of an entity, or a center of a circle or arc.
5. Select entities, such as model edges or curves, to create the dimension between them. Alternatively, select two surfaces to measure the distance or angle between them.
6. Click the middle mouse button to place the dimension.
7. If the dimension is between two points, use one of the **DIM ORIENT** menu commands to specify the dimension orientation. If the dimension is attached to a spherical or cylindrical surface, use one of the **ARC PNT TYPE** menu commands to specify whether the dimension should be created from the center or tangent of the surface.
8. Repeat Steps 2 through 6 to create more driven dimensions using the same Annotation plane.
9. When finished, click **Return > Done**.

Example: Some Reference Dimensions



To Create a Reference Dimension Between Surfaces

1. Define the active annotation orientation before creating the first annotation in any session using the **ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Reference Dimension**. The **REFDIM** menu appears with **Create** selected by default. If not then, click **Create**.
3. Click **Create**. The **ATTACH TYPE** menu appears.
4. Click **On Surface**.
5. Select two surfaces to measure the distance or angle between them. After you select the second surface, Pro/ENGINEER creates and places the reference dimension.

To Move a Reference Dimension

1. Select the reference dimension in the Model Tree or graphics window and right-click. The shortcut menu appears.

Note:

- To view the reference dimension in the Model Tree, click **Settings > Tree Filters > Annotations**.
 - To select the reference dimension in the graphics window, select **Annotation** from the Filter list on the status bar.
2. Click **Move** on the shortcut menu. Alternatively, click **Move** in the **Dimension Properties** dialog box that opens when you click **Properties** on the shortcut menu.
 3. Move the pointer to the new location.
 4. Click to place the reference dimension in the new location or middle-click to cancel the move operation.

To Delete a Reference Dimension

1. Click **Insert > Annotations > Reference Dimension**. The **REFDIM** menu appears.
2. Click **Delete**.
3. Select the reference dimension.

REF VIEW Menu

The **REF VIEW** menu lets you set options for controlling the visibility of a reference dimension. It opens when you click **Insert > Annotations > Reference Dimension** and then click **Set view** on the **REFDIM** menu. It contains the following commands:

- **Any View**—Make the reference dimension visible in all views.
- **Single View**—Make the reference dimension visible only in the view in which it was created.

Tip: Displaying View-Dependent Reference Dimensions

When you associate a reference dimension with a view, Pro/ENGINEER does not limit the views in which that dimension can be displayed. Instead, Pro/ENGINEER provides a mechanism to display view-dependent reference dimensions in one or all views of the model. No action is required to turn this mechanism on or off.

To Change the View Dependency of a Reference Dimension

1. Reorient the model to the view in which the reference dimensions were created.
2. Click **Insert > Annotations > Reference Dimension**. The **REFDIM** menu appears.
3. On the **REFDIM** menu, click **Show**.
4. Select a feature in the model to show the reference dimensions in the view.
5. On the **REFDIM** menu, click **Set View**. The **REF VIEW** menu appears.
6. On the **REF VIEW** menu, click **Any view** or **Single view**.
7. Select the reference dimensions to which the model changes apply.
8. Click **Done** to accept your selection.

Ordinate Dimensions

About Ordinate Dimensions

Ordinate dimension Annotation Elements and model ordinate dimension annotations use a single witness line and are associated with a baseline reference. Note the following points about ordinate dimensions.

- You can create ordinate driven dimensions in 3D models as model annotations or as Annotation Elements. You must first define an ordinate baseline Annotation Element, and then define model ordinate dimension annotations and ordinate dimension Annotation Elements that reference the ordinate baseline Annotation Element.
- Multiple ordinate dimension annotations and ordinate dimension Annotation Elements can reference the same ordinate baseline Annotation Element. Such annotations and Annotation Elements can belong to one or more annotation Features.
- All ordinate dimension annotations and ordinate dimension Annotation Elements use the annotation plane of the baseline Annotation Element they reference. To change the orientation of an ordinate dimension group, you must change the orientation of the baseline.
- You can select geometry such as a surface, linear entity, point, datum (datum plane or datum axis) or coordinate system as the reference for an ordinate baseline Annotation Element. You can also select an intent object, that is, a surface, linear entity, point, or coordinate system.
- When creating a new ordinate dimension, you can choose to select an existing baseline annotation element or create a new one. If you create a new baseline Annotation Element, Pro/ENGINEER creates a new baseline Annotation Element in the same Annotation feature as the ordinate dimension Annotation Element being created. When creating a standalone ordinate dimension annotation, Pro/ENGINEER creates a new Annotation feature with an ordinate baseline Annotation Element.
- If you choose to use an existing baseline to create a new ordinate dimension, you can select an existing ordinate baseline Annotation Element, or the geometry that it references or the ordinate dimension (either model or Annotation Element).

style="x-condition: include_for_toyota;">About

To Create an Ordinate Dimension

1. Define the active annotation orientation before creating the first annotation in any session by using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Ordinate Driven Dimension** or **Ordinate Reference Dimension**. The **ATTACH TYPE** menu appears enabling you to select an entity, a surface as a baseline.

Note: If the model contains an ordinate baseline Annotation Element, then Pro/ENGINEER prompts you to either select an existing baseline or create a new baseline. To select an existing baseline, click **Select Existing Baseline**. To create a new one, click **Create a New Baseline**.

3. Click **On Entity** or **On Surface** and select an edge or surface. Select an attachment point and middle-click to select the dimension text placement point. An ordinate baseline Annotation Element is created and the **On Entity**, **On Surface**, **Midpoint**, **Center**, **Intersect**, and **Make Line** commands appear on the **ATTACH TYPE** menu.
4. Use the **ATTACH TYPE** menu commands to select entities, such as model edges or curves, a surface, vertex, midpoint of an entity, or center of a circle or arc to create the dimensions.
5. Middle-click to place the dimension.
6. If the dimension is attached to a spherical or cylindrical surface, use one of the commands on the **ARC PNT TYPE** menu to specify whether the dimension should be created from the center or tangent to the surface.
7. Repeat steps 2 through 6 to create more driven dimensions using the same Annotation plane.

Setting the Display Style for Ordinate Dimensions

You can set the display standard for ordinate dimensions by using **Ordinate Style** in the **Properties** tabbed page of the **Dimension Properties** dialog box. Consider the following points when setting the display standard.

- If the model units are English, the default display standard of the dimension is ANSI. However, if the model units are Metric, the default display standard of the dimension is ISO.
- If you change the units of the model after you create an ordinate dimension, the display standard of the existing ordinate dimensions does not change. However, subsequently created ordinate dimensions use the new display standard based on the current model units.
- All dimensions, including the ordinate baseline Annotation Element, in a single ordinate dimension group use the same display standard. Changing the display standard of one dimension changes the display standard of all others in the group. An ordinate dimension group contains all the dimensions that reference the same ordinate baseline Annotation Element.
- If you modify the display standard of an ordinate dimension by using the **Dimension Properties** dialog box after the dimension was created, then the next dimension that you create uses the last used display standard. However, Pro/ENGINEER does not save the display settings when you save the model. The display settings are valid only for a single window in the current Pro/ENGINEER session.

Tolerances

Working with Dimensional Tolerances

Dimensional Tolerance Basics

About Dimensional Tolerances

When you design a part, you specify allowable variations in size referred to as dimensional tolerances. All dimensions are controlled by tolerances (except basic dimensions, which, for the purpose of reference, are considered exact).

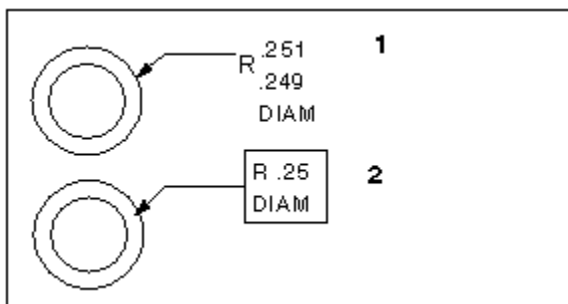
In Pro/ENGINEER, you can express dimensional tolerances in one of two forms: general or individual. General tolerances apply to those dimensions that appear in a nominal format, that is, without tolerances. They are displayed in a tolerance table. Individual tolerances specify individual dimensions.

You can set the tolerance standard as ANSI or ISO and drive dimensional tolerances using a set of tolerance tables.

- When you switch from ISO to ANSI, the system assigns the ANSI tolerances based on the number of digits in the nominal dimension and deletes the tolerance table reference.
- When you switch from ANSI to ISO, a set of tolerance tables drives the ISO-standard tolerances.

About Basic Dimensions

A basic dimension is a theoretically exact dimension and does not have tolerances. The following figure shows how Pro/ENGINEER indicates basic dimensions in a model. You can create basic dimensions using the **GEOM TOL** menu or the **Modify Dimension** dialog box.



1. Dimension in limits format
2. Dimension after changing into basic

To Temporarily Display Dimension Tolerances

Use the following procedure to temporarily display dimension tolerances during your current Pro/ENGINEER session (default settings remain the same):

1. Click **View > Display Settings > Model Display**. The **Model Display** dialog box opens.
2. Under **Display** on the **General** tabbed page, select or clear the **Dimension tolerances** check box.
3. Click **Apply > OK**.

Note: You can also temporarily display tolerance dimensions by the using the following procedure:

- a. Click **Tools > Environment**.
- b. Select or clear the **Dimension tolerances** check box.
- c. Click **Apply > OK**.

To Set the Dimensional Tolerance Format

Set the configuration file option `tol_mode` to one of the following values:

- `nominal`—Displays dimensions without tolerances.
- `limits`—Displays dimensions as upper and lower limits.
- `plusminus`—Displays dimensions as nominal with plus-minus tolerances. The positive and negative values are independent.
- `plusminussym`—Displays dimensions as nominal with a single value for both the positive and the negative tolerance.
- `plusminussym_super`—Same as `plusminussym`. However, displays plus-minus tolerance beginning at the half-way point of the dimension text extending up.

Note:

- When you bring in groups from other models, they carry the Tolerance Display mode that was in effect when they were created.
- In order to change the tolerance display for dimensions, the tolerance display must be shown. If the tolerance table is not visible on the screen, the **Dimension** dialog box will not allow the tolerance mode to be changed.

Tip: Using Configuration File Options to Modify Dimensional Tolerance Format

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.

Controlling the Display of Dimension Tolerances

Control the display of dimension tolerances by setting configuration options and tolerance values. Dimension tolerance values are displayed as one of the following:

- *nominal*—Without tolerance
- *limits*—Upper and lower limits
- *plusminus*—Upper and lower tolerance
- *plusminussym*—Nominal with single tolerance
- *plusminussym_super*—Same as *plusminussym*. However, displays the plus-minus tolerance value at the half-way point of the dimension text extending upwards.

The order by which you set the configuration options to display dimensional tolerances is:

Action	Configuration Option
1. Display tolerance	<i>tol_display</i> to <i>yes</i>
2. Set default type of tolerance display	<i>tol_mode</i> to <i>nominal</i> , <i>limits</i> , <i>plusminus</i> , <i>plusminussym</i> , or <i>plusminussym_super</i>
3. Set display to 1000	<i>display_tol_by_1000</i> to <i>yes</i>
4. View tolerance values in 2D (Drawing mode only)	<i>display_dwg_tol_tags</i> to <i>yes</i>

Note: Pro/ENGINEER can multiply a tolerance value for nonangular dimensions by 1000. The tolerance value is displayed in the graphics area (only) as *x*, but is interpreted by Pro/ENGINEER *0.00x*.

Specifying Default Dimensional Tolerances

When you begin creating a model, Pro/ENGINEER uses default dimensional tolerance values. Therefore, you must specify default tolerances beforehand by setting the configuration file options *linear_tol* and *angular_tol*. Using *linear_tol*, you can set default tolerances for integer dimensions as well.

A linear dimensional tolerance does not have to have the same number of decimal places as the dimension. That is, dimensions can acquire tolerances that have a higher significant number of decimal places (*num digits*).

For example, a dimension with six decimal places can have a tolerance value with seven decimal places. To set the default display of linear dimensional tolerances, specify the value of *linear_tol* as follows:

```
# tolerance x
```

- where # is an integer representing the number of decimal digits for the dimension, tolerance represents the default tolerance value, and x specifies the number of decimal digits for the tolerance.

The x value must be equal to or greater than the # value. If x is less than #, the system uses the default value (equal to the number of decimal digits for the dimension) and rounds it to the significant number of digits of the dimension.

Note: If you do not include x in the value, the system sets the default display equal to the number of decimal digits of the dimension.

Example

```
Linear_tol 0 .05 2
```

For the tolerances specified in the example, a dimension of a whole number would have a tolerance of .05.

When you begin creating a part, the system displays a table at the bottom of the window with the current default tolerances. This table appears only when you have set the configuration file option `tol_display` to `yes`.

If you have not specified tolerances, Pro/ENGINEER assumes the system defaults.

System and User-Supplied Tolerance 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.

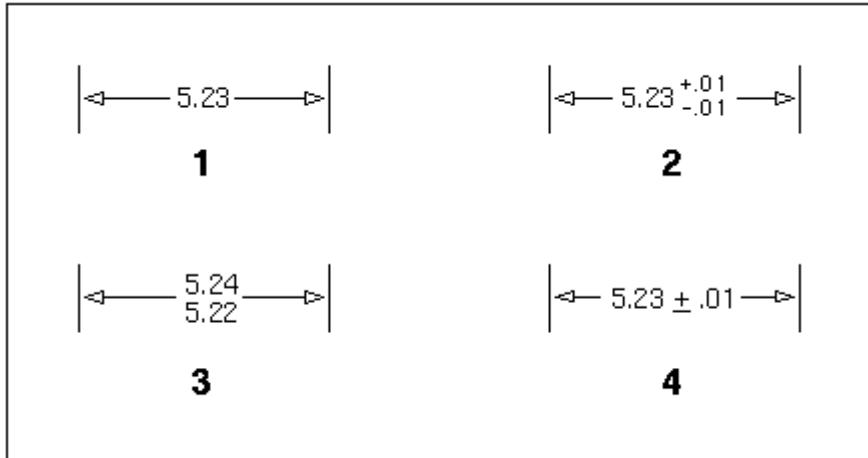
When loading General and Broken Edge tables, keep in mind the following:

- If you load one table that has the same set of class names as the model, 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. When you regenerate the model, the system reassigns all the tolerances of the dimensions from the tolerance tables. If you modify a dimension tolerance, it 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.

Example: Tolerance Formats

The tolerance limit in this example is $x.xx = 0.01$.



1. Nominal
2. Limits
3. Plus-Minus
4. +/- Symmetric

To Modify a Tolerance Dimension Format

1. Select a dimension and click **Edit > Properties**. The **Dimension Properties** dialog box opens.
2. Select a tolerance format from the **Tolerance mode** list.
3. Click **OK**.

To Modify Default Tolerance Values

1. Set the filter to Annotation.
2. Select the default tolerance value in the table shown at the bottom of the screen for ANSI tolerances and click **Value** on the shortcut menu.
3. Type a new value.

To Modify Individual Tolerance Values

1. Select a dimension and click **Edit > Properties**. The **Dimension Properties** dialog box opens.
2. Change any of the following values, as necessary; then click **OK**:
 - For the Limits format, type a new value in the **Upper limit** and **Lower limit** boxes.
 - For the Nominal, Plus-Minus and (As Is) formats, type a new value in the **Nominal value**, **Upper tolerance**, and **Lower tolerance** boxes.
 - For the +/-Symmetric formats, type a new value in the **Nominal value** and **Tolerance** boxes.

Note: The tolerance table must be set to none if a tolerance value of a table-driven tolerance is to be modified.

Setting Up Tolerances

To Set the Dimension Bounds

When you design a model, you expect the actual part dimensions to be within certain predetermined limits of size. Pro/ENGINEER defines these limits of size—upper and lower dimension boundaries—as dimension bounds. The system creates model geometry based on nominal values for each dimension.

You can create a hypothetical model of a part or assembly based on upper and lower allowable dimensions, and later use this new geometry to measure the effects of various analysis functions.

1. Click **Edit > Setup > Dim Bound**.
2. On the **DIM BOUND** menu, do one of the following:
 - Choose **Set All** to set all part dimensions to the specified boundary (in Assembly mode, select parts or subassemblies).
 - Choose **Set Selected** to set only selected dimensions to the specified boundary.
3. Specify the value limits by choosing one of the following commands:
 - **Upper**—Sets the dimension value to its maximum (generates geometry based on a nominal dimension value plus the tolerance).
 - **Middle**—Sets the dimension value to the nominal value plus the mean of the upper and lower tolerance values.
 - **Lower**—Sets the dimension value to its minimum (generates geometry based on a nominal dimension value minus the tolerance).
 - **Nominal**—Sets the dimension value to nominal (generates geometry based on exact ideal dimensions).
4. Select part features to display dimensions whose boundaries must be set.
5. Select the dimensions.
6. Click **Done** to accept the dimension bounds, or click **Quit** to abort.
7. To reset the part to its original state, click **DIM BOUND > Set All > Nominal**.

Note: When you modify dimension bounds, the system generates geometry based on new dimensions. To reflect these changes, it updates reference dimensions present in the model accordingly. The part dimensions appear grey until the dimension bounds are set back to normal.

Tolerance Stack-Up Measurement

When you set new dimension bounds, the system generates new model geometry from which you can obtain various engineering information, for example, tolerance stack-up, clearance and interference calculations, mass properties, assembly constraints, measurement, and reference dimensions.

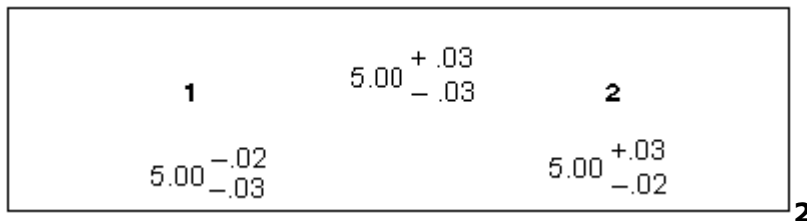
Tolerance stack-up is the cumulative tolerance that occurs when Pro/ENGINEER uses a series of dimensions with tolerances to dimension a certain feature in a part or an assembly member. To determine the tolerance stack-up for a particular model, you must first set dimensional bounds.

For a part, set dimension bounds for selected dimensions; for an assembly, set dimension bounds for selected assembly members. After you have regenerated the model according to the new dimension bound settings, you can use the Measure command from the **Analysis** menu to calculate the tolerance stack-up.

Example: Modifying Tolerance Values

After you modify the tolerance value for a selected dimension, when you switch to the nominal format, the dimension automatically accepts the default tolerance.

For example: Consider a case in which the tolerance of a dimension appears in symmetric format. If you modify it from 100.00 ± 0.01 to 100.00 ± 0.05 , when you change the dimension format back to nominal, the tolerance acquires the nominal default value (for example, ± 0.01). After changing the format back to symmetric, the value appears as 100.00 ± 0.01 .



1

1. Modify upper tolerance to -.02
2. Modify lower tolerance to -.02

To Work with Dimension Bound Groups

1. Click **Edit > Setup > Dim Bound > Dim Bnd Table**.
2. Using the commands in the **DIMBND TABLE** menu, do one of the following:
 - Use **Save Current** to save the current dimension bound information as a separate named set in the dimension bound table.
 - Use **Apply Set** to select a predefined dimension set that is stored in the table and make it the current set in the table.
 - Use **Delete Dim** to remove a dimension from the dimension bound table.

- Use **Edit** to edit the dimension bound table using Pro/TABLE (and enter new dimensions if desired).
- Use **Show** to show the dimension bound table without editing capabilities.

ANSI-Standard Models

To Create an ANSI-Standard Model in Part or Assembly Mode

1. Click **Edit > Setup**. The menu manager opens.
2. Click **Tol Setup > Standard > ANSI**.

Pro/ENGINEER loads the system and user-supplied tables, and the General table drives all dimensions.

ISO-Standard Models

To Create an ISO-standard Model in Part or Assembly Mode

1. Click **Edit > Setup**. The menu manager opens.
2. Click **Tol Setup > Standard > ISO/DIN**.

Pro/ENGINEER loads the system and user-supplied tables, and the General table drives all dimensions.

ISO-Standard Models

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. Because 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 govern the tolerance values of the dimension. You can switch the tolerance table reference of the dimension to any other table.

Changing Tolerance Values

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 **Edit > Setup**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **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 **Edit > Setup > Tol Setup > Tol Tables**. The **TOL TBL ACT** menu appears with the following commands:
 - **Modify Value**—Displays the tables in the **TOL TABLES** menu. You can modify their contents using Pro/TABLE.
 - **Retrieve**—Retrieves a set of tables into the model.
 - **Save**—Saves the tolerance table.
 - **Show**—Displays the tolerance table.
2. Do one of the following:
 - Choose **Modify Value**, select the table by choosing **General Dims** or **Broken Edges**, and select the dimensions.
 - Choose **Holes** or **Shafts**; then type the table name and class number. Hole and shaft tables must be retrieved into the model before values in the table can be modified.

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, remember 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 the tolerances of a dimension, you can not 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).

Example: A Tolerance Table

TABLE_TYPE	GENERAL		
TABLE_NAME	DEFAULTS		
TABLE_UNIT	MILLIMETER		
RANGE_UNIT	MILLIMETER		
DESCRIPTION	0.05–3	3 – 6	6 – 30
FINE	0.05–	0.05	0.1
MEDIUM	0.1	0.1	0.2
COARSE	0.2	0.3	0.5
VERY COARSE	0.5	0.5	1

Example: Tolerance Table (Holes)

The following example shows a Holes table.

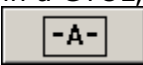

TABLE_TYPE	HOLES				
TABLE_NAME	A				
TABLE_UNIT	MICROMETER				
RANGE_UNIT	MILLIMETER				
BASIC SIZE	9	10	11	12	13
–3	295/270	310/270	330/270	370/270	410/270
3 - 6	300/270	318/270	345/270	390/270	450/270
6 - 10	316/280	338/280	370/280	430/280	500/280
10 - 18	333/290	360/290	400/290	470/290	630/300


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 Geometric Tolerances


About Setting Reference Datum


About Setting Reference Datum

Before you can reference a datum plane or axis in a GTOL, you must set it as a reference. You can set a reference datum using  and .

If you set a reference datum using , then Pro/ENGINEER creates a set datum tag and displays the datum name in a box that is attached to the datum. This is the old style display of set datum tag.


If you set the reference datum using , then Pro/ENGINEER creates a set datum tag as a standalone annotation placed on an annotation plane and displays the datum name in a box attached to a solid triangle. This solid triangle is attached to the datum.

The `old_style_set_datum` configuration option allows you to control the display of  in the **Datum** or **Axis** dialog box. Set the value of this configuration

option to `no` if you do not want the **Datum** or **Axis** dialog box to display . The default for this option is `yes`.

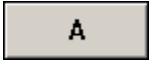
After you have set a datum, you can still use it in the usual way to create features and assemble parts.

To Set a Reference Datum

1. Define the active annotation orientation before creating the first annotation in any session using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Geometric Tolerance**. The **GEOM TOL** menu appears.
3. Click **Set Datum** and select the datum plane or axis. The **Datum** or **Axis** dialog box opens.
 - Under **Type**,  is selected by default.

- Under **Placement, On Datum** is selected by default and Pro/ENGINEER displays the datum name in a box attached to the datum in the graphics window.
 - **On Geometry** is not available.
- Note:** You can also open **Datum** or **Axis** dialog box by clicking **Properties** on the shortcut menu that appears when you right-click a datum plane or a datum axis, respectively, on the Model Tree or in the graphics window.
4. To specify a placement type other than **On Datum**, click one of the following under **Placement**.
 - **In Dim**—Click to place the set datum tag attached to the dimension.
 - **In Gtol**—Click to place the set datum tag attached to the GTOL.
 5. Depending on the placement type that you have selected, perform the following to place the set datum tag.
 - For **In Dim**, select a dimension in the graphics window. The set datum tag is placed on the dimension and is displayed with the datum name in a box that is attached to the dimension.
 - For **In Gtol**, select a GTOL in the graphics window. The set datum tag is placed on the GTOL and is displayed with the datum name in a box that is attached to the GTOL.
 6. If required, change the selection of the dimension or GTOL using **Pick Dim** or **Pick Gtol**.
 7. If required, type a new name in the **Name** box of the **Datum** dialog box and click **OK**.

Note:

- You can change the name by choosing **Name** from the **PART SETUP** or **ASSEM SETUP** menu.
- You can blank a reference datum plane only by placing it on a layer and then blanking the layer.
- The **Surface** and **Define** options in the **Datum** dialog box are used to create 3D datums in drawings.
- To return the datum to its unset state, click .


To Set a Reference Datum Using Set Datum Tag Annotations

1. Define the active annotation orientation before creating the first annotation in any session using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Geometric Tolerance**. The **GEOM TOL** menu appears.

3. Click **Set Datum**, and then select the datum plane or axis. The **Datum** or **Axis** dialog box opens.

Note: You can also open the **Datum** or **Axis** dialog box by clicking **Properties** on the shortcut menu that appears when you right-click a datum plane or datum axis respectively on the Model Tree or in the graphics window.



4. Click  under **Type. On Geometry** under **Placement** is selected by default.
5. To specify the placement type click one of the following under **Placement**.
 - **On Datum**—Click to place the set datum tag on the selected datum.
 - **In Dim**—Click to place the set datum tag attached to the dimension. **Pick Dim** is selected by default.
 - **In Gtol**—Click to place the set datum tag attached to the GTOL. **Pick Gtol** is selected by default.
 - **On Geometry**—Click to place the set datum tag on a geometry. **Pick Geom** is selected by default.
6. Depending on the placement type that you have selected, perform the following to place the set datum tag.
 - For **On Datum**, the set datum tag is placed on the datum in a default position.
 - For **In Dim**, select a dimension in the graphics window. The set datum tag is placed on the dimension. The selected dimension determines the annotation plane of the set datum tag annotation.
 - For **In Gtol**, select a GTOL in the graphics window. The set datum tag is placed on the GTOL. The selected GTOL determines the annotation plane of the set datum tag annotation.
 - For **On Geometry**, select a geometry. The set datum tag is placed on the geometry at the selected location.

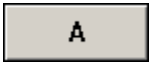
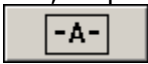
Note: On geometry placement is limited to coplanar surfaces with set datum planes and cylindrical surfaces that share a center with a set datum axis, or any edge by a coaxial cylinder, or a cylindrical curve that is coaxial with the datum axis.
7. If required, specify a new location for the set datum tag annotation.
 - To specify a new location on the datum axis click **Pick on Datum**. Select a point on the axis. The set datum tag is placed at the selected point.

Note: **Pick on Datum** is not available if you are creating a set datum tag annotation for a datum plane.

 - Similarly, to specify a new location for the set datum tag annotation on the geometry use **Pick Geom**.

- To specify a new dimension or GTOL use **Pick Dim** or **Pick Gtol**, respectively.
8. If required, type a new name in the **Name** box of the **Datum** dialog box and click **OK**.

Note:

- You can blank a reference datum plane only by placing it on a layer and then blanking the layer.
- The **Surface** and **Define** options in the **Datum** dialog box are used to create 3D datums in drawings.
- To return the datum to its unset state, click .
- To display the default set datum, that is, display the datum name in a box attached to the datum feature, click .

Creating Geometric Tolerances

About Geometric Tolerances

Geometric tolerances (GTOLs) provide a comprehensive method of specifying where on a part the critical surfaces are, how they relate to one another, and how the part must be inspected to determine if it is acceptable.

They provide a method for controlling the location, form, profile, orientation, and run out of features. When you store a Pro/ENGINEER GTOL in a solid model, it contains parametric references to the geometry or feature it controls—its reference entity—and parametric references to referenced datums and axes. As a result, Pro/ENGINEER updates a GTOL's display when you rename a referenced datum.

In Assembly mode, you can create a GTOL in a subassembly or a part. A GTOL that you create in Part or Assembly mode automatically belongs to the part or assembly that occupies the window; however, the GTOL can refer only to set datums belonging to that model itself, or to components within it. It cannot refer to datums outside of its model in some encompassing assembly, unlike assembly created features.

You can add GTOLs in Part or Drawing mode, but they are reflected in all other modes. Pro/ENGINEER treats them as annotations, and they are always associated with the model. Unlike dimensional tolerances, though, GTOLs do not affect part geometry.

When adding a GTOL to the model, you can attach it to an edge, existing dimension, or existing GTOL, as well as display it as a note without a leader. However, before you place the GTOL, you must set datums and basic dimensions.

```
style="x-condition: include_for_toyota;">
```

To Transform Existing Dimensions into Basic Dimensions

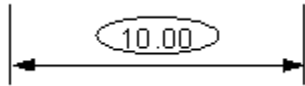
1. Click **Insert > Annotations > Geometric Tolerance**. The **GEOM TOL** menu appears.
2. Click **Basic Dim**.
3. Select dimensions. As you select each one, the system encloses it in a feature control frame.

Note: To change the dimensions back to their original status, choose **GEOM TOL > Clear** before exiting the menu.

4. Click **EXIT > Done**.

To Set Inspection Dimensions

1. You can set dimensions for inspection according to the DIN standard using the **GEOM TOL** menu or the **Modify Dimension** dialog box.



2. Click **Insert > Annotations > Geometric Tolerance**.
3. On the **GEOM TOL** menu, click **Inspect Dim**.
4. Select dimensions. The system encloses the value of each selected dimension in an oval box, along with any associated text.

Note: To change the dimensions back to their original status, choose **GEOM TOL > Clear** before exiting the menu.

5. Click **EXIT > Done**.

To Create a Geometric Tolerance in a Part

To add a GTOL to a model, you may use two, three, or all four tabs of the **Geometric Tolerance** dialog box, depending on the specific characteristics of the GTOL you are creating. This section explains the general process for creating a GTOL in a part and presents an example.

1. Define the active annotation orientation before creating the first annotation in any session by using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Geometric Tolerance**. The **GEOM TOL** menu appears.
3. Click **Specify Tol**. The **Geometric Tolerance** dialog box opens.
4. Specify the model in which to add the GTOL. By default, the current GTOL model is the current model.

5. Select the GTOL type and the reference entity (the entity to which the GTOL applies).
6. Select the placement type and click **Place Gtol**. If you are placing the GTOL on a dimension or another GTOL, select the dimension or GTOL. If you are placing the GTOL with any kind of leader, select the item or items to attach the GTOL.
Note: As you continue creating the GTOL, the system updates it on the model. You can check your work as you go along and make corrections, if necessary.
7. Specify the datum reference or references and material condition or conditions, if applicable.
8. Type a tolerance value and material condition, if applicable.
9. Specify symbols and modifiers, the profile direction, and the projected tolerance zone, if applicable. The completed GTOL now appears in the model.
10. Do one of the following:
 - Close the dialog box and save the changes by clicking **OK**. The system clears the reference entity selection and placement information from the dialog box, but retains all other data. When you reenter GTOL Creation mode, it retains all commands in the previous session of GTOL creation for the object in the current window.
 - Click **New Gtol** to continue.
 - Delete the GTOL from the model by choosing **GEOM TOL > Clear**.
 - Exit GTOL Creation mode and cancel the changes by clicking **Cancel**.










To Specify the Geometric Tolerance Type

1. Click **Insert > Annotations > Geometric Tolerance**.
2. On the **GEOM TOL** menu, click **Specify Tol**. The **Geometric Tolerance** dialog box opens.

The left side of the dialog box displays the GTOL symbols available for selection. When you place the cursor on one of the GTOL symbols, the type appears in the dialog box status line.

The following table shows the types of tolerances available and the appropriate types of entities that you can reference. Pro/ENGINEER guides you in building a GTOL by limiting your choices to those that make sense in the context of the information already provided. The system also does error checking to help you avoid mistakes in selecting entities and datums.

Start by selecting a tolerance type; then select the reference entity, specify tolerance values, and so on. When you have fully defined the GTOL and placed it, the system informs you that the GTOL is complete.

Class	Type	Symbol	Reference Entity
Form	Straightness	—	Surface of revolution, axis, straight edge
	Flatness		Plane surface (not datum plane)
	Circularity	○	Cylinder, cone, sphere
	Cylindricity		Cylindrical surface
Profile	Line		Edge
	Surface		Surface (not datum plane)
Runout	Circular		Cone, cylinder, sphere, plane
	Total		Cone, cylinder, sphere, plane
Orientation	Angularity	∠	Plane, surface, axis
	Parallelism	//	Cylindrical, surface, axis
	Perpendicularity	⊥	Planar surface
Location	Position		Any
	Concentricity		Axis, surface of revolution
	Symmetry		Any

To Specify the Model References to Place a GTOL

1. Click **Insert** > **Annotations** > **Geometric Tolerance**. The **GEOM TOL** menu appears.
2. Click **Specify Tol**. The **Geometric Tolerance** dialog box opens.
3. Use the **Model Refs** tabbed page, to specify the model to which to add the GTOL, choose the reference entity to use, and place the GTOL in the model.

Model—Specifies the model in which to add the GTOL. You must explicitly select the model first. You can select the top model from the **Model** list or click **Select Model** to select a submodel. If you are in Part mode, the current part name appears in the Model field and you have no choice. If you are in Assembly mode, by default the system selects the top assembly.

Reference: To Be Selected—Specifies the reference entity type (edge, axis, surface, feature, datum, entity, or none). Select an item from the **Type** list or click **Select Entity**.

Note: Not all reference entity types are available for all GTOL types. You must select a new entity whenever you change the entity type. You cannot complete the creation of a GTOL until you select a reference entity. The reference entity is the geometry or feature that the GTOL controls; you must not use it in place of a set datum or as an attachment type for the GTOL.

Placement: To Be Placed—Places the GTOL in the model. Select one of the following items from the **Type** list and click **Place GTOL**:

- **Dimension**—Attaches the GTOL to a dimension.
 - **With Leader**—Attaches the GTOL to on or multiple edges or surfaces by using leader lines.
 - **Normal Leader**—Attaches the GTOL to an edge along a leader line that is perpendicular to the selected edge.
 - **Tangent Leader**—Attaches the GTOL to an edge along a leader line that is tangent to the selected edge, orienting the GTOL text box at the same angle as the leader.
 - **Other GTOL**—Attaches the new GTOL to an existing one. You cannot attach the GTOL to an existing GTOL attached to a dimension.
4. To place a GTOL attached to a dimension or other GTOL, select the dimension or GTOL.
5. To place a GTOL attached with any kind of leader,
- a. For **With Leader** placement type, you can select the attachment type:
 - On Entity**—Attaches the GTOL to model edges.
 - On Surface**—Attaches the GTOL to points on model surfaces.
 - b. Select the attachment point for the leader.

For **Normal Leader** and **Tangent Leader** placement, the selected entity also specifies the GTOL orientation.

If you are using **With Leader** placement, you can select multiple attachment points by holding down the CTRL key, then click **Done** when finished.
 - c. Select the placement location for the GTOL anywhere on the screen. The system places the GTOL in this location, with the leaders pointing to the attachment points.

To Specify GTOL Datum References and Material Conditions

Click **GEOM TOL > Specify Tol**. The **Geometric Tolerance** dialog box opens.

Using the **Datum Refs** tabbed page, you can specify the datum references and material conditions for a GTOL, as well as the value and datum reference of a composite tolerance.

- **Datum References**—Specifies the primary, secondary, and tertiary datum references for any GTOL that permits datum references. Click **Primary**, **Secondary**, or **Tertiary** and select an item from the **Basic** and **Compound** lists. The lists contain the currently selected datum and all other datums in the current GTOL model. If you want to select another set datum or axis, click the select arrow. For basic and compound datum references, you can also specify a material condition by selecting an item from the **Basic** and **Compound** lists.
- **Composite Tolerance**—Creates a composite GTOL. Type a value in the **Value** box and select the datum references to show from the **Datum Reference** list.




Note: The reference entity is the geometry or feature that the GTOL controls; you must not use it in place of a set datum or as an attachment type for the GTOL.

To Specify GTOL Tolerance Value and Material Conditions

Click **GEOM TOL > Specify Tol**. The **Geometric Tolerance** dialog box opens.

Using the **Tol Value** tabbed page, you can specify the tolerance value and material condition.

- **Tolerance Value**—Specifies the tolerance value. Type a value in the **Overall Tolerance** box, or select the **Per Unit Tolerance** check box and type a value in the **Value/Unit** and **Unit Length** boxes (or **Unit Area** for some GTOL types).
- **Material Condition**—Specifies the material condition as LMC, MMC, RFS, or RFS/Default.

LMC		Least material condition
MMC		Maximum material condition
RFS		Regardless of feature size
RFS/Default		RFS, but does not show a symbol in frame

To Specify GTOL Symbols and Modifiers

1. Click **Insert > Annotations > Geometric Tolerance**. The **GEOM TOL** menu appears.
2. Click **GEOM TOL > Specify Tol**. The **Geometric Tolerance** dialog box opens.
3. Using the **Symbols** tabbed page, you can specify the following:

- **Symbols and Modifiers**—Specifies the GTOL symbols and modifiers. Select the **ASME Y14.41 Style, Statistical Tolerance, Diameter Symbol, Free State, All Around Symbol, or Tangent Plane** check box.
- **Projected Tolerance Zone**—Specifies the location of the projected tolerance zone. Select **None, Below GTOL, or Inside GTOL**. You can also specify the height of the projected tolerance zone by typing a value in the **Zone Height** box.
- **Profile Boundary** (available for profile GTOLs only)—Specifies unilateral direction, bilateral direction, or unequal tolerance disposition.

Note: The **ASME Y14.41 Style** check box is available only for surface profile GTOLs and the **Unequal** option is available only when **ASME Y14.41 Style** check box is selected. You can control the display of surface profile GTOLs in 3D mode using the **ASME Y14.41 Style** check box.

To Specify the Projected Tolerance Zone

Click **GEOM TOL > Specify Tol**. The **Geometric Tolerance** dialog box opens.

Using the **Symbols** tabbed page, you can specify the projected tolerance zone for GTOLs of the following types: angularity, perpendicularity, parallelism, and position.

- To indicate the projected tolerance zone and specify the height, select the **Zone Height** box and type a value.
- To indicate the projected tolerance zone without specifying the height, placing it on a separate line below the GTOL, select **Below GTOL**.
- To indicate the projected tolerance zone without specifying the height, placing it on the same line next to the GTOL, select **Inside GTOL**.
- To specify no projected tolerance zone, select **None**.

To Specify the Profile Boundary

1. Click **Insert > Annotations > Geometric Tolerance**. The **GEOM TOL** menu appears.
2. Click **Specify Tol**. The **Geometric Tolerance** dialog box opens.
3. Click **Symbols** tab. The **Symbols** tabbed page displays the **ASME Y14.41 Style** checkbox as not selected by default and the **Unilateral** and **Bilateral** options as available. If you click the **ASME Y14.41 Style** checkbox, the **Bilateral** and **Unequal** options are available.
4. To specify the profile boundary for GTOLs, click one of the following:
 - **Unilateral**—Click to apply the tolerance disposition 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. Click **Flip side** to change the direction of application.

When defining the profile boundary for a surface profile GTOL, an arrow in the graphics window indicates the direction of a unilateral disposition application and points away from the selected surface reference.

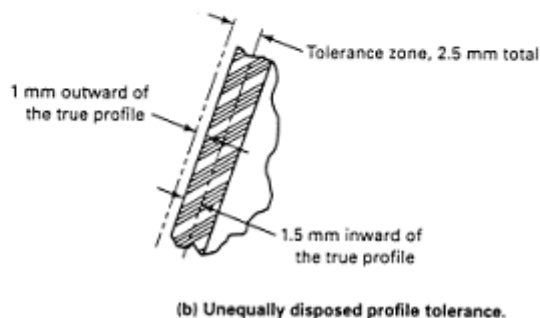
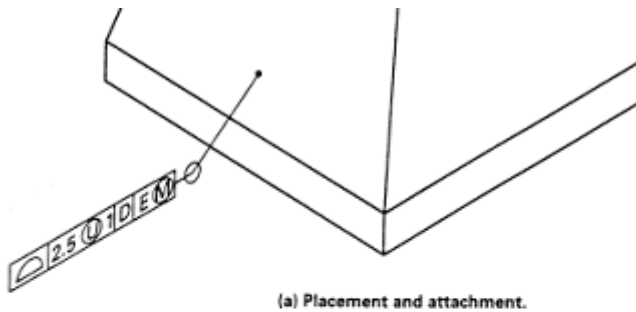
- **Bilateral**—Click to apply the tolerance disposition divided evenly about the selected profile. The annotation displays the tolerance value only.
- **Unequal**—Click to apply the tolerance disposition unequally about the selected profile. In the **Value** box, type a disposition value. This value 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: Unequal disposition is available for surface profile GTOLs only when **ASME Y14.41 Style** checkbox is selected.

Example: Display of Surface Profile GTOLs


The surface profile GTOL annotation displays the tolerance value, the Unequal symbol, and the Unequal disposition value.



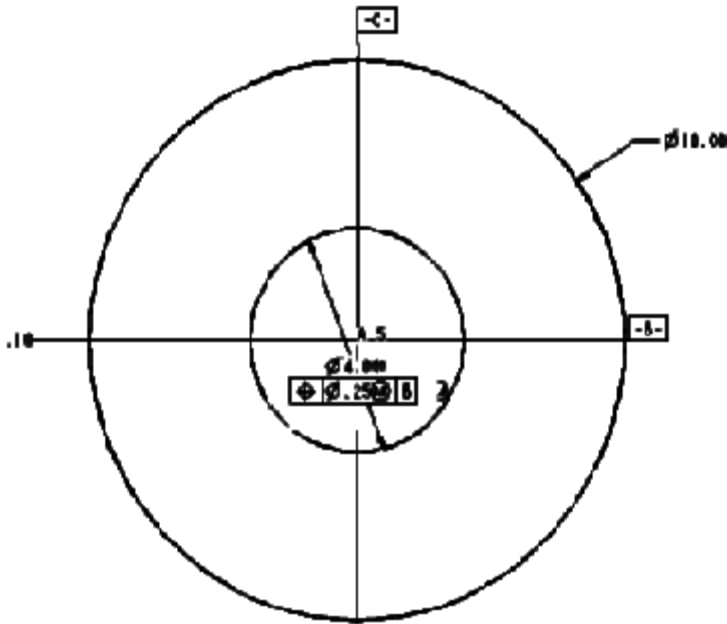
Example: Creating a Position GTOL

Use the characteristics and values given in the following table to create a sample geometric tolerance.

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

1. Click **Insert > Annotations > Geometric Tolerance**. The **GEOM TOL** menu appears.
2. Click **Specify Tol**. The **Geometric Tolerance** dialog box opens.
3. Click the **Position** button ().
4. Specify the type of entity to which the tolerance applies. Select **Feature** from the **Reference Type** list; then click **Select Entity** and select the hole in the model.
5. Select **Dimension** from the **Placement Type** list, and then click **Place GTOL**. Select the dimension of the hole. The GTOL appears in the model below the dimension.
6. Click the **Datum Refs** tab. Click **Primary**; then select **B** and **RFS** (no symbol) from the **Basic** lists.
7. Click the **Tol Value** tab and type 0.25 in the **Overall Tolerance** box. Select **MMC** from the **Material Condition** list.
8. Click the **Symbols** tab. Select the **Diameter Symbol** check box to indicate that the tolerance value refers to a diameter zone; then select **None** as the projected tolerance zone. The completed GTOL appears.
9. Click **OK** to close the dialog box and save the changes.

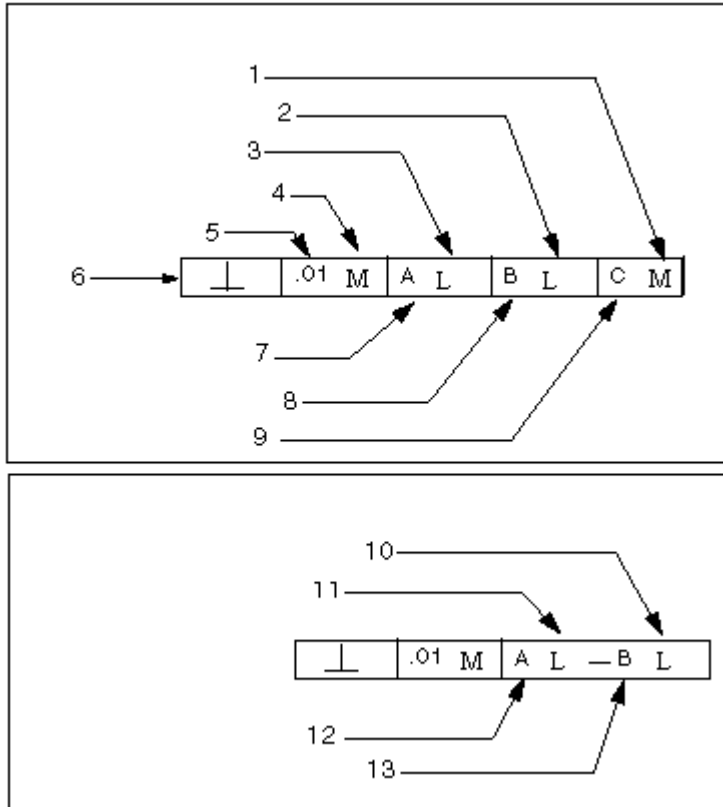
The GTOL appears as shown:



Example: Geometric Tolerance Layout

Pro/ENGINEER specifies a GTOL for an individual feature by means of a feature control frame (a rectangle) divided into compartments containing the GTOL symbol followed by the tolerance value. Where applicable, it also follows the tolerance with a material condition symbol.

If a GTOL is related to a datum, it places the reference datum name in a compartment following the tolerance value. Where applicable, it follows the datum reference letter with a material condition symbol.



1. Matl Cond, Tertiary
2. Matl Cond, Secondary
3. Matl Cond, Primary
4. Matl Cond, Tolerance
5. Value
6. Type
7. Datum Ref, Primary
8. Datum Ref, Secondary
9. Datum Ref, Tertiary
10. Matl Cond, Primary, Compound
11. Matl Cond, Primary, Basic
12. Datum Ref, Primary, Basic
13. Datum Ref, Primary, Compound

Modifying GTOL Values

To Modify the Value of an Individual Material Condition

1. Right-click on a geometric tolerance and select **Properties** from the shortcut menu.
2. In the **Geometric Tolerance** dialog box, click the **Datum Refs** tab.
3. Click **Primary**, **Secondary**, or **Tertiary** (whichever is appropriate) in the **Datum References** box.
4. For a basic datum reference, select a material condition from the **Basic** list. For a compound datum reference, select a material condition from the **Compound** list. The system updates the GTOL on the screen.
5. Click **OK** to close the dialog box and save the changes.

To Modify a Reference Datum to a GTOL

1. Do one of the following:
 - Add a new reference datum by converting an existing simple reference datum into a compound reference feature. Add a secondary or tertiary reference datum by clicking either **Secondary** or **Tertiary**. Select a reference datum from the **Compound** list. Specify a material condition, if needed.
 - Replace a reference datum 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 reference datum. Click **Primary**, **Secondary**, or **Tertiary** in the **Datum References** box (whichever is appropriate); then select **None** from the **Basic** or **Compound** list.

The system updates the GTOL on the screen.

2. Click **OK** to close the dialog box and save the changes.

To Modify a GTOL Value on the Screen

1. Right-click on a geometric tolerance and select **Value** from the shortcut menu.
2. Type a value. The system updates the display.

To Modify GTOL Values Using the Geometric Tolerance Dialog Box

1. Right-click on a geometric tolerance and select **Properties** from the shortcut menu.
2. In the **Geometric Tolerance** dialog box, click the **Tol Value** tab.

3. Modify the overall value or the **Per Unit Tolerance Values**.
4. To modify the Zone Height Value, select **Symbols**.
5. Select the **Zone Height** check box and type a value. The system updates the display to reflect the changes.

To Move a Geometric Tolerance

1. Select the geometric tolerance in the Model Tree or graphics window and right-click. The shortcut menu appears.

Note:

- To view the geometric tolerance in the Model Tree, click **Settings > Tree Filters > Annotations**.
 - To select the geometric tolerance in the graphics window, select **Annotation** from the Filter list on the status bar.
2. Click **Move** on the shortcut menu. Alternatively, click **Move** in the **Geometric Tolerance** dialog box that opens when you click **Properties** on the shortcut menu.
 3. Move the pointer to the new location.
 4. Click to place the geometric tolerance in the new location or middle-click to cancel the move operation.

Working with Surface Finish Tolerances

About Surface Finish Tolerances

Surface finish is a measure of the deviation of a part surface from its normal value. In Pro/ENGINEER, you can specify the roughness of the surface in micrometers or microinches in the range of 0.001 through 2000. Surface finish can be associated with any model surface. As geometric tolerances, surface finish symbols represent annotations and do not affect model geometry. Surface finish symbols can be included in Annotation features.

A surface can have only one surface finish tolerance associated with it.

You can add surface finish symbols to a model using standard surface finish symbols available in Detailed Drawings, or you can create your own surface finish symbols. If you have a Detailed Drawings license, you can define surface finish symbols on a part. These symbols will appear in the associated drawings of the part, and vice versa.

You can also gain access to a set of standard surface finish symbols that you can attach to edges, and dimensions. In Assembly mode, you can create surface finish symbols on surfaces created by assembly features (holes, cuts, and slots)

style="x-condition: include_for_toyota;">

To Create a Surface Finish Tolerance

1. Define the active annotation orientation before creating the first annotation in any session by using the **ACTIVE ANNOTATION ORIENTATION** dialog box that opens when you click **View > Annotation Orientation**.
2. Click **Insert > Annotations > Surface Finish**. The **SURF FIN** menu appears with **Create** selected by default. The **Open** dialog box opens and allows you to retrieve the surface finish symbol definition. The name of this symbol definition is displayed in the **Symbol name** box in the **Surface Finish** dialog box that opens subsequently.

Note: If a surface finish symbol already exists in the model, the **Open** dialog box does not open and the **Surface Finish** dialog box opens directly.

3. In the **Symbol name** box, select the name of the symbol from the list. The list shows symbols that are associated with the model, if any. If the symbol you want has not been added to the model, browse to locate it in the symbol library, or click **New** to create a new symbol definition.
4. Select reference surfaces, that is, surfaces that you want the Surface Finish symbol to represent. You can select only single surfaces as reference surfaces. The names of the selected references are displayed in the **References** collector.

Note:

- While including a surface finish annotation in an Annotation Feature, you can select single surfaces, intent surfaces, excluded surfaces, all surfaces of a quilt, seed and boundary surfaces, loop surfaces, or a combination of any of these as reference surfaces. You can use the **Surface Sets** dialog box that opens when you click **Details** under **References** to create, review, or edit the surface sets.
 - When creating standalone Surface Finish Annotations or Surface Finish Annotation Elements in an assembly, you can select surfaces that were created by assembly features, specifically, holes, cuts, or slots only.
5. Under **Placement** Pro/ENGINEER displays the default placement type as **Normal to Entity**. You can select a different type of placement from the **Type** list and define the placement of the symbol.

Note:

- **Normal to Entity** places the surface finish annotation normal to the selected entity and in an annotation plane. However, if the annotation plane is not normal to the selected entity, then the surface finish annotation too is not normal to the entity.
 - You can change the default placement type for surface finishes by setting the `default_placement_surfacefinish` configuration option.
6. If you want to move the symbol, click **Move**. Move the pointer to the new location and click to place the symbol at the new location or middle-click to cancel the move operation.

7. Under **Properties**, you can define the height, angle, and color of the symbol.
8. Use the **Grouping** tab to specify which groups will show in the instance.
9. Use the **Variable Text** tab to select the content of notes appearing with the symbol.
10. Click **OK**.

style="x-condition: include_for_toyota;">

To Place Surface Finish Symbols

While creating a Surface Finish Annotation, after selecting reference surfaces, that is, surfaces that you want the Surface Finish symbol to represent, you must specify the placement of the Surface Finish Annotation symbol.

1. In the **Surface Finish** dialog box, select the placement type from the **Type** list in the **Placement** box.
 - **With Leaders**—Select to attach the symbol to model surfaces using leaders.
 - **On Entity**—Select to attach the symbol directly to a surface of the model.
 - **Normal to Entity**—Select to attach the symbol directly to a surface of the model such that it is normal to the surface.
 - **Free**—Select to place the symbol without attaching it to model geometry.
2. For **With Leaders** attachment, select one or several locations on model surfaces to attach the leaders by holding down the CTRL key and selecting them. Middle-click at the required location in the graphics window to place the symbol. A leader is generated for each selected location. For **On Entity** and **Normal to Entity** attachments, select a point on a surface of the model. For **Free**, select a location for the symbol anywhere in the graphics window.

Note: When you select **With Leaders**, **On Entity**, or **Normal to Entity** attachments, you can select points on those surfaces that have been selected as references.

3. If you want to move the symbol, click **Move**. Move the pointer to the new location and click to place the symbol in the new location or middle-click to cancel the move operation.
4. Click **OK** when finished.

To Modify the Value of a Surface Finish Tolerance

1. Select the surface finish tolerance on the screen or in the Model Tree. You can select multiple tolerances.
2. Right-click on one of the selected names and select **Value** from the shortcut menu.

3. Type the new value. The system updates the surface finish value for all the selected tolerances.

To Modify the Name of a Surface Finish Tolerance

1. Click **Insert > Annotations > Surface Finish**. The **SURF FIN** menu appears.
2. Click **Modify Name**.
3. Select the surface finish tolerance on the screen or in the Model Tree.
4. Type the new name in the message area and press ENTER. The system updates the name of the surface finish tolerance.

To Move a Surface Finish Tolerance Symbol

1. Select the surface finish tolerance symbol in the Model Tree or graphics window and right-click. The shortcut menu appears.

Note:

- To view the symbols in the Model Tree click **Settings > Tree Filters > Annotations**.
 - To select the symbol in the graphics window, select **Annotation** from the Filter list on the status bar.
2. Click **Move** on the shortcut menu.
 3. Move the pointer to the new location. An outline of the symbol along with the leader line moves with the pointer.
 4. Click to place the symbol in the new location or middle-click to cancel the move operation.

To Delete a Surface Finish Tolerance

1. Click **Insert > Annotations > Surface Finish**. The **SURF FIN** menu appears.
2. Click **Delete**.
3. Select the surface finish tolerance you want to delete. The tolerance symbol and value are deleted from the surface.

Working with Model Properties

About Setting Up Model Properties

In **Part** mode, use the **PART SETUP (Edit > Setup)** menu to define basic database input values like material types, part accuracies and units of measure.

- Edit feature or datum names
- Create parameters and assign them to database objects

- Set measurement units for the current session
- Create and edit notes
- Define mass properties
- Define dimension boundaries
- Define Geometric Tolerances
- Define and use three-dimensional (model) grids
- Apply shrinkage values to the part, to be used in a mold design or casting process

Renaming Objects

About Renaming Objects

You can use the **Name** command to assign names to components, features, and geometry. Naming these objects makes it easier to select them. For example, you could name an alignment hole for an assembly `align_hole_1`.

Note:

- Names can contain up to 31 characters. They cannot include spaces.
- If a datum point array feature contains a single point only, the name of the point and the name of the feature are identical.

You can rename Family Table instances using the Family Table functionality. In the **Family Table**, select the instance you want to rename and type in a new name.

To Rename an Object

Use the **Name** command to assign names to components, features, and geometry.

1. Click **Edit > Set Up > Name**. The **NAME SETUP** menu appears.
2. On the **NAME SETUP** menu, click one of the following commands:
 - **Component**—Name or rename components (Assembly mode only).
 - **Feature**—Name or rename all features, including datum points.
 - **Detail Item**—Name a surface finish symbol or the primary value of a geometric tolerance.
 - **Other**—Name datum axes that were created as part of another feature (such as a hole), curve segments making up composite curves, datum points, surface features, and feature edges.
3. Select the feature or item to be modified. A text box appears below the Model window.
4. Type the new name.

Units

About Measurement Units

Each model has a basic system of metric and nonmetric units to ensure that all material properties of that model are consistently measured and defined. All Pro/ENGINEER models have length, mass/force, time, and temperature units defined.

Pro/ENGINEER comes with some predefined systems of units, one of which is a default system of units. You can change the assigned system of units, and you can define your own units and systems of units (called custom units and custom systems of units). You cannot change the predefined systems of units.

Use the **Units** command to set, create, change, review, or delete a system of units or a custom unit for your model.

When you use the **Units** command, the **Units Manager** dialog box opens. This dialog box lists the predefined systems of units and any already defined custom systems of units. Using this dialog box, you can create new custom units and systems of units. A red arrow indicates the current system of units for the model.

Use custom units only if your model does not contain standard SI or British units or if your Material file (which you set up by using the **Material** command under **Edit > Setup**) contains units that cannot be derived from the system of units or both. Pro/ENGINEER uses the definition of a custom unit in interpreting material properties. You can also use custom units to create a new system of units.

Note: Although the configuration options `pro_unit_length` and `pro_unit_mass` are still available, they are no longer recommended for use. It is recommended that you create a template for models that has a fully defined system of units. Use **File > New**, clear **Use default template** in the **New** dialog box, and then use the **New File Options** dialog box to create a template or modify one of the PTC standard templates. In any case, you must make sure that you have a defined set of units before you design your model.

To Select a Measurement Units System

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Units**. The **Units Manager** dialog box opens to the **Systems of Units** tab. In the system list, a red arrow points to your current system of units.
3. Select a system of units. A description of the metric and non-metric measurement units appears under **Description**. The **PTC** icon (blue stripes) appears in front of the descriptions of the system-provided systems of units. You can use any of the system-provided systems of units or create a custom system of units.
4. Click **Set**. If the units require no scaling, the model converts to the new setting. If the units have changed, a **Warning** dialog box opens offering the following options:

- **Convert dimensions (for example 1" becomes 25.4mm)**—Scales the dimension values so that the model is the same size. For example, if you change a cube that is 1 inch square to millimeters, it becomes 25.4 mm square.

For this option:

User-defined parameters are not scaled. You must update these parameters yourself.

Angular dimensions are not scaled. For example, a 30 degree angle remains at 30 degrees.

The model is regenerated.

- **Interpret dimensions (for example 1" becomes 1mm)**—Does not change dimension values. Use this option if you want to get mass properties and then realize that your units are in inches. For example, if you change a cube that is 1 inch square to millimeters, it becomes 1 mm square.

After you click **Set**, the red arrow points to your newly chosen system of units.

5. Click **Close** to close the **Units Manager** dialog box.

To Create Custom Units

You create a custom measurement unit in the same way you create a custom system of units, by using the **Units Manager** dialog box.

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Units**. The **Units Manager** dialog box opens to the **Systems of Units** tab.
3. Click the **Units** tab. All system-defined and custom units are listed. A description of the selected unit appears under **Description**.
4. Click the **New** button. The **Unit Definition** dialog box opens.
5. Select a physical dimension from the drop-down list provided.
6. Under **Unit Definition**, do the following:
 - Under **Name**, type a name for the new unit or accept the default name.
 - Enter a scale, and then select a reference unit from the drop-down list. The default unit is **in** (inches).
7. Click **OK** to close the **Unit Definition** dialog box, and then click **Close** to close the **Units Manager** dialog box.

Note:

Use the **Copy** button to create a copy of the selected unit with a different name. You can also rename a unit by using this procedure.

You cannot edit or delete system-provided units. System-provided units are indicated by the **PTC** icon under **Description**.

To Delete a Custom Unit

You can delete a *custom* unit that you previously created. You *cannot* delete a predefined unit. Use caution when deleting units; you cannot undo the deletion.

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Units**. The **Units Manager** dialog box opens to the **Systems of Units** tab.
3. Click the **Units** tab. A description of the selected unit appears under **Description**.
4. Select a custom unit that you want to delete, and then click **Delete**. A warning message appears and lists the unit you have selected for deletion.
5. Click **Confirm** to delete the unit.

To Create a Custom Units System

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Units**. The **Units Manager** dialog box opens to the **Systems of Units** tab, which lists all available custom and system-defined units systems. A description of the selected units system appears under **Description**.
3. Click the **New** button. The **System of Units Definition** dialog box opens, showing the units for the predefined system.
4. Under **Name**, type a name for the new unit.
5. Under **Units**, select the desired types of units in the appropriate drop-down lists. You can select either Mass or Force units by clicking the **Mass** or **Force** button.
6. Click **OK** to return to the **Units Manager** dialog box. The new system appears in the list.
7. Select the new units system and click **Set** to assign your custom system of units to your model.

Note:

Use the **Copy** button to create a copy of the selected units system with a different name.

You cannot edit or delete the systems of units provided by the system. System-provided units systems are indicated by the **PTC** icon under **Description**.

To Edit a Custom Units System

To redefine a units system that you previously created,

1. In Part or Assembly mode, click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.

2. Click **Units**. The **Units Manager** dialog box opens to the **Systems of Units** tab.
3. Double-click the custom units system you want to edit, or select the system and then click **Edit**. The **System of Units Definition** dialog box opens.
4. Redefine the units system as necessary, and then click **OK** to close the dialog box.
5. Click **Close** to close the **Units Manager** dialog box.

To Delete a Custom Units System

You can delete a *custom* system of units that you previously created. You *cannot* delete a system-provided system of units. Use caution when deleting units systems; you cannot undo the deletion. Before you delete the units system, make sure it is not the set system for the current part or assembly.

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Units**. The **Units Manager** dialog box opens to the **Systems of Units** tab, which lists all available custom and system-defined units systems. A description of the selected units system appears under **Description**.
3. Select a custom units system that you want to delete, and then click **Delete**. A warning message appears and lists the units system you have selected for deletion.
4. Click **Confirm** to delete the units system.

About Modifying Measurement Units

You can edit, rename, delete, or obtain information about existing measurement units and unit systems using the **Units Manager** dialog box. To open the dialog box, click **Units** from the **PART SETUP** or **ASSEM SETUP** menu that appears when you click **Edit > Setup** in Part or Assembly mode. You modify measurement unit systems using the **Systems of Units** tab, and you modify individual units using the **Units** tab.

The following modification options are available on the **Units Manager** dialog box. For specific procedures, click *See Also*.

- **Copy**—You can make a copy of an existing unit or system of units, or you can use this command to rename a unit or a custom system of units by creating a copy with a different name.
- **Edit**—You can edit a custom unit or a custom system of units that you previously created. You *cannot* edit an existing system or unit provided by Pro/ENGINEER, such as Meter Kilogram Second (MKS) or cm.
- **Delete**—You can delete a *custom* unit or system of units and revert to a Pro/ENGINEER-provided unit or system. You *cannot* delete Pro/ENGINEER-defined units or unit systems.
- **Info**—This command is available for unit *systems* only, not for individual units. You can display information about the basic units and dimensions of the current

system of units and about the derived units that follow from this system. You can save, copy, or edit this information by using the options in the Information window.

To Create a Copy of a Measurement Units System

You can create a copy of an existing units system, including custom systems you previously created, or Pro/ENGINEER-defined unit systems. You can also use this method as a way of renaming an existing units system.

1. In Part or Assembly mode, click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Units**. The **Units Manager** dialog box opens to the **Systems of Units** tab.
3. Select the units system you want to copy, and then click **Copy**. The **Copy System of Units** dialog box opens.
4. Under **To new Unit**, use the default name for the new system of units (`unit_system1`, `unit_system2`, and so on), or type a new name.
5. Click **OK**. The new units system is created and appears in the list.
6. To edit the new units system, click **Edit**, and use the **System of Units Definition** dialog box to redefine the units system.

To Create a Copy of an Individual Unit

You can create a copy of any existing unit, including both custom units you previously created and system-provided units. You can also use this method as a way of renaming an existing unit.

1. In Part or Assembly mode, click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Units**. The **Units Manager** dialog box opens to the **Systems of Units** tab.
3. Click the **Units** tab.
4. Select the unit you want to copy, and then click **Copy**. The **Copy Unit** dialog box opens.
5. Under **To new Unit**, use the default name provided for the new unit (`unit1`, `unit2`, and so on), or type a new name.
6. Click **OK**. The new unit is created and appears in the list.

To edit the new units system,

Double-click the new unit in the list, or select the new unit and then click **Edit**. Then use the **Unit Definition** dialog box to redefine the unit.

Tip: Specifying Metric and Nonmetric Units

When specifying your system of units, keep in mind the following guidelines:

- You can specify units in Part or Assembly mode. If you are working in Assembly mode, the units for all the parts must be the same as those for the assembly.
- When you enter numerical data for quantities with physical dimensions, Pro/ENGINEER interprets the data as having units consistent with your selected system of units.
- Modifying the units can invalidate your relations, as they are not scaled along with the model. Other non-parametric features, such as cosmetic, or IGES models are also not scaled with the change of units.
- A part can have only one set of units.
- All parts default to inches and pounds (unless otherwise set by the configuration file options `pro_unit_length` and `pro_unit_mass`).

Tip: Rules for Modifying Units

- Modifying the units can invalidate your relations, as they are not scaled along with the model. Other non-parametric features, such as cosmetic, or IGES models are also not scaled with the change of units.
- A part can have only one set of units.
- All parts default to inches and pounds (unless otherwise set by the configuration file options `pro_unit_length` and `pro_unit_mass`).
- The initial units for an assembly are those of its base component. If, however, the units of the base component have been changed, the assembly units do not automatically change. You must re-specify the units of the assembly.
- You cannot change the units of an assembly that contains assembly features which intersect a part.
- User-defined parameters have no units; therefore, include appropriate conversion factors in relations.
- Cross-sections have no units.
- If you delete a custom measurement units system, make sure the system is not the selected system for the current part.

To Get Information About Measurement Units

You can obtain information about your system of metric and nonmetric units.

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Units**. The **Units Manager** dialog box opens.
3. Select a system of units. A brief description appears under **Description**.
4. For more information, click the **Info** button. An Information window opens and displays the following information:
 - Units Information—The current system of units at the top of the window.

- Basic Quantities—The basic units and dimensions of your system of units.
- Gravity—Displayed in a separate section.
- Derived Quantities—The derived units that follow from your system of units.

You can save the information to a file by using the **File** menu in the Information window.

5. Click **Close** to close the Information window, and then click **Close** to close the **Units Manager** dialog box.

Scaling a Model

To Scale a Model

You can scale the dimension values of your model by a factor you specify.

1. If you are working in Part or Assembly, click **Edit > Scale Model**, otherwise, click **Modify > Scale Model** from the PART or ASSEMBLY menu. Pro/ENGINEER prompts you to enter a scale.
2. Enter a scale in the message window. The default is 1.0.

Pro/ENGINEER scales the length dimensions of your model by using the factor you specified. If the scale is modified, the model is regenerated.

Notes:

- When you scale your model, the model units are not changed.
- Imported geometry cannot be scaled.

Materials

About Material Parameters

You can create and modify material data using the **Materials** and the **Materials Definition** dialog boxes. The material data can be saved to the model or saved to material data files that have a `.mt1` extension. The `.mt1` files are stored in the material library directory. Each file contains the name of the material and a set of material parameters that are identified by an ID and units information.

User-defined material parameters are parameters that you define by using the **User Defined** tab in the **Materials Definition** dialog box. All other parameter values that you define through the **Materials Definition** dialog box are stored as reserved parameters. If you define a user-defined parameter that has the same name as a reserved parameter, Pro/ENGINEER issues a warning.

Note the following points about material parameters:

- You can use the material parameter values in relations.

- You can assign materials to family table instances by adding `PTC_MATERIAL_NAME` parameter to the family table.
- You cannot designate material parameters except the `PTC_MATERIAL_NAME` parameter.
- When you save a material, parameters with default values are saved only if you have changed the default value. Additionally, only parameters that are valid for a specified material type are saved, for example, the Orthotropic Young's Modulus is not saved for an isotropic material definition.

The reserved material parameters that you can specify are:


`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_FATIGUE_MATERIAL_TYPE`
`PTC_FATIGUE_MATERIAL_FINISH`
`PTC_FATIGUE_STRNGTH_REDUCT_FCTR`
`PTC_TSAI_WU_INTERACT_TERM_F12`
`PTC_TENSILE_YIELD_STRESS`
`PTC_TENSILE_ULTIMATE_STRESS_ST1`
`PTC_TENSILE_ULTIMATE_STRESS_ST2`
`PTC_COMPR_ULTIMATE_STRESS_SC1`
`PTC_COMPR_ULTIMATE_STRESS_SC2`
`PTC_POISSON_RATIO_NU21`
`PTC_POISSON_RATIO_NU31`
`PTC_POISSON_RATIO_NU32`
`PTC_YOUNG_MODULUS_E1`
`PTC_YOUNG_MODULUS_E2`
`PTC_YOUNG_MODULUS_E3`
`PTC_SHEAR_MODULUS_G12`
`PTC_SHEAR_MODULUS_G13`
`PTC_SHEAR_MODULUS_G23`
`PTC_THERMAL_EXPANSION_COEF_A1`
`PTC_THERMAL_EXPANSION_COEF_A2`
`PTC_THERMAL_EXPANSION_COEF_A3`
`PTC_THERMAL_CONDUCTIVITY_K1`
`PTC_THERMAL_CONDUCTIVITY_K2`
`PTC_THERMAL_CONDUCTIVITY_K3`
`PTC_HARDNESS_TYPE`

PTC_XHATCH_FILE
 PTC_FATIGUE_TYPE
 PTC_FAILURE_CRITERION_TYPE
 PTC_MATERIAL_DESCRIPTION
 PTC_MATERIAL_NAME

To Add and Assign a Material to a Part

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Material**. The **Materials** dialog box opens.
 - The **Look In** box displays the directory located in the path specified by the `pro_material_dir` configuration option. If you have not set the `pro_material_dir` configuration option, then the **Look In** box displays the default material directory, that is, `<Pro/ENGINEER loadpoint>\text\materials-library`.
 - The **Materials in Library** list displays the contents of the directory in the **Look in** box.
 - The **Materials in Model** list displays the list of materials present in the model.
3. If required, use **Look In** to browse to the directory that contains the required material files. The names of the materials in the directory are displayed in **Materials in Library** list. Material files have a `.mtl` and `.mat` extension. The files with the `.mat` extension are files from release prior to Pro/ENGINEER Wildfire 3.0.
4. To add materials to a model, move the required materials from the **Materials in Library** list to the **Materials in Model** list.

Note: If a model does not contain any material, then the first material that you add to the model is assigned to the model by Pro/ENGINEER.

5. To assign a material, select the material that you want to assign to the model from the **Materials in Model** list and click  or click **File > Assign**. The assigned material is denoted by a red arrow that precedes the name of the material in the **Materials in Model** list.
 - You can assign only one material to a model at a time even if a model has multiple materials.
 - You cannot assign a material to a model in assembly mode.
 - To remove the assignment of a material from the model, select the assigned material from the **Materials in Model** list and click **File > Unassign**.

To Create or Copy a Material

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.

2. Click **Material**. The **Materials** dialog box opens.
3. Click **File > New**. The **Materials Definition** dialog box opens.
4. Type a name for the material in the **Name** box.
5. Type a description for the material in the **Description** box.
6. In the **Density** box, type a value for the density of the material and select the unit of measurement from the adjacent list.

Note: If you select a unit of measurement that is different from the assigned system of units for the model, then the **Changing Parameter units** dialog box opens and allows you to convert or interpret the parameter value. **Convert Value (for example 1 "becomes 25.4mm)** is the default.

7. Use the following tabs to define the properties of the material:

Structural—Defines the structural properties for the material of a model.

Thermal—Defines thermal properties for a material of a model.

Miscellaneous—Defines sheet metal properties, surface properties, and cross-hatching.

Appearance—Defines appearances.

User Defined—Defines parameter definitions and adds them to the materials database. For more information, click the *See Also* link To Create a Parameter and start at step 4.

8. To save the material to the library, perform the following steps.
 - a. Click **SaveTo Library**. The **Save a Copy** dialog box opens.
 - b. Browse to the library folder and click **OK**. If you have added or edited appearances then the **Save** dialog box opens.
 - c. Click **OK**. By default, the appearances are saved in a file named `global.dmt` in the same location where the material file is saved or in the location defined by the `global_appearance_file` configuration option.
9. To save a material to the model, click **Save To Model**. The material is saved to the model and appears in the **Materials** dialog box in the **Materials in Model** column.
10. Click **OK** in the **Materials** dialog box.

Note:

- To copy an existing material in a model, select the material that you want to copy in the **Materials in Model** list and click **Edit > Copy** in the **Materials** dialog box. The copy appears in the **Materials in Model** list as `Copy_of_<name>` while subsequent copies of the same material appear as `Copy1_of_<name>`, `Copy2_of_<name>` and so on.
- You can define material properties as a function of temperature using the **Functions** dialog box that opens when you click **Function** on the shortcut menu.

The shortcut menu appears when you right-click in the box next to **Poisson's Ratio**, **Young's Modulus**, or **Coeff. of Thermal Expansion** in the **Structural** tabbed page. After you select or define a function and click **OK** in the **Function** dialog box, the function name appears in the box next to the material property.

Similarly, you can also assign a parameter value to define a material property by using the **Select Parameter** dialog box that opens when you click **Parameter** on the shortcut menu. The shortcut menu appears when you right-click in the box next to the material property in the **Structural** or **Thermal** tabbed pages. Select a parameter from the dialog box. After you click **Insert Selected**, the parameter name appears in the box next to the material property.

About Conversion of Material Data

The material data in releases before Pro/ENGINEER Wildfire 3.0 is stored in `mmat1.lib` files or in files with the `.mat` extension.

You can convert a `mmat1.lib` file created in Mechanica before Pro/ENGINEER Wildfire 3.0 to `.mtl` files by clicking **File > Convert pre-Wildfire 3.0 lib** in the **Materials** dialog box. You can edit existing materials definitions, that is, the `.mat` files in Pro/ENGINEER. These files are saved to the `.mtl` format when you save the material definition.


If the following parameters are present in existing material definitions, then Pro/ENGINEER adds the new material as user-defined parameters.

- SHEAR_MODULUS
- THERM_EXPANSION_REF_TEMPERATURE
- STRUCTURAL_DAMPING_COEFFICIENT
- EMMISIVITY
- STRESS_LIMIT_FOR_TENSION
- STRESS_LIMIT_FOR_COMPRESSION
- STRESS_LIMIT_FOR_SHEAR
- Cost (in Mechanica)

Conversion of Material in the Model

Before editing a model that contains a material created in Mechanica before Pro/ENGINEER Wildfire 3.0 and if this material is a non-isotropic material or a material with functions, parameters, fatigue properties, or failure criteria, then you must convert material data in the model to the latest format. If you edit such a model or material in Pro/ENGINEER without converting the material to the latest format, you may lose your data associated with the material.

To Edit a Material


1. If you are editing a model that contains a material created in Mechanica before Pro/ENGINEER Wildfire 3.0 and if this material is a non-isotropic material or a material with functions, parameters, fatigue properties, or failure criteria, then you must convert material data in the model to the latest format in Mechanica before you begin editing.
2. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
3. Click **Material**. The **Materials** dialog box opens.
4. In the **Materials in Library** or **Materials in Model** list, select the material that you want to edit.
5. Click  or click **Edit > Properties**. The **Material Definition** dialog box opens.
6. Modify the name, description, and other properties of the material.
7. If you have selected a material in the **Materials in Library** list, then **Save to Library** becomes available. To save the material to the library, perform the following steps.
 - a. Click **SaveTo Library**. The **Save a Copy** dialog box opens.
 - b. Browse to the material library directory and click **OK**. If you have added or edited appearances, then the **Save** dialog box opens.
 - c. Click **OK**. By default, the appearances are saved in a file named `global.dmt` in the same location where the material file is saved or in the location defined by the `global_appearance_file` configuration option. You are taken back to the **Materials** dialog box.
8. If you have selected a material in the **Materials in Model** list, click **OK** in the **Material Definition** dialog box. The **Material Definition** dialog box closes and you are taken back to the **Materials** dialog box.

To View Material Information

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Material**. The **Materials** dialog box opens.
3. In the **Materials in Model** list, select the material whose information you want to view.
4. Click **Show > Info**. The material information is displayed in the Pro/ENGINEER browser.


Note: To view only the description of a material, select the material in the **Materials in Library** or **Materials in Model** list and click **Show > Description**. The description is displayed in the **Description** area in the **Materials** dialog box.

To Save a Material From a Model to a Library

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Material**. The **Materials** dialog box opens. The **Materials in Model** list displays the list of materials present in the model.
3. Select the material that you want to save to the library directory on disk from the **Materials in Model** list.
4. Click  or click **Edit > Save a Copy**. The **Save a Copy** dialog box opens. You can browse to the material library directory and click **OK**. If you have added or edited appearances, then the **Save** dialog box opens.
5. Click **OK**. By default, the appearances are saved in a file named `global.dmt` in the same location where the material file is saved or in the location defined by the `global_appearance_file` configuration option. You are taken back to the **Materials** dialog box.

Note: The units in the material file that you saved to the disk are the same as those used in the model when the material file was created or last updated.

To Remove a Material from the Model

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Material**. The **Materials** dialog box opens.
3. In the **Materials in Model** list, select the material that you want to delete.
4. Click  or click **Edit > Delete** in the **Materials** dialog box. Pro/ENGINEER prompts you for confirmation.
5. Click **OK**. The material is removed from the **Materials in Model** list. The material is also removed from the model database.

Note: If a material is referenced by other entities, for example, in Mechanica, you must first remove the dependencies before deleting the material.

To Define Sheetmetal, Surface, and Detailing Properties

1. Click the **Miscellaneous** tab in the **Materials Definition** dialog box when creating a material. The **Miscellaneous** tabbed page opens displaying a default value of 0.5 for **Initial Bend Y-Factor**.

Note: Pro/ENGINEER checks for errors in the material properties that you specify whenever you save a material, add a material from the library to the model, and at run time in both Mechanica and FEM modes.

2. Under **Sheetmetal Properties**, type a new value for **Initial Bend Y-Factor** and **Bend Table**, if required. The valid value for Initial Bend Y-Factor is a positive real number (includes 0.0) and for bend table is a string with a maximum length of 31 characters.

3. Under **Surface Properties**, type values for **Hardness**, **Hardness Type**, and **Condition**, if required. The valid value for hardness is a positive real number (includes 0.0) and for hardness type and condition, a string with a maximum length of 31 characters.

Note: You can choose not to specify values under **Surface Properties** blank. If you specify a value for one of the surface properties, then you must specify values for all the surface properties.

4. In the **Cross Hatching** box type the name of a file that has a `.xch` extension. This file type contains the cross hatching pattern that you want to apply to cross sections. Type the name without the `.xch` extension and limit it to 31 characters.

Note:

- Pro/ENGINEER searches for the specified file with `.xch` extension in the path defined by the `pro_crosshatch_dir` configuration option, the working directory, paths specified by the `search_path` configuration options, and the default crosshatch directory, that is, `<loadpoint>/text/crosshatch`. If the specified file cannot be located, then the default cross hatch pattern is used.
- When creating a new material or during automatic conversion of a material that existed before Pro/ENGINEER Wildfire 3.0, by default, the **Cross Hatching** box displays a value that corresponds with the name of the material.
- Cross sections that were created before a material is assigned to a model are not updated after you assign a material with a different cross hatching pattern to the model or when you change the material assigned to a model.

To Add and Assign an Appearance to a Material

1. Click the **Appearance** tab in the **Materials Definition** dialog box when creating a new material.
2. Click **Chooser**. The **Appearances** dialog box and the **Material Appearance Editor** dialog box open. The appearances in the `global.dmt` file are listed in the **Available Appearances** list in the **Appearances** dialog box.
3. To filter the list of appearances in the **Available Appearances** list by a specified name, type the name in the **Apply Filter** box and click **Apply Filter**.
4. Select the appearances that you want to add to the material in the **Available Appearances** list and move them to the **Visible Appearances** list.
5. Click **OK**. Alternatively, click **Apply** to apply changes without closing the **Appearances** dialog box. The appearances listed in the **Visible Appearances** list are displayed in the palette in the **Material Appearance Editor** dialog box and the **Appearance** tabbed page.
6. To assign an appearance as the default, select the appearance in the **Appearance** tabbed page and click **Default**. A red arrow appears next to the appearance indicating that the appearance is the default.

Note: If the material to which you have assigned an appearance is the default material that is assigned to the part, then the appearance is automatically applied to the part in the graphics window.

However, if you have set the `mat_assign_appearance` configuration option to `no`, then the appearance is not applied to the model. The default for this option is `yes`.

To Create an Appearance for a Material

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Material**. The **Materials** dialog box opens.
3. Do one of the following:
 - If you are creating an appearance for a material that already exists, then select the material in the **Materials in Library** or **Materials in Model** list and click **Edit > Properties**. The **Material Definition** dialog box opens.
 - Click **File > New**. The **Materials Definition** dialog box opens.
4. Click the **Appearance** tab in the **Materials Definition** dialog box.
5. Click **New**. The **Material Appearance Editor** dialog box opens.
6. Create a new appearance. This appearance is added to the palette in the **Material Appearance Editor** dialog box and in the **Appearance** tabbed page of the **Materials Definition** dialog box.
7. Click **Close** in the **Material Appearance Editor** dialog box.
8. Click **OK** in the **Materials Definition** dialog box.

Note: To copy an appearance, select the appearance in the **Appearance** tabbed page and click **Copy**. A copy of the appearance is added to the palette in the **Material Appearance Editor** dialog box and in the **Appearance** tabbed page of the **Materials Definition** dialog box. The name of the copied appearance is incremented appropriately based on the name of the selected appearance.

Files Associated With Material Appearances

You can define appearances for a material using the **Appearance** tab in the **Material Definition** dialog box. The names of the material appearances that are associated with a material are stored in the material definition file, that is, the file with a `.mt1` extension. Information about material appearances is stored in the `global.dmt` file. The default location of the `global.dmt` file is `\<loadpoint>\graphic-library\appearances`. However, you can set the `global_appearance_file` configuration option to a path that points to a different `global.dmt` file. This file takes precedence over the `global.dmt` file in the default location.

Note: The `global.dmt` file is different from the `appearance.dmt` file that stores the information of the appearances created using the **Appearance Editor** that opens when you click **View > Color and Appearance**.

When transferring a model you must transfer the associated `global.dmt` file along with the model. The appearances information other than default is not available for all the materials saved with the model if the `global.dmt` file is missing.

To Edit an Appearance of a Material

1. Click **Edit > Set Up**. The **PART SETUP** or **ASSEM SETUP** menu appears.
2. Click **Material**. The **Materials** dialog box opens.
3. Select the material in the **Materials in Library** or **Materials in Model** list and click **Edit > Properties**. The **Material Definition** dialog box opens.
4. Click the **Appearances** tab and select the appearance that you want to modify.
5. Click **Edit**. The **Material Appearance Editor** dialog box opens displaying the selected appearance.
6. Modify the appearance.
7. If you have selected a material in the **Materials in Library** list, then **Save to Library** becomes available. To save the material to the library, perform the following steps.
 - a. Click **SaveTo Library**. The **Save a Copy** dialog box opens.
 - b. Browse to the material library directory and click **OK**. If you have added or edited appearances, then the **Save** dialog box also opens.
 - c. Click **OK**. By default, the appearances are saved in a file named `global.dmt` in the same location where the material file is saved or in the location defined by the `global_appearance_file` configuration option.
8. If you have selected a material in the **Materials in Model** list, click **OK** in the **Material Definition** dialog box. The **Material Definition** dialog box closes and you are taken back to the **Materials** dialog box.

Note: If you assign a material to a part by using the **Material Appearance Editor** dialog box, then you can edit the appearance of this material by using **From Model** in the **Assignment** container in the **Appearance Editor** dialog box.

About the Material Library

A material library is a directory that contains the material definition files, that is, the .mtl files. The **Look In** box in the **Materials** dialog box displays the directory located in the path specified by the `pro_material_dir` configuration option. If you have not set the `pro_material_dir` configuration option, then the **Look In** box displays the default material directory, that is, `<Pro/ENGINEER loadpoint>\text\materials-library`. The default material directory contains the standard materials available in Pro/ENGINEER.

Adding a material to the library and saving it saves the material to a material file bearing the same name as the material name in Pro/ENGINEER.

When you save a material, Pro/ENGINEER writes the value and the unit for each material quantity to the material file. If the unit is a user-defined unit, then the definition of the user-defined unit is also written to the material file.

When you add an existing material to a model, any user-defined units that are defined in the material file are also added to the model. However, note the following points:

- If the name of the user-defined unit in the material file does not exist in the model, then the user-defined unit is added to the model.
- If the user-defined unit in the material file and the unit in the model have the same name and definitions, then the user-defined unit is not added to the model again.
- If the user-defined unit in the material file and the unit in the model, have the same name but the definitions are different, then Pro/ENGINEER prompts you to specify a new name for the unit before adding it to the model.

Tip: Using a Material Library

When you finish defining the materials you use in everyday design, create a material library directory to which all users have access.

Set up the directory with read and execute privileges only, to help protect its contents. Then assign its path to the configuration file option `pro_material_dir`. The **Look In** box in the **Materials** dialog box displays the directory located in the path specified by the `pro_material_dir` configuration option. If you do not set the `pro_material_dir` configuration option, then the **Look In** box displays the default material directory, that is, `<Pro/ENGINEER loadpoint>\text\materials-library`.

To Specify or Change the Material Density of a Part

1. On the **PART SETUP** menu, click **Mass Props**. The **Setup Mass Properties** dialog box opens.
2. Enter the density data in the **Density** field. Press **OK**.

The density of the part changes to the new value and the system updates the `mp_density` value in relations. If a material has been assigned to the part and the value of the corresponding system parameter in relations, its density is

updated. Any values dependent on those parameters are updated after regeneration.

Note: The configuration option `mass_property_calculate` is set to `request` by default. Set it to `automatic` to automatically calculate density.

Specifying the Density of a Part

You can use the system parameter `mp_density` in a relation to specify the density of a part. This density value is used for calculating mass properties. If you change the material density in a part, the system updates the `mp_density` value in relations.

If `mp_density` is being driven by a relation and you change the relation, the density of the part is updated to the new `mp_density` value when the part is regenerated. In addition, if a material was assigned to the part at the time of regeneration, the material density is also updated to the new `mp_density` value.

If you change the density of the part using the **Density** field in the **Setup Mass Properties** dialog box, `mp_density` is updated immediately. If a material has been assigned to the part, the density of the material is also updated immediately.

If a material has been assigned to the part and you change the density of the assigned material using the **Material Definition** dialog box, the density of the part and `mp_density` are both updated immediately.

In both of these cases, any values that are dependent on `mp_density` are updated after regeneration.

Note: Set the configuration option `mass_property_calculate` to `automatic`.

About Adding Material Parameters to a Family Table

You can assign materials to family table instances by adding a parameter to the family table. Use family tables to control the assigned material by adding the `PTC_MATERIAL_NAME` parameter to a family table as an item. After the material parameter is added to the Family Table, you can select materials already in the model from the list that appears when you click in a cell in the `PTC_MATERIAL_NAME` column in the **Family Table** dialog box.

Example: Specifying Material Parameters through Relations

You can use material parameter values in relations.

For example, in a part:

- Assign a value to the parameter `HARDNESS` in the material with `MTRL_ID` 64. The material must be in the internal database of the part (but is not necessarily assigned to the part):

```
PTC_HARDNESS:MTRL_ID_64 = 55
```

- Assign a value to the parameter `POISSON_RATIO` in the assigned material:

```
PTC_POISSON_RATIO:MTRL_ID = 0.3
```

(where MTRL_ID is ID of the assigned material)

- Make a dimension dependent on the material parameter YOUNG_MODULUS:

```
d12 = PTC_YOUNG_MODULUS:MTRL_ID/1000
```

For example, in an assembly:

- Assign a value to the material parameter POISSON_RATIO in the component with Session ID 10:

```
PTC_POISSON_RATIO:MTRL_ID:10 = 0.30
```

- Make dimension d17 dependent on the material parameter HARDNESS in the component with session ID 21:

```
d17 = PTC_HARDNESS:MTRL_ID:21 * 40
```

If a material parameter is used as driving variable and you change it, the driven variable or dimension, likewise, is updated after regeneration.

Mass Property Parameters

About Working with Mass Property Parameters

Pro/ENGINEER can compute the mass properties of a part or assembly either based on the actual geometry of the object, or based on user-assigned parameter values. For example, when you create a simplified part for an assembly, you may want its mass properties to correspond to a fully detailed part. You can assign mass properties to a part or assembly either by assigning values to alternate mass parameters, or by using a mass properties file.

When working with the mass properties of a part or assembly, you can at any time generate a mass properties report based on either of the following sources:

- Geometry—Calculates mass properties based on model geometry.
- Geometry and parameters—Calculates mass properties based on user-assigned alternate parameter values. Where the alternate parameters are not specified, the system uses the parameter values derived from the model geometry.
- File—Calculates mass properties based on the parameter values specified in a mass properties file. The file has to contain all the parameter values. If the file is incomplete, the system does not generate a report.

When you generate a mass properties report, it is displayed in an information window and written to a file in the current working directory, named `objectname.m_p`, where `objectname` is the name of the part or assembly.

Reported and Alternate Parameters

The values generated in the mass properties report are stored in the model as its reported mass property parameters. You can view them by clicking **Tools >**

Parameters and selecting **Reported Mass Properties** from the list at the bottom of the **Parameters** dialog box.

You can not modify values of reported parameters, they are derived from the last generated mass property report. To change the reported parameter values, you can:

- Specify alternate mass property parameter values and generate a report using **Geometry and Parameters** as the source.
- Use a file to assign mass properties and generate a report using **File** as the source.

If you generate a report using **Geometry** as the source, the reported parameter values are based on the model geometry. Before you calculate model mass properties for the first time, all the reported parameter values are zero.

Alternate mass property parameters are also stored in the model. You can modify values of these parameters using either of the following methods:

- When generating the mass properties report, set the source to **Geometry and Parameters** and click **Edit**.
- Click **Tools > Parameters** and select **Alternate Mass Properties** from the list at the bottom of the **Parameters** dialog box. Type the parameter values in the appropriate boxes.
- Click **Tools > Relations** and add relations to the model to define the appropriate parameter values.

After you change the alternate parameter values using either of these methods, you have to make the changes reflect in the reported parameters. You can:

- Generate a mass properties report with **Geometry and Parameters** as the source.
- Make sure the `PRO_MP_SOURCE` parameter is set to `PARAMETERS`, and the configuration option `mass_property_calculate` is set to `automatic`, and regenerate the model.

You can use both reported and alternate mass property parameter values in relations, to drive other model parameters.

Configuring for Mass Property Computations

You can use the following configuration options when working with mass property parameters:

- To set Pro/ENGINEER to automatically calculate all mass properties, set the configuration option `mass_property_calculate` to `automatic`. Then the reported parameters will automatically update, based on the `PRO_MP_SOURCE` parameter value, every time you regenerate the model.
- To include into an assembly mass property report complete listings of the mass properties of the components, set the configuration option `mp_calc_level` to `all_models`. If the report is longer than 2,000,000 characters, it is not shown on screen, but the `objectname.m_p` file is written to your working directory.

To Generate a Mass Properties Report Based on Model Geometry

Generating a mass properties report based on model geometry produces the same results as a mass properties computation through model analysis (accessible by clicking **Analysis > Model Analysis**). However, a model analysis computation does not change the reported mass property parameter values.

1. Click **Edit > Setup > Mass Props**. The **Setup Mass Properties** dialog box opens.
2. Select **Geometry** as the source for mass properties calculation. If you have not changed the PRO_MP_SOURCE parameter value, this is the default source.
3. Select the coordinate system to be used.
4. If the object is a part, type a value in the **Density** box. If the object is an assembly, the system calculates the average density based on the densities of assembly components.
5. Click **Generate Report**.
6. If the object is an assembly, the system prompts you for density of each component part in turn. Enter the density values.
7. The system calculates the mass properties, displays the results in the Information Window, and saves them in a file `objectname.m_p` in the current working directory. You can use the **Edit** command in the Information Window to edit the file, or use the **File > Save As** command to save it under a different name.
8. Click **Close** when finished viewing the report.

To Assign Parameters and Generate a Mass Properties Report

This functionality lets you assign alternate values to certain mass property parameters. The system then generates the report based on these assigned values. If some of the values are not assigned, the system uses the default parameter values derived from the model geometry.

1. Click **Edit > Setup > Mass Props**. The **Setup Mass Properties** dialog box opens.
2. Select **Geometry and Parameters** as the source for mass properties calculation.
3. Select the coordinate system to be used.
4. If you want to change the default density, type a value in the **Density** box. If the object is an assembly, the **Density** box is empty. If you do not specify a value, the system will prompt you for densities of component parts at the time of generating the report, and calculate the assembly density as the average of component densities.
5. To specify the alternate parameters, click **Edit**. The **Mass Properties Parameters** dialog box opens.

6. Type the parameter values in the appropriate boxes, where desired. If you do not specify an alternate parameter value, the system will use model geometry to compute the reported parameter. Follow the link under *See Also* for a complete list of mass properties parameters.

Note: If you have previously specified alternate parameter values through the Parameters functionality, these values appear in the appropriate boxes. If an alternate parameter is driven by a relation, the appropriate box is grayed out and you can not type a value.

7. Click **OK** when finished, to close the **Mass Properties Parameters** dialog box.
8. Click **Generate Report**. The system calculates the mass properties, displays the results in the Information Window, and saves them in a file `objectname.m_p` in the current working directory. You can use the **Edit** command in the Information Window to edit the file, or use the **File > Save As** command to save it under a different name.
9. Click **Close** when finished viewing the report.

Reported and Alternate Mass Property Parameters

The following table lists all the reported (system) and alternate (user assigned) mass property parameters available in the model.

Reported Parameters	Alternate Parameters	Type	Description
	PRO_MP_SOURCE	string	Defines the default source for generating a mass properties report. Can be one of the following strings: GEOMETRY, PARAMETERS, FILE.
PRO_MP_DENSITY	MP_DENSITY	real number	Model density.
PRO_MP_MASS	PRO_MP_ALT_MASS	real number	Model mass.
PRO_MP_VOLUME	PRO_MP_ALT_VOLUME	real number	Model volume.
PRO_MP_AREA	PRO_MP_ALT_AREA	real number	Model area.

Reported Parameters	Alternate Parameters	Type	Description
PRO_MP_COGX	PRO_MP_ALT_COGX	real number	Center of gravity, X coordinate.
PRO_MP_COGY	PRO_MP_ALT_COGY	real number	Center of gravity, Y coordinate.
PRO_MP_COGZ	PRO_MP_ALT_COGZ	real number	Center of gravity, Z coordinate.
PRO_MP_IXX	PRO_MP_ALT_IXX	real number	Moment of inertia with respect to the X axis.
PRO_MP_IYY	PRO_MP_ALT_IYY	real number	Moment of inertia with respect to the Y axis.
PRO_MP_IZZ	PRO_MP_ALT_IZZ	real number	Moment of inertia with respect to the Z axis.
PRO_MP_IXZ	PRO_MP_ALT_IXZ	real number	Moment of inertia with respect to the XZ axes.
PRO_MP_IYZ	PRO_MP_ALT_IYZ	real number	Moment of inertia with respect to the YZ axes.
PRO_MP_IXY	PRO_MP_ALT_IXY	real number	Moment of inertia with respect to the XY axes.

To Use a File for Assigning Mass Properties

This functionality lets you assign mass property values by using a file. If you use this method, you have to specify values for all the parameters listed in the file template. If the file is incomplete, the system does not generate a report.

1. Click **Edit > Setup > Mass Props**. The **Setup Mass Properties** dialog box opens.
2. Select **File** as the source for mass properties calculation. A system window opens with the system editor active. Type the values for all the parameters in the

template file. When you exit the editor, the file is automatically saved in the current working directory as `objectname_amp.dat`.

3. Click **Generate Report**. The system calculates the mass properties, displays the results in the Information Window, and saves them in a file `objectname.m_p` in the current working directory. You can use the **Edit** command in the Information Window to edit the file, or use the **File > Save As** command to save it under a different name.
4. Click **Close** when finished viewing the report.

Example: An Assigned Mass Properties Template File

```

!
!   ASSIGNED MASS PROPERTIES DATA FILE
!
! Enter numerical values desired after the keywords,
! (Density, Volume etc.).
!
! Comment lines should begin with an exclamation mark (!).
! You may have as many comment lines as you wish.
! There must be at least one space between the keywords
! and values.
!
! Enter name of an existing user coordinate system which
! the center of gravity and inertia are with respect to.
! Enter SYSTEM if they are about the default system axes.
!
COORD_SYS
!
! Average density of the model
!
DENSITY
!
! Volume of the model
!
VOLUME
!
! Area of the model
!
AREA
!
! Center of gravity x,y,z with respect to the specified axes:
!
!           X           Y           Z
C_OF_G
!
! Inertia with respect to the specified axes:
!
!           Ixx          Ixy          Ixz
INERTIAX
!           Iyx          Iyy          Iyz
INERTIAY
!           Izx          Izy          Izz
INERTIAZ
!

```

Notes:

- You must enter all numerical values in the preceding template.
- Ixx, Iyy, and Izz are always > 0.
- Ixy=Iyx, Iyz=Izy, Izx=Ixz

Model Grids

About Model Grids

When you are in Part or Assembly mode, you can define and use three-dimensional model grids.

You define a model grid by choosing **Grid** from the **PART SETUP** or **ASSEM SETUP** menu that appears when you click **Edit > Setup** and then using the **Model Grids** dialog box to select a coordinate system that defines the origin of the grid.

For more information about creating, using, and modifying model grids, refer to the topics on model grids in the Detailed Drawings portion of the online Help system.

Relations and Parameters

Terminology Used in Relations and Parameters

Local parameters—Parameters that are created in the current model. You can edit local parameters in the model.

External parameters—Parameters created outside the current model and used to control some aspects of the model. External parameters cannot be modified in the model. For example, you can add parameters in Layout mode to define certain part dimensions. When you open that part, these part dimensions are controlled from Layout mode and are read-only in the part. Likewise, you can create parameters in the PDM system and apply them to the part.

User-defined parameters—Additional information that you can attach to geometry. You can add user-defined parameters to an assembly, part, feature, or entity. For example, you can create a COST parameter for each part in the assembly. Then, you can include the COST parameter in your Bill of Materials to calculate the total cost of the assembly.

System parameters—Parameters defined by the system, for example, the Mass Properties parameter. These parameters are generally read-only. You can use them in relations, but you cannot control their value.

Annotation Element parameters—Parameters defined for an Annotation Element.

Restricted value parameters—Parameters whose values and other properties are defined by an external file.

Relations—Relations are user-defined equations written between symbolic dimensions and parameters. Relations let you capture design intent by defining relationships within features or parts, or among assembly components. You can create relations on different levels: assembly, part, feature, or section.

Section relations—Relations created in Sketcher to control geometry of complex sections. Unless a section dimension changes, section relations generally are regenerated during a part regeneration.

Part relations—Relations added on a part level. During a part regeneration, these relations get resolved before features are regenerated.

Feature relations—Relations added to a selected feature. During a part regeneration, these relations are resolved after part relations, but before the regeneration of the feature to which they apply. You use feature relations to change feature geometry only after part relations are applied and some features have been regenerated.

Relations

Basic Relations Information

About Relations

Relations (also known as parametric relations) are user-defined equations written between symbolic dimensions and parameters. Relations let you capture design intent by defining relationships within features or parts, or among assembly components.

You can use relations to control the modeling process in the following ways:

- To control the effects of modifications on models
- To define values for dimensions in parts and assemblies
- To set constraints for design conditions. For example, by specifying the location of a hole in relation to the edge of a part.
- To describe conditional relationships between different parts of a model or assembly

Relations can be simple values (for example, $d1=4$) or complex, conditional, branching statements.

You can also use units in relations.

Accessing Relations

To access relations, open a model and click **Tools > Relations**. The **Relations** dialog box opens.

Relation Types

There are two types of relations:

- **Equality**—Equates a parameter on the left side of the equation with an expression on the right side. This type of relation is used for assigning values to dimensions and parameters. For example:
 - Simple assignment: $d1 = 4.75$
 - More complex assignment:


$$d5 = d2 * (\text{SQRT}(d7/3.0+d4))$$

- **Comparison**—Compares an expression on the left side of the equation with an expression on the right side. This type of relation is commonly used as a constraint or in conditional statements for logical branching. For example:
 - As a constraint: $(d1 + d2) > (d3 + 2.5)$
 - In a conditional statement: $\text{IF } (d1 + 2.5) \geq d7$

Rules for Using Relations

Consider the following rules:

- If you try to assign a relation outside the section to a parameter that is already driven by a section relation, you get an error message when the system regenerates the model. The same applies if you try to assign a relation to a section parameter that is already driven by a relation outside the section. Remove one of the relations and regenerate model.
- If an assembly tries to assign a value to a dimension variable that is already driven by a part or subassembly relation, you get an error message. Remove one of the relations and regenerate the model.
- Modifying the units of a model can invalidate your relations, as they are not scaled along with the model.

Tip: In some cases, relations defined at different levels may conflict. Check the validity of relations by clicking  and make sure your relations fulfill your design intent.

About Order of Evaluation of Relations During a Regeneration

Relations are not evaluated until the model is regenerated. During a regeneration, relations are evaluated in the following order:

1. At the beginning of a regeneration, the system solves model relations in the order in which they were entered.

In an assembly, the assembly relations are calculated first. Then the system calculates all subassembly relations in order of component placement. This means that all subassembly relations are calculated before any features or components begin regeneration.
2. The system starts regenerating features in the order of creation. If a feature has feature relations attached to it, these relations are solved before regenerating that feature.
3. If you specified any relations as Post Regeneration, the system solves these relations after the regeneration is complete.

About Simultaneous Equations

Simultaneous equations are relations in which several variables or dimensions must be solved simultaneously.

Consider these rules for creating simultaneous equations:

- You must declare variables used in simultaneous equations in the beginning of the equations. For example, `area = 100`, as shown in the next example.
- When solving the simultaneous equation, the system returns only one set of results even if more than one solution for the system of equations is possible.
- You can intermix simultaneous equations with single variable relations.

Example: Sample Simultaneous Equations

For example, you have a box of width `d1` and height `d2` and you want to specify the following conditions:

- The area equals 100
- The perimeter equals 50

You can enter the following simultaneous equations:

```
SOLVE
d1*d2 = 100
2*(d1+d2) = 50
FOR d1 d2 ...or... FOR d1,d2
```

All lines between the `SOLVE` and `FOR` statements become part of the simultaneous equations. The `FOR` line lists the variables to be solved. Any variables that appear in the simultaneous equations but not in the `FOR` list are interpreted as constants.

Alternatively, you can set the same conditions by entering the following simultaneous equations:

```
area = 100
perimeter = 50
SOLVE
d1*d2 = area
2*(d1 + d2) = perimeter
FOR d1 d2
```

Tips for creating simultaneous equations:

- Omitting `area = 100` in the preceding relation causes an error.
- You can add extra code downstream from simultaneous equations to specify a solution when there is more than one equation. For example, in the preceding examples, the two possible sets of solutions are `d1=5, d2=20` and `d1=20, d2=5`. You can add the constraint `d1 <= d2` by adding the following conditional code:

```
IF d1 > d2
temp = d1
d1 = d2
```

```
d2 = temp
ENDIF
```

Relations Errors

After you add or edit a relation and click **OK** to exit the **Relations** dialog box, Pro/ENGINEER checks the validity of relations. If errors are found, Pro/ENGINEER flags the erroneous relations so you can correct them.

Error types detected by Pro/ENGINEER include:

- Long line—The relations line has more than 80 characters. Edit the line, or break the relation into two lines by entering a backslash (\) to indicate that the relation continues on the next line.
- Long symbol—A symbol name has more than 31 characters. Edit the symbol name to be less than 31 characters.
- Parse error—A parse error occurs, for example, if a parameter is not defined.

Note: Constraint violations are not captured by this error check. If a simultaneous relation fails to converge, a warning appears in the message area. If the system detects a simultaneous relation that is not closed, an error message appears on an empty line after the last relation line.

Avoiding Circular Relations

You may erroneously create a circular relation such as this one:

```
d0 = d1
d1 = d0
```

When you sort relations, the sorting utility detects a circular relation and issues a warning. However, the preceding lines can be valid parts of a simultaneous equation.

The next example of a circular relation illustrates how you can erroneously redefine a symbol in the course of entering relations:

```
d0 = d1
...
d0 = 2*d1
```

These relations are order-dependent and cannot be sorted. The system issues a warning.

Operators and Functions Used in Relations

About Operators Used in Relations

The following operators can be used in relations, both in equations and in conditional statements.

Arithmetic Operators

+	Addition
-	Subtraction
/	Division
*	Multiplication
^	Exponentiation
()	Parentheses for grouping for example, $d0 = (d1-d2)*d3$

Assignment Operator

=	Equal to
---	----------

The equal (=) sign is an assignment operator that equates the two sides of an equation or relation. When the equal sign is used, the equation can have only a single parameter on the left side.

Comparison Operators

Comparison operators are used when a `TRUE/FALSE` value can be returned. For example, the following relation returns `TRUE` whenever `d1` is greater than or equal to 3.5. It returns `FALSE` whenever `d1` is less than 3.5:

```
d1 >= 3.5
```

The following comparison operators are supported:

==	Equal to
>	Greater than
>=	Greater than or equal to
!=, <>, ~=	Not equal to
<	Less than
<=	Less than or equal to
	Or
&	And
~, !	Not

Note: The "equal to" assignment operator is different from the "equal to" comparison operator.

The operators `|`, `&`, `!`, and `~` extend the use of comparison relations by enabling several conditions to be set in a single statement. For example, the following relation returns `TRUE` whenever `d1` is between 2 and 3, but not equal to 2.5:

```
d1 > 2 & d1 < 3 & d1 ~= 2.5
```

Conditional Statements in Relations

Note: Relations containing conditional statements are not sorted. A condition is an expression that is either `TRUE` (or `YES`) or `FALSE` (or `NO`). These values can be used interchangeably in the conditional statement. For example, the following statements can all be evaluated the same way:

```
IF ANSWER == YES
IF ANSWER == TRUE
IF ANSWER
```

IF Statement

You can add `IF` statements to relations to create conditional statements. For example,

```
IF d1 > d2
length = 14.5
ENDIF
IF d1 <= d2
length = 7.0
ENDIF
```

ELSE Statement

By adding the `ELSE` statements in the branches, you can create more complex conditional constructions. With these statements, the previous relation may be modified as follows:

```
IF d1 > d2
length = 14.5
ELSE
length = 7.0
ENDIF
```

There can be several features listed between the `IF`, `ELSE`, and the `ENDIF` statements. In addition, the `IF|ELSE|ENDIF` constructions can also be nested within feature sequences.

The syntax of the `IF` clause is as follows:

```
IF <condition>
Sequence of 0 or more relations or IF clauses
ELSE <optional>
Sequence of 0 or more relations or IF clauses <optional>
ENDIF
```

Consider the following rules:

- `ENDIF` is spelled as one word.
- `ELSE` is added on a separate line.
- Equal in conditional statements is entered as two equal signs (`==`). Assignment is entered as a single equal sign (`=`).

About Functions Used in Relations

Mathematical, graph, and curve evaluation functions can be used in relations.

Mathematical Functions

The following operators can be used in relations, both in equations and in conditional statements.

Relations can include the following mathematical functions:

<code>cos ()</code>	cosine
<code>tan ()</code>	tangent
<code>sin ()</code>	sine
<code>sqrt ()</code>	square root
<code>asin ()</code>	arc sine
<code>acos ()</code>	arc cosine
<code>atan ()</code>	arc tangent
<code>sinh ()</code>	hyperbolic sine
<code>cosh ()</code>	hyperbolic cosine
<code>tanh ()</code>	hyperbolic tangent

Note: All trigonometric functions use degrees.

<code>log()</code>	base 10 logarithm
<code>ln()</code>	natural logarithm
<code>exp()</code>	e to an exponential degree
<code>abs()</code>	absolute value
<code>ceil()</code>	the smallest integer not less than the real value
<code>floor()</code>	the largest integer not greater than the real value

Graph Evaluation Function

Graph evaluation function enables you to use graph features to drive dimensions through relations. The dimensions can be section, part, or assembly dimensions. The format is the following:

```
evalgraph("graph_name", x)
```

where:

- `graph_name`—the name of a graph.
- `x`—the value along the x-axis of the graph for which the `y` value is returned.

For sweep features, you can specify the trajectory parameter `trajpar` as the second argument of this function.

Note: A graph feature is usually evaluated for its defined x-axis values. When it is evaluated beyond the defined range, the y-axis values are extrapolated. The system calculates the extrapolated value for values of `x` less than the initial value by extending a tangent line back from the initial point. Likewise, the system calculates the extrapolated value for values of `x` greater than the final value by extending a tangent line out from the final point.

Composite Curve Trajectory Function

The trajectory parameter of a composite curve, `trajpar_of_pnt`, can be used in relations.

The following function returns a value between 0.0 and 1.0:

```
trajpar_of_pnt("trajname", "pointname")
```

where:

- `trajname`—the name of a composite curve.
- `pointname`—the name of a datum point.

The trajectory is a parameter along the composite curve at which the plane, normal to the tangent of the curve, passes through the datum point. The datum point does not have to be located on the curve—the parameter is calculated at the point on the curve that is closest to the datum point.

If the composite curve is used as the spine of a multitrajectory sweep, then `trajpar_of_pnt` either agrees with the parameter `trajpar` at this plane, or else with `1.0 - trajpar` (depending on your choice of the starting point for the sweep feature).

Operators and Functions for Strings

The following operators and functions are supported for strings:

<code>==</code>	Compares strings as equal.
<code>!=, <>, ~=</code>	Compares strings as unequal.

<code>+</code>	Concatenates strings.
<code>itos(int)</code>	Converts integers to strings. Here, <code>int</code> can be a number or an expression. Nonintegers are rounded off.
<code>search(string, substring)</code>	Searches for substrings. The resulting value is the position of the substring in the string (0 if not found).
<code>extract(string, position, length)</code>	Extracts pieces of strings.

For example:

If `param = abcdef`, then:

- `flag = param == abcdef`—returns TRUE
- `flag = abcdef != ghi`—returns TRUE
- `new = param + ghi`—new is abcdefghi
- `new = itos(10 + 7)`—new is 17
- `new = param + itos(1.5)`—new is abcdef2
- `where = search(param, bcd)`—where is 2
- `where = search(param, bcd)`—where is 0
- `new = extract(param, 2, 3)`—new is bcd

Note: If you use the `itos` function on a parameter whose value is zero (0), the return value is an empty string.

The following examples illustrate the `itos` function:

```
integer_param = 4
string_param = itos(integer_param)
/*string_param will return 4 */
integer_param = -7
string_param = itos(int_param)
/*string_param will return -7 */
```

For an integer with zero (0) value, the `itos` function returns a null ("") value as shown below:

```
integer_param = 0
string_param = itos(int_param)
/*string_param will return an empty or null string ("" ) */
```

To return a zero string value ("0"), use the following `IF` statement:

```
integer_param = 0
string_param = itos(integer_param)
IF string_param == ""
string_param = "0"
ENDIF
```

Specifying the Number of Decimal Places

You can add an optional argument to both the `ceil` and the `floor` functions that allows you to specify the number of decimal places for rounding. The syntax for functions with the rounding parameter is as follows:

```
ceil (parameter_name OR number, number_of_dec_places)
floor (parameter_name OR number, number_of_dec_places)
```

where `number_of_dec_places` is an optional value.

Consider the following information about `number_of_dec_places`:

- It can be represented as a number or as a user-defined parameter. If the parameter value is a real number, it is truncated to an integer value.
- It has a maximum value of 8. If this value is exceeded, the number being rounded (the first argument) is not rounded and the initial value is used.

The next examples show how `ceil` and `floor` are used without specifying the number of decimal places:

```
ceil (10.2) evaluates to 11
floor (-10.2) evaluates to -11
```

The next examples show the use of `ceil` and `floor` when specifying the number of decimal places:

```
ceil (10.255, 2) evaluates to 10.26
ceil (10.255, 0) evaluates to 11 [This is the same as ceil (10.255)]
floor (10.255, 1) evaluates to 10.2
floor (-10.255, 2) evaluates to -10.26
```

Passing Strings as Arguments in Relations

The following functions allow you to pass strings as arguments:

- `string_length()`—Returns the number of characters in a parameter. For example, if the value for the string parameter `material` is defined as `steel`, `string_length(material)` equals 5, because the word "steel" has five letters.
- `rel_model_name()`—Returns the current model name. For example, if you are currently working in a part called A, `rel_model_name()` is equal to A. To use this in a relation in an assembly, write the relation as follows:

```
name = rel_model_name:2()
```

Note: The parentheses () are empty.

- `rel_model_type()`—Returns the current model type. If you are working in Assembly mode, `rel_model_type()` is equal to `assembly`.
- `exists()`—Evaluates whether an item, such as a parameter or dimension, exists. This can apply to the model for which the relation is being evaluated, or to any model, component, or submodel structure.

For example:

- `if exists("d5:20")`—Checks if the model with runtime ID 20 has a dimension `d5`.
- `if exists ("par:fid_25:cid_12")`—Checks if the feature ID 25 in the component ID 12 has parameter `par`.

This allows evaluation to be based on a parameter that exists in only one part of a large assembly. For example, suppose that there are several systems in a large assembly (such as hydraulic, pneumatic, or electrical systems), but the majority of objects belong to no system.

In this case, to make evaluations that are based on the parameter, you must assign the appropriate parameter to those models that belong to the system. For example, if items in the electrical system must use a part number in the BOM report table, instead of the model name, you can create a report parameter `bom_name` and write the following relation:

```
if exists("asm_mbr_cabling")
bom_name = part_no
else
bom_name = asm_mbr_name
endif
```

Example: Graph Evaluation Function

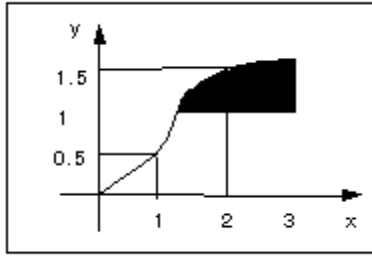
The graph evaluation function enables you to use graph features to drive dimensions through relations. The dimensions can be section, part, or assembly dimensions. The format is:

```
evalgraph("graph_name", x)
```

where

- `graph_name` is the name of a graph.
- `x` is the value along the x-axis of the graph for which the `y` value is returned.

The next figure shows a sample graph.



Graph name: `stretch`

- `d1=0.5`
- `d2=1.5`
- `d1=evalgraph ("stretch", 1)`
- `d2=evalgraph ("stretch", d1*4)`

Note: For sweep features, you can specify the trajectory parameter `trajpar` as the second argument of this function.

Parameters Used in Relations

Using Parameter Symbols in Relations

You can use the following types of parameters in relations:

- **Dimension symbols**—The following dimension symbol types are supported:
 - **d#**—Dimensions in Part or Assembly mode.
 - **d#:#**—Dimensions in Assembly mode. The Session ID of the assembly or component is added as a suffix.
 - **rd#**—Reference dimensions in part or top-level assembly.
 - **rd#:#**—Reference dimensions in Assembly mode. The session ID of assembly or component is added as a suffix.
 - **rsd#**—Reference dimensions in Sketcher (section).
 - **kd#**—Known dimensions (in parent part or assembly) in Sketcher (section).
 - **Ad#**—Driven dimensions in Part, Assembly, or Drawing mode.
- **Tolerances**—Parameters associated with tolerance formats. These symbols appear when dimensions are switched from numeric to symbolic.
 - **tpm#**—Tolerance in plus-minus symmetrical format; # is the dimension number.
 - **tp#**—Positive tolerance in plus-minus format; # is the dimension number.
 - **tm#**—Negative tolerance in plus-minus format; # is the dimension number.

- **Number of instances**—Integer parameters for the number of instances in a direction of a pattern.
 - **p#**—Where # is the number of instances.

Note: If you change the instance number to a noninteger value, Pro/ENGINEER truncates it. For example, 2.90 becomes 2.

- **User-defined parameters**—Parameters defined by adding a parameter or a relation. For example:

```
Volume = d0*d1*d2
Vendor = "Stockton Corp."
```

- **System parameters**—You can change the values of the system parameters in the **Local Parameters** box in the **Relations** dialog box. The changed values are then used in all models for the current session.

The following parameters are reserved for use by the system:

- PI (geometric constant)—Value = 3.14159
Note: You cannot change this value.
- G (gravitational constant)—Default value = 9.8 meters/sec²
(C1, C2, C3, and C4 are default values and equal 1.0, 2.0, 3.0, and 4.0, respectively.)
- **Application parameters**—You can reference application-specific parameters from the following applications:
 - Layout
 - Sheetmetal
 - Welding
 - Piping
 - Cabling
 - Mechanism
 - Simulations
 - Manufacturing

Using String Values for Parameters

You can enter string values for parameters by enclosing them within double quotation marks.

Consider the following example: A drawing note uses a parameter name within the note. The corresponding relation is as follows:

```
IF d1 > d2
MIL_REF = "MIL-STD XXXXA"
ELSE
```

```
MIL_REF = "MIL-STD XXXXB"  
ENDIF
```

Creating and Editing Relations

About Adding Relations

Typically, you add relations by clicking **Tools > Relations** and using the **Relations** dialog box.

Additionally, you can add relations in the following ways:

- Edit the relations file and add more relations.
- When creating a feature, you can type an expression in the dimension box located on the dashboard. When editing a feature, type an expression as a value for the dimension.
- When editing a dimension in the graphics window, double-click a dimension and type an expression as a dimension value.

Adding Relations to Different Types of Objects

You can add relations to different types of objects. The supported object types, listed under **Look In** in the **Relations** dialog box, are:

- **Part**—Access part relations both in the Part and Assembly modes.
- **Assembly**—Access relations in an assembly.
- **Feature**—Access relations specific to a feature in the Part or Assembly mode.
- **Inherited**—Access relations both in the Part and Assembly modes.
- **Section**—If a feature has a section, access section relations in Sketcher while in the Part or Assembly mode.
- **Pattern**—Access relations specific to a pattern in the Part or Assembly mode.
- **Skeleton**—Access relations for a skeleton model in Assembly mode.
- **Component**—Access relations for an assembly component.

Rules for Creating Relations in Parts

The following rules apply to creating or modifying relations in parts:

- Driven variables can be as follows:
 - Dimensions in the part (`d#`)
 - User parameters in the part (`parameter_name`)
 - User parameters in features (`parameter_name:fid_N` or `parameter_name:fid_feature_name`)

- Driving variables can be the same as the driven variables. In addition, you can use:
 - Reference dimensions from the part (rd#)
 - Evaluation feature measurements in the following format:
 - measurement_name:fid_N
 - measurement_name:fid_feature_name

Rules for Creating Relations in Features

When you create relations in a feature, they are saved with the feature, and stay with it regardless of the model in which the feature is used.

Feature relations are evaluated after part relations and they are solved when the feature to which they belong is regenerated. Therefore, if a relation performs geometry evaluation (such as the distance between two points), it can give different results if used as a feature relation versus being used as a part relation.

Note: You can modify existing model parameters by name from the feature level.



To Add a Relation to a Model

1. With a model open, click **Tools > Relations**. The **Relations** dialog box opens.
2. The **Look in** box shows the type of object (for example, part or feature) where the relation is applied. If you are in Part mode, by default the relation is added at the part level. You can change the object type by changing the selection in the **Look in** box, for example, you can change feature relations or section relations.


Note: If the `pattern_relations_copy` configuration option is set to `true`, the relations defined for the geometry to be patterned are copied to other instances of the pattern.

3. If you selected **Feature**, **Inherited**, **Section**, or **Pattern** as the object type, you must select the object for which you want to list relations. The selected object shows dimensions in symbolic form.

4. Type a relation in the text box and press ENTER. You can type multiple relations by pressing ENTER after each one. Use the following tools for adding a relation:

- To insert a dimension, geometric tolerance, or surface finish, click it in the graphics window. **Tip:** If dimensions are not displayed, double-click a feature to show its dimensions.
- To insert a function, click .
- To insert an operator, click an operator listed on the left toolbar of the dialog box.
- To insert a parameter from a list of existing parameters, click .

Note: The name of a parameter, variable, or symbol cannot contain more than 31 characters.

5. To specify if the relation should be evaluated in regular order or after a regeneration, select **Initial** or **Post Regeneration** from the list.
6. You can verify the validity of the relations you have entered by clicking .
7. To accept the relations, click **OK**. To start over, click **Reset**.

Specifying a Feature and Model in a Relation

You can write relations that use the user parameters and manufacturing parameters of a feature.

In a feature relation, you can identify a parameter attached to that feature by the parameter name. For example:

```
XYZ = parameter_name
```

where `parameter_name` is the user parameter.

However, if you write a relation in the feature's parent model (part or assembly) or in another feature, you must use one of the following formats to reference that feature:

```
XYZ = parameter_name:fid_N
```

or

```
XYZ = parameter_name:fid_feature_name
```

where:

- o `parameter_name`—the user parameter for the feature.
- o `N`—the internal feature ID (as displayed when you use **Info > Feature**).
- o `feature_name`—the optional user-defined feature name.

In Assembly mode, use the following format to access a user parameter in a feature of another model:

```
XYZ = parameter_name:fid_N:session_id
```

or

```
XYZ = parameter_name:fid_feature_name:session_id
```

where:

`session_id`—the session ID of the model.

For example, suppose that `dia` is a number-type user parameter in a part feature called `CUT_A`, `CUT_A`'s internal feature ID is 20, and `d5` is a dimension in another feature.

You can establish a relationship between `d5` and `dia` in the part as follows:

```
d5 = dia:fid_20
```

or

```
d5 = dia:fid_CUT_A
```

In an assembly, if the driving parameter belongs to another model with the Session ID 12, you can write the relation as follows:

```
d5 = dia:fid_20:12
```

or

```
d5 = dia:fid_CUT_A:12
```

Example: Sample Feature Relations

The next table shows a sample Relations and Parameters Information window listing feature relations.

Relation	Parameter	New Value
/** Relations for BRICK feature id = 282:		
param_feat2_x = 6.666	param_feat2_x	6.666000e+00
param_feat2_yy = param_feat1_1:FID_1	param_feat2_yy	4.567000e+00
param_feat2_xy:FID_1 = param_feat2_x	param_feat2_xy:FID_1	6.666000e+00
/*		
DEPTH = 2*param_feat2_x	DEPTH	1.333200e+01
/** Relations for feature id 282, section 1:		
param_sect_1 = sd0	param_sect_1	5.500000e+00
sd1 = 2*sd0	sd1 (D32)	1.100000e+01
/*		
sd3 = 0.03*param_part_angle1	sd3 (D34)	2.700000e+00
Symbolic constant	X-refs	Current value
PARAM_FEAT_X	Local	6.666000e+00

Relation	Parameter	New Value
PARAM_FEAT2_YY	Local	4.567000e+00
PARAM_FEAT2_X	Local	6.666000e+00

Example: Sample Part Relations

The next table shows a sample Relations and Parameters Information window listing part relations.

Relation	Parameter	New Value
/** Relations for BRICK:		
param_part_1 = 1.234	param_part_1	1.234000e+00
/*		
/* Part parameter part_param_part_angle1 is used as /* a bridge, so that the eval function parameter /* "angle:fid_angle1" can be accessed from a section.		
param_part_angle1 = angle:FID_ANGLE1	param_part_angle1	9.000000e+01
/*		
param_part_1 = 1.234	param_part_1	1.234000e+00
/** Relations for feature id 282, section 1:		
param_sect_1 = sd0	param_sect_1	5.500000e+00
sd1 = 2*sd0	sd1 (D32)	1.100000e+01
/*		
sd3 = 0.03*param_part_angle1	sd3 (D34)	2.700000e+00
Symbolic constant	X-refs	Current value
PARAM_PART_1	Local	1.234000e+00

Relation	Parameter	New Value
PARAM_PART_ANGLE1	Local	9.000000e+01
PARAM_SECT_1	Local	5.500000e+00

Using Comments in Relations

It is good practice to use comments in your relations. Comments help you remember why you added the relation. They also benefit others who use your models.

Every comment line must begin with a slash and an asterisk. For example:

```
/* Width is equal to 2*height
d1 = 2*d2
```

The comment must precede the relation to which it applies. When relations are sorted, the comment moves with the relation and remains above it.

To Determine the Value of an Expression

You can obtain a value of an expression, either a single parameter or an equation, without adding it to the model as a relation.

1. Click **Tools** > **Relations**. The **Relations** dialog box opens.
2. Click **=?**. The **Evaluate Expression** dialog box opens.
3. Enter the parameter symbol (for example, `d20`, `$d20`, `d10:2`, `depth`), user parameter (for example, `color`), or expression (for example, `d5 + d6`, `d2 != d4`) in the **Expression** box.
4. Click **Evaluate**. The result of the evaluation appears in the **Result** box. Comparison equations such as `d2 != d4` or `d3 == d2` return either a 0 (false) or a 1 (true).
5. Click **Close**.

To Edit a Relation

You can edit an existing relation using the editing tools in the **Relations** dialog box.

1. Click **Tools** > **Relations**. The **Relations** dialog box opens, listing existing part relations.
2. To show relations for a different object type, change the selection in the **Look in** box.
3. Click **Edit** and choose an editing command.
4. When finished editing, click **OK**.
5. Regenerate the model to reflect changes in the relations.

To Delete an Obsolete Relation

When you delete or change a dimension or parameter that is used in a relation, the relation becomes obsolete. The driven symbol retains the last correct value of the dimension.

Note: Pro/ENGINEER does not automatically delete them. You must manually delete obsolete relations.

To delete an obsolete relation, click **Edit > Delete** on the **Relations** dialog box.

Viewing Relations

To View a Relation

You can view existing relations using the **Show** command in the **Relations** dialog box.

Tip: You can also use **Info > Feature** to view feature relations.

1. Click **Tools > Relations**. The **Relations** dialog box opens.
2. By default, the system shows part relations. You can change the object type in the **Look in** box to show relations of a particular object type, for example, feature relations or section relations.
3. If you selected **Feature, Inherited, Section, or Pattern** in the **Look in** box, you must select the object for which you want to list relations.
4. Click **Show > Info**. The **Relations and Parameters** Information window opens in the Pro/ENGINEER browser.

Relations and Parameters Information Window

The **Relations and Parameters** Information window is organized as follows:

- **In a part or assembly**—Displays the relations and user parameters created in that model and in its feature sections. It does not list relations or parameters created in its features.
- **In a feature**—Displays relations created in that feature and in its sections (if applicable), as well as user parameters created in that feature.
- **In a pattern**—Displays pattern parameters, relations, and user parameters created in that pattern for the selected direction, as well as user parameters and section relations in the parent model.
- **In a section**—Displays the relations created in that section only.

User parameters are displayed under the heading **Symbolic constant** with the following information:

- Parameter name
- Cross-reference status—either Local (associated only with a part or assembly) or Global (associated with a project)

- Number of cross-references if the parameter is global
- Current value of the parameter

Sorting Relations

About Sorting Relations

The sort function sorts relations in order of precedence, based on how they are evaluated by the system. If one relation depends on the value of another relation, it is reordered accordingly. Sorting relations helps you detect undesired circular relations.

To sort relations, click  on the **Relations** dialog box or click **Utilities > Reorder Relations**.

Note: Relations containing conditional statements are not sorted.

Sorting Relations with Comment Lines

Relations with comment lines can also be sorted. The comment line is attached to the relation below it, and moves with that relation during a sorting. If multiple comment lines precede a relation, they are attached to that relation.

Example: Sorting Relations

Example 1: Sorting Regular Relations

For example, you entered the following relations:

$$d0 = d1 + 3*d2$$

$$d2 = d3 + d4$$

When you sort the relations, the system reorders them based on the order in which they will be evaluated. The sorted relations appear as follows.

$$d2 = d3 + d4$$

$$d0 = d1 + 3*d2$$

Example 2: Sorting Simultaneous Equations

A set of simultaneous equations is treated as if it is a single relation, and is moved around as a unit. This unit is considered to be dependent upon the union of all the quantities that its individual lines depend upon. A simultaneous equation may contain additional relations that are not being solved, as shown in the following example:

SOLVE

$$d1 + d2 = 500$$

$$d55 = d56 + 50$$

$$d56 = d54$$

```
d1-d2 = 0
FOR d1 d2
```

The extra lines are sorted within the body of the simultaneous equation, producing the following result:

```
SOLVE
d1 + d2 = 500
d1-d2 = 0
d56 = d54
d55 = d56 + 50
FOR d1 d2
```

Note: Lines are never inserted into or removed from simultaneous equations.

Dimensions and Relations

To Switch Dimension Format to Symbolic

When you click **Tools > Relations**, Pro/ENGINEER switches the dimension display to symbolic format (for example, d_0).

You can switch between symbolic and numeric dimension formats in one of the following ways:

- In the **Relations** dialog box, click **Utilities > Switch Dimensions**.
- On the menu bar, click **Info > Switch Dimensions**.

Tip: If dimensions are not already displayed on the model, select the feature or part whose dimensions you want to display.

Note: Symbolic tolerances appear only for dimensions with plus-minus or symmetrical formats. Other format types display only the dimension symbol.

Negative Dimensions in Relations

If you are using negative dimensions, and you want to capture the true signed value in your relation, precede the symbol with a dollar sign (for example, $\$d_{20}$ or $\$depth$).

You must do this regardless of the setting for the `show_dim_sign` configuration option.

Tip: Click **Utilities > Highlight** in the **Relations** dialog box to display a specific dimension.

Modifying Dimensions Driven by Relations

If a dimension is driven by a relation, the dimension cannot be modified directly. If you try to do so, Pro/ENGINEER displays an error message.

For example, if you enter the relation $d_0=d_1+d_2$, you cannot modify d_0 directly. You must modify d_1 or d_2 , or edit the relation in order to change the value of d_0 .

If a dimension symbol is modified, the change is automatically reflected in the relation file.

Relations in Sections

About Creating Relations in Sections

A section relation captures relationships among section entities. The section relation is stored with the section and is evaluated by the system regardless of where the section is used.

You can use the following variables in the section relations:

- Driven variables can be as follows:
 - Dimensions in the section (`sd#`) or in the parent model (`d#`)
 - In Assembly mode, dimensions in another model (`d#:#`)
 - User parameters in the parent model only (no session-ID suffix allowed)
- Driving variables can be as follows:
 - Dimensions in the section (`sd#`, `rsd#`, or `kd#`), parent model (`d#`, `rd#`), or other model in an assembly (`d#:#`, `rd#:#`)
 - User parameters in the parent model only (no session-ID suffix allowed)

Tips for Creating Section Relations

- Relations that define dimensions other than section dimensions are best defined at the feature or model level.
- You cannot reference a parameter in another feature as a driving variable. For example, if `dia` is a dimension in another feature with `fid_20`, the system does not accept the following section relationship:

```
sd3 = dia:fid_20
```

However, you can establish the same relationship at the feature level or model level by using the model-level equivalent (`d#`) of the section dimension (`sd#`). Alternatively, you can create an intermediate user parameter in the parent model and then reference it from the section.

- If you try to assign a relation outside of the section to a parameter that is already driven by another relation in the section, the system issues an error message upon regeneration. The same applies if you try to assign relations in a section to a parameter that is already being driven by a relation outside the section. Remove one of the relations and regenerate again.
- When you create a relation of a graph feature, for example, `sd1=evalgraph("CONIC.....", trajapar....)`, this relation becomes a feature relation. Feature relations are evaluated after part relations and are solved when the feature to which they belong is regenerated. Therefore, if a

relation performs geometry evaluation (for example, the distance between two points), it can give different results if used as a section relation as opposed to being used as a part relation.

Specifying User-Defined Parameters in Sections

At the section level (Sketcher mode), user parameters can be created only through relations.

The user parameters are evaluated and stored in the parent model where they can be accessed as other model parameters. You can also assign relations in a section to user parameters in the parent model.

Note: You cannot access user-defined parameters from the parent feature of the section.

If you choose **Show** for the parent model (**Tools > Relations > Show** in the **Relations** dialog box), the Information window displays the relations in the top level of that model and in the feature sections (but not at the feature level) and it lists the user parameters under the heading Symbolic constant. It does not make any distinction between parameters created at the model level and those created at the section level. The parameters are listed in order of creation.

You can modify or use user-defined parameters from the parent model in section relations. You can assign values to the same parameter in several sections. After regeneration, the parameter has the value of the last relation to be evaluated.

Note: If you try to assign a relation outside the section to a parameter that is already driven by another relation in the section, you get an error message when the system regenerates the model. The same applies if you try to assign relations in a section to a parameter that is already being driven by a relation outside the section. Remove one of the relations and regenerate.

Example: Sample Sections Relations

The following table shows a typical Information window for one of the sections used in the part.

Relation	Parameter	New Value
/** Relations for section:		
param_sect_1 = sd0	param_sect_1	5.500000e+00
sd1 = 2*sd0	sd1 (D32)	1.100000e+01
/*		
sd3 = 0.03*param_part_angle1	sd3 (D34)	2.700000e+00

Relations in Assemblies

To Specify a Parameter in Another Model

If you refer to a parameter that belongs to another model, the parameter must include the Session ID of the referenced model.

The format for assembly relations is as follows:

```
XYZ = parameter_name:session_id
```

where

- XYZ is a parameter in the currently selected model.
- parameter_name is a parameter in the referenced model.
- session_id is the Session ID of the referenced model.

In the general case, you can express the preceding relations as follows:

```
XYZ:session_id_1 = parameter_name:session_id_2
```

where

- session_id_1 is the Session ID of the driven parameter's model.
- session_id_2 is the Session ID of the driving parameter's model.

To Determine the Session ID of a Model in Assembly Mode

1. With an assembly open, click **Tools > Relations**. The **Relations** dialog box opens.
2. Click **Show > Session ID**. The **MODEL INFO** menu appears.
3. Select one of the following object types:
 - **Assembly**
 - **Part**
 - **Skeleton**
4. Select a component. The name and Session ID of the object appear in the message window.

Using a Session ID of a Component in Assembly Relations

When you create a parameter that belongs to a model (assembly or component) other than the current model, the parameter must include the Session ID of that model as a suffix in the dimension symbols.

Note: The use of Session ID for parameters in the current model is optional.

You can view the Session ID of a component by clicking **Show > Session ID** in the **Relations** dialog box.

In an assembly, the following terms are equivalent for components:

- Component Number = Feature Number
- Component ID = Feature ID

How a Session ID is Generated

When an assembly is created or retrieved into a session, each unique model (whether a top assembly, subassembly, or part) is assigned its own Session ID.

If a model is used more than once in an assembly, each occurrence of the model has a unique Component Number and Component ID, which relate to its parent assembly. However, all occurrences of the model share the same Session ID. While the model is in session, it retains the same Session ID, even if it is used in different assemblies.

Different instances of a table-driven model have different Session IDs.

Example: Session IDs for Models in an Assembly Tree

The following example lists typical Session IDs assigned to the components during a Pro/ENGINEER session.

Model	Session ID
TEST_ASSY.ASM	1
BASE_ASSY.ASM	3
BASEPLATE.PRT	6
BUSHING.PRT	2
SNAPRING.PRT	4
BASEPLATE_1.PRT	8

Using an Internal Component ID in Assembly Relations

When creating a relation in Assembly mode, you may need to know a Component internal ID. Every component in an assembly has a unique Component ID, even if two components have the same component name and therefore the same Session ID.

Use the following syntax when referencing a Component ID:

```
d#:cid_(component ID #)
```

To determine the Component internal ID:

1. Click **Info > Component**. The **Component Constraints** dialog box opens.
2. Select a component.

- Click **Apply** on the **Component Constraints** dialog box. The Information window displays the component information including the Component ID.

Note: The Model Tree in Assembly mode displays the Feature ID (the same as the Component ID) of each assembly component.

Example: Using a Component ID in an Assembly Relation

If you create an assembly in which the same generic components is assembled twice, you can write the following relation to sum up the lengths of these two components:

$$\text{Total_length} = \text{d3:2} + \text{d3:2}$$

where `d3` is the dimension and `2` is the Session ID of the generic.

If you replace one of the generic components with an instance from its Family Table, this relation is no longer accurate. If the assembly contains one generic and one instance, you can change the relation to read as follows:

$$\text{Total_length} = \text{d3:4} + \text{d3:4}$$

where `d3` is the dimension and `4` is the Session ID of the instance that replaced the generic.

Pro/ENGINEER cannot determine which of the two dimensions must be updated to reflect the Session ID of the instance. To address this situation, you can write assembly relations using internal Component IDs (`cid`) which are unique for every assembly component.

If you are using two identical generic components as in the first example, the relation changes as follows:

$$\text{Total_length} = \text{d3:cid}_1 + \text{d3:cid}_3$$

where:

- `d3`—the dimension
- `cid_1`—refers to the first generic component
- `cid_3`—refers to the other generic component.

If one of the generic components is replaced with one of its instances, the relation is properly updated to reflect this change and `total_length` properly calculates the sum of the lengths of the generic component and the instance component.

Example: Sample Assembly Relations

The following example shows some typical relations for the assembly Model Tree.

```

CODING TABLE      SESSION ID      MODEL NAME
-----
CODING TABLE      1              TEST_ASSY.ASM
CODING TABLE      12             BASEPLATE_1.PRT
CODING TABLE      8              SNAPRING.PRT

RELATION              PARAMETER
-----
/** Relations for TEST_ASSY:
param_assy_1 = 1.234              param_assy_1
param_assy_2:1 = D7:12           param_assy_2:1
param_assy_3 = 0.1*RD5:12 + 9    param_assy_3
param_snapring:12 = D6:1         param_snapring:1

/** Relations for feature id 31, section 1:
sd0 = 15*param_assy_1           sd0 (D4)
sd2 = 0.025*RD0:1 + 1          sd2 (D6)

Symbolic constant      X-refs      Current va
-----
PARAM_ASSY_1           Local       1.234000e+
PARAM_ASSY_2           Local       4.128000e+
PARAM_ASSY_3           Local       1.700000e+

```

C Programs and Relations

About an Interactive C Program

You can create your own interactive program to drive model geometry through relations. Consider the following information on creating user programs:

- The **User Prog** command is available only in Part and Sketcher modes.
- Only one user program may be edited or run at a time. However, you can write multiple programs for each part and then run them sequentially.
- Each program is limited to a single entry subroutine, called "USRMAIN ()". The entry subroutine can call other subroutines in that program file.
- Pro/ENGINEER provides a C file template `usermain.c` to help you write and use user programs.
- You must capitalize Pro/ENGINEER dimensions in user programs.
- Use `D[i]` to refer to a dimension on the screen. This symbol corresponds to `di` in Part or Assembly mode or to `sdi` in Sketcher mode. The system does not accept

a user-assigned symbol name, such as width, instead of the system-assigned dimension symbol.

- Do not use exit statements in any of your routines. To build error handling into your program, use `printf` or `fprintf` statements.
- You cannot drive assembly dimensions or pattern instance numbers using user programs.

To use an interactive program, you must complete the following steps:

1. Create a program. Alternatively, you can create a program by editing an existing program.
2. Compile and link the program.
3. Run the program to implement the changes.
4. When the program is finished running, regenerate the model to update geometry.

To Edit an Interactive C Program

Use this procedure to create a new or edit an existing program.

1. Click **Tools > Relations**. The **Relations** dialog box opens.
2. Click **Utilities > User Prog > Edit**.
3. Enter the program name without the `.c` extension. An editor window opens. If you are editing an existing program, Pro/ENGINEER retrieves the specified program file.
4. If you are creating a new program, you must first copy the C file `templateusermain.c` into the editor. Do one of the following:
 - On UNIX—Use the editor defined by the configuration file option `pro_editor_command`. If you are using the vi editor, enter `[:]`, and then `[r $PRO_USRMAIN]`. `PRO_USRMAIN` is an environmental variable that contains the path to `usermain.c`.
 - On Windows—Use the Windows Notepad editor. Open a template file and save it using **Save As**.
5. Edit the program.
6. Save the program. Pro/ENGINEER becomes active.

Note: You can define only dimension symbols.

Using the usermain.c Template

Pro/ENGINEER provides a C file template `usermain.c` to help you write user programs. This template has full read and write access to the parameter list through the C programming environment. The template describes how to use parameters as variables, how to declare nonparameter variables, and where to enter the executable routines. The template is stored in the `.../usrprog/umain` directory.

If you want to add your own include files, enclose their names in quotes:

```
#include "my_funcs.h"
```

The system looks for those files first in the current directory and then in the directory defined by the environment variable `PRO_USRINCS`. By default, `PRO_USRINCS` is usually defined as the directory that contains `usermain.c`. If you want the system to look in another directory, you can either change the value of `PRO_USRINCS` appropriately or provide the full path of the file in the include statement:

```
#include "/usr/C_libs/my_funcs.h"
```

To Determine Environment Variables

Use this procedure to determine the paths of the current environment variables.

1. Click **Window > Open System Window**.
2. When the shell window opens, do the following:

- o On UNIX, enter the following command:

```
[echo $PRO_USRMAIN $PRO_USRINCS]
```

- o On Windows, enter the following command:

```
[echo %PRO_USRMAIN% %PRO_USRINCS%]
```

The system prints the full paths in the shell window.

To Link an Interactive C Program

After you edit a C program, you must compile and link it. Consider the following information for compiling a program:

- On UNIX—You must define the directory path to the compiler/linker utility "cc" in the search path of the environment variable `PATH`.
- On Windows—You must define two environment variables:
 - o `NT_COMPILER`—Its value must be the 32-bit incremental C compiler command.
 - o `LIB`—Its value must be the directory path from the load point of the compiler to its libraries (`.LIB` files).

1. Click **Tools > Relations**. The **Relations** dialog box opens.
2. Click **Utilities > User Prog > Link**. The system starts compiling the program. If the system detects errors, it prints error messages in the startup window. If necessary, edit and relink the program.

To Execute an Interactive C Program

After you successfully link a user program, you can run it to implement the changes.

Note:

- The system does not run user programs automatically. You must rerun the program each time you want to update the part.
 - Upon completion of the program, you must regenerate the part to update geometry.
1. Click **Tools > Relations**. The **Relations** dialog box opens.
 2. Click **Utilities > User Prog > Run**.
 3. Enter the program name. The program begins executing. Any printed output is written to the startup window. The program prompts you for necessary input. When you execute the program, the system creates two temporary files:
 - spgtousr.dat lists the parameters and their values before the program is executed.
 - usrtospg.dat lists the parameters and their values after the program is executed.

You can delete these files after you make sure the program runs properly.

4. When the program is finished running, regenerate the model to update geometry.

Example: User Program

The program computes the thickness of a pressure vessel based on its diameter and material parameters such as working pressure and allowable stress. The program derives the diameter from the model. The user enters material parameters in the startup window according to prompts. The computed thickness is passed back to Pro/ENGINEER.

[Program starts with next line]

```
/* Template for writing user programs in the Pro/ENGINEER environment */
/* the following definition and include file are necessary */
```

```
#define USRMAIN
```

```
#include "spgusrgl.h"
```

```
/* This include file establishes useful variables for user needs. Those are:
```

```
1. D[i] refers to dimensions you see on the screen. 'i' is the index of a
```

```
dimension you see on the screen.
```

```
So, D[0] has a value corresponding to "d0" on the screen (or "sd0" if you are
```

```
in SKETCHER mode). D[4] corresponds to "d4" (or "sd4") on the screen, and so forth.
```

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2. DIM_NUM Total number of dimensions on the screen.

```
*/

usrmain()

/* This program computes the thickness for a pressure vessel given the
diameter from the model and various material and design constants input
by the user. */

{
/* ----- local variables ----- */
/* Declare any additional variables or functions you need here. */

double sqrt ();

double thickness, f_s, joint_eff, allow_stress, press, dia;

/* ----- executable code ----- */

/* Prompt user for input. Prompts appear in the startup window. You must
also
enter your responses there. */

printf("Enter material parameters\n");
printf("\n Working Pressure [psia]:\n");
scanf("%lf", &press);

printf("\n Factor of Safety [ ]:\n");
scanf("%lf", &f_s);

printf("\n Joint efficiency [ ]:\n");
scanf("%lf", &joint_eff);

printf("\n Allowable stress [psia]:\n");
scanf("%lf", &allow_stress);

/* Compute thickness of pressure vessel based on equations in Mechanical
Analysis
and Design, A.H. Burr, Elsevier, 1982.

Note use of dimension D[2], the pressure vessel diameter, in computing
the new
thickness */

thickness = press * D[2] * f_s / (2 * joint_eff * allow_stress);

/* Pass new thickness back to Pro/ENGINEER */

D[3] = thickness;
```

}

[Program ends with previous line]


Parameters

About User Parameters

User parameters are useful for providing additional information about an object. For example, in Family Tables the parameter **Cost** may have a different value for each instance. User parameters are stored with the model. You can add a parameter without defining a relation. To view existing user parameters, click **Tools > Parameters**.

Methods for Creating Parameters

You can add parameters in different ways:

- Click **Tools > Parameters** and use the **Parameters** dialog box.
- In the Model Tree, right-click a feature for which you want to add parameters, and then select **Edit Parameters** on the shortcut menu. **Note:** You can select multiple objects and then use the **Edit Parameters** command to simultaneously create or edit common parameters in all selected objects.
- For Annotation element parameters, click  to access the **Annotation Feature** dialog box. Click an element for which you want to define a parameter, and then click **Parameters**.

Specifying the Object Type

You can add user parameters to different object types that are listed in the **Look in** box in the **Parameters** dialog box. The following object types are supported:

- **Assembly**—Access assembly parameters.
- **Skeleton**—Access skeleton parameters.
- **Component**—Access component parameters.
- **Part**—Access part parameters.
- **Feature**—Access feature parameters.
- **Inherited**—Access inherited parameters.
- **Quilt**—Access quilt parameters
- **Surface**—Access surface parameters.
- **Edge**—Access edge parameters.
- **Curve**—Access curve parameters.
- **Composite Curve**—Access composite curve parameters.

- **Annotation Element**—Access parameters defined for an element of an annotation feature.
- **Material**—Access material parameters.

Rules for Naming Parameters

Consider the following rules for naming user parameters:

- User parameter names must begin with a letter if they are to be used in relations.
- You cannot use `d#`, `kd#`, `rd#`, `tm#`, `tp#`, or `tpm#` as user parameter names because they are reserved for use with dimensions.
- User parameter names cannot contain nonalphanumeric characters such as `!`, `@`, `#`, and `$`.
- You cannot change the name of a user parameter once it has been created.

About Parameter Properties

For each parameter, you can define the following properties:

- **Name**—You cannot edit the name of an existing user-defined parameter.
- **Type**—The following parameter types are supported:
 - **Integer**—The value for this parameter is a number.
 - **Real Number**—The value for this parameter is a decimal number.
 - **String**—The value for this parameter is a string.
 - **Yes No**—The value for this parameter is YES or NO.
 - **Note**—The value for this parameter is the ID of a model note.
- **Value**—Specifies the parameter value.
- **Unit**—Defines units for a parameter from the list of units.

Note: You can define units only for the parameter type Real Number and only while creating a parameter.
- **Designate**—You can designate selected system and user parameters for use as attributes in Pro/INTRALINK or another PDM system.
- **Access**—Defines access to the parameter as follows:
 - **Full**—Parameters with full access are user-defined parameters. You can modify these parameters from any application.
 - **Limited**—You can set full access parameters to limited access. You cannot modify parameters with Limited access by a relation. Limited access parameters are modified only through Family Tables and Program.

- **Locked**—Parameters with locked access are parameters that can be locked either by an external application such as Data Management System, Analysis features, Relations, or Program, or by a user. You can modify parameters locked by an external application only from within an external application. You cannot modify user-defined locked parameters from within any external application.
- **Source**—Indicates where the parameter was created or from where it is driven.
- **Description**—Provides a description for the parameter.
- **Restricted**—Indicates restricted value parameter whose properties are defined by an external file.

Nonassociativity of User Parameters

The values of parameters defined using **Tools > Parameters** are non-associative. Even if you define the parameter value using a system-defined parameter such as a dimension or mass property parameter, these values do not update upon regeneration of the model. Pro/ENGINEER requires you to add a relation to make parameter values associative.

For example, if you have a cube that measures 10x10x10 units, and if you create a parameter `LENGTH = d5`, `LENGTH` is initially evaluated to 10. However, if you modify `d5` to 20, `LENGTH` remains 10.

To associate the parameter value with the model:

1. Click **Tools > Parameters** and add a parameter `LENGTH=15`.
2. Click **Tools > Relations** and add a relation `d5=LENGTH`.
3. When you regenerate the model, `d5` updates to the new value of 15.


Using the Parameters Dialog Box

You can access the **Parameters** dialog box by clicking **Tools > Parameters** on the Pro/ENGINEER menu bar. However, while working with NC or CMM objects, you can open the **Parameters** dialog box by clicking **Setup > Parameters** on the menu manager.


Use the **Parameters** dialog box to:

- Search for parameters
- Select parameters
- Use filters
- Delete parameters
- Use parameter table information
- Work with rows and columns

To view the properties of a parameter, select a parameter in the **Parameters** dialog box and click **Properties** to open the **Parameter Properties** dialog box.

You can update the value of an existing parameter and use  or **Edit > Regenerate** to regenerate the model without leaving the **Parameters** dialog box.

Searching for Parameters

Use  or **Edit > Find** in the **Parameters** dialog box to search for parameters either by name, value, or any other property. You can select the following parameter properties in the **Search** dialog box:

- Name
- Type
- Value
- Unit
- Designated
- Access
- Source
- Description
- Restricted

You can search for parameter types by typing the first letter or the first few letters of its value followed by an asterisk (*). For example, `NAME = len*` displays parameter names starting with the letters `len` in the **Parameters** dialog box.

Selecting Parameters

You can select parameters in the **Parameters** dialog box in the following ways:

- Select **Only Shared** in the **Filter By** list to view, select, and edit all the parameters shared by the multiple features selected in the **Look In** list.
- Click the row header for the required parameter to select the entire row in the parameter table. You can also right-click the row header and use the shortcut menu to select a parameter, create a new parameter, edit parameter properties, delete a parameter, designate a parameter, specify a unit for a parameter, and view information for a parameter.

Note: The row header precedes the first column of the each parameter row in the parameter table.

- Press `CTRL` as you click for random selection of multiple items.
- Press `SHIFT` as you click to select all the parameters between the first and last selected entries.

- You can select multiple parameters and perform the following operations:

Operation	Condition	Result
Delete	Not applicable if a parameter is locked.	Deletes all selected parameters.
Designate	Applicable if at least one or more parameters are not designated.	Designates all selected parameters.
Undesignate	Applicable if at least one or more parameters are designated.	Undesignates all selected parameters.

Filtering Parameters

Using **Filter By**, you can filter the parameters by the following main categories:

- All
- User Defined ONLY
- PDM ONLY
- Designated ONLY
- Type
- Access

Note: The **Sub Items** option is available only when you select **Type** and **Access**.

You can hide or show the filter by clicking **Edit > Hide Filter** or **Edit > Show Filter**, respectively.

Deleting Parameters

Consider the following rules when deleting parameters:





- You cannot delete a user parameter that is driven by a relation or used in a relation. You must first delete the relations where the parameter is used before deleting the parameter.
- You cannot delete an individual parameter belonging to a table restricted set. However, you can delete the entire table restricted set.
- If you have added a parameter column to the Model Tree, you can delete the parameter by clearing the corresponding cell on the Model Tree.

Using Parameter Table Information

Use **File > Export** in the **Parameters** dialog box to export the entire parameter table in the Comma Separated Value (CSV) format or in the text format. You can open such a CSV file in Microsoft Excel.

Working with Rows and Columns

You can manipulate columns in the **Parameters** dialog box as follows:

- You can add or remove a column in the **Parameters** dialog box. To add a column, click  to open the **Parameter Table Columns** dialog box. Select the required column name from the **Not Displayed** section, and click . Click **OK**. The column is added to the **Parameters** dialog box. Similarly, to remove a column click , select the required column name from the **Displayed** section and click  to transfer the column name to the **Not Displayed** section.
- You can sort the listed parameters 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 resize the columns. Pro/ENGINEER retains the changed column size and its position.
- You can also reorder parameter rows by moving one or more rows in the upward or downward direction by clicking the arrows located at the right of the parameters table in the **Parameters** dialog box. The new order of parameters is retained even after the model is saved.

Syntax for Including Parameters in Notes

To include parameters in a note, use the syntax shown in the next table.

Syntax	Usage
&<param_name>:FID_<feat_ID>	Include feature parameters in a note.
&<param_name>:SID_<surface_name>	Include surface parameters in a note.
&<param_name>:EID_<edge_name>	Include edge parameters

Syntax	Usage
	rs in a note.
&<param_name>:CRV_<curve_name>	Include curve parameters in a note.
&<param_name>:CCRV_<composite_curve_name>	Include composite curve parameters in a note.
&<param_name>:NID_<annotation_element_name>	Include Annotation Element parameters in a note.
&<param_name>:QID_<quilt_name>	Include quilt parameters in a note.
&<param_name>:MTRL_<material_ID>	Include material parameters in a note.

Including Multiple Parameters in a Single Note

You can include multiple parameters in a single Annotation Element note.

Consider the following example:

Suppose, in your note you want to reference parameters of two features:

- Cut (id 66), which has the following parameters defined:
 - HOLE_TYPE with a value M6
 - HOLE_DEPTH with a value 10
- Chamfer (id 115), which has the following parameters defined:

- CHAMFER_LENGTH with a value of 1
- CHAMFER_ANGLE with a value of 45

To include these parameters, create the following note:

```
&HOLE_TYPE:FID_66 hole, &HOLE_DEPTH:FID_66 deep  
with chamfer &CHAMFER_LENGTH:FID_115 X &CHAMFER_ANGLE:FID_115°
```

The resulting note appears as follows:

```
M6 hole, 10 deep  
with chamfer 1 X 45°
```

Using a Note Parameter to Include Model Notes in Family Tables

Use a note parameter to include model notes in Family Tables and lists of relations. The value of a note parameter is the ID of a model note. The parameter can be included in a Family Table, with different instances using different notes present in the generic component. In addition, when you show relations for the model, all note parameters are listed, including their ID and textual content.

To Create a Parameter

1. Click **Tools > Parameters**. The **Parameters** dialog box opens.
2. Under **Look In**, select the desired object type to which you want to add a parameter. If you select **Feature, Inherited, Section, Pattern, or Annotation Element** as the object type, you must select the object for which you want to list relations.

3. Click  and type a new parameter name in the **Name** column.

Note: Parameter names cannot contain nonalphanumeric characters such as !, ", @, and #.

4. Under **Type**, select the type of parameter that you want to add.
5. Specify the value of the selected parameter in the **Value** box.
6. Optionally, select the **Designate** box to make the parameter visible in the PDM system such as Pro/INTRALINK or Windchill.

Note: You can also designate feature level or geometry level parameters in a part. When you assemble a part with designated parameters in an assembly, the designated parameters are listed in the assembly BOM if you have configured the BOM to show the designated objects.

7. Under **Access**, specify access to the parameter as **Full, Limited, or Locked**.
The **Source** cell reflects the origin of the parameter, for example, **User-Defined**.
8. Optionally, add a note about the parameter under **Description**.

9. Click **OK**.

Note:


- You can use **Edit Parameters** on the shortcut menu to edit single as well as multiple items.
- If you have included a parameter in a Family Table, layout note, or relation, then you can also modify the parameter by editing that Family Table, note, or relation.

Tip: Creating Common Parameters


Use the **Edit Parameters** command as follows to create parameters common to different features in a model.

1. Holding down the CTRL key, select multiple entities or Annotation Elements on the Model Tree, for which you want to create common parameters.
2. Right-click the selected entities and click **Edit Parameters** on the shortcut menu. In the **Edit Parameters** dialog box that opens, the **Look In** list displays the entities selected on the Model Tree. The parameters table displays common parameters with the first common parameter selected by default.

Note: To create a new common parameter for all the entities selected in the **Look In** list, you must clear the selection of the parameter selected by default. Otherwise, the new parameter is created for the feature to which the selected parameter belongs.

3. Click  to create a parameter. A new common parameter is created.

To Delete a Parameter

1. Click **Tools > Parameters**. The **Parameters** dialog box opens.
2. Click the parameter name that you want to delete.
3. Click . The selected parameter is deleted.

About Copying Parameter Definitions

You can copy parameter definitions between models or features in the same session. However, you can copy the parameter definitions only within the **Parameter** dialog box.

While copying the parameters or parameter definitions, you can copy:

- Multiple parameter definitions at the same time
- The user-defined parameters and user-locked parameters but not system parameters
- Parameters from a restricted parameter table

- Only the value of the parameters definition but not the relation argument while copying a relation-driven parameter
- Parameters from different context, such as a feature parameter and a model parameter at the same time if you are viewing multiple parameters

To Copy Parameters between Two Models in Session

1. Click **Tools** > **Parameter**. The **Parameters** dialog box opens.

Note: In NC Manufacturing or Pro/CMM, you can open the **Parameters** dialog box by clicking **Setup** > **Parameters** on the menu manager.

2. Right-click any cell for the parameter definition that you want to copy. A shortcut menu appears.

Note: To select more than one parameter definition at a time, hold down the CTRL key and select the required parameter definitions.

3. Click **Select Parameter**.
4. Click **Edit** > **Copy**. The selected parameter definition is copied to the clipboard.

Note: You can paste the copied parameter even if you close the Parameters dialog box, and open it but in the same session. The clipboard content remains valid as long as all source models, from where the parameters are copied, are in session. If you end the session, the clipboard is cleared.

5. Open another model or select another feature to which you want to copy the selected parameter definition.
6. Click **Edit** > **Paste**. The parameter definition is copied to the selected model or feature.

Note: If the name of the parameter being copied already exists in the model to which it is being copied, the **Paste Override** dialog box opens. Click **Confirm** to replace the existing parameter definition with the copied parameter definition.

To Designate Parameters for Use in Pro/INTRALINK

Before you designate the parameter in Pro/ENGINEER, a system administrator for Pro/INTRALINK must create a parameter within Pro/INTRALINK. This parameter must have the same name as the model parameter that you want to designate, and must be specified as file-based. For more information about setting up parameters in Pro/INTRALINK, refer to the *Pro/INTRALINK Administration Guide*.

1. Click **Tools** > **Parameters**. The **Parameters** dialog box opens.
2. Locate the parameter and click the cell under **Designate**.

Note: The designated dimensions are displayed along with its database names. For example, LENGTH (d0), WIDTH (d1), and so on.

Using the Designate Dialog Box

The purpose of designating parameters is to:


- Make model parameters visible to a PDM system such as Pro/INTRALINK or Windchill.
- Make the part level feature or geometry level designated parameters appear as a component in an assembly BOM.

You can designate a parameter using one of the following methods:

- Click **Tools > Designate**.
- Click **Edit > Setup > PART SETUP > Designate**.

The **Parameters Table** tab that is selected by default, displays or lists the properties of the designated parameters.

To add attributes of a designated item such as dimensions or detail items, click the

Attributes tab and click .



You can add designated items such as a Feature, Curve, Surface, Edge, Composite Curve, Quilt, or Annotation Element to the list of designated objects by clicking the **Objects** tab.

Arranging Columns in the Designate Dialog Box

You can manipulate columns in the **Designate** dialog box as follows:

- Add or remove columns.
- Sort the listed parameters in ascending or descending order. To sort, click the header of the column whose contents are to be sorted. For example, you can sort common parameters by type. The arrow next to the column header indicates the order of sorting, that is, ascending or descending.

To add a column:

1. In the **Designate** dialog box, click  to open the **Parameter Table Columns** dialog box.
2. Click the column name, for example, **Description**, and click .
3. Click **OK**. The column is added to the **Designate** dialog box.

Filtering Parameters

Using **Filter By**, you can filter the parameters by the following main categories :


- **All**
- **User Defined ONLY**
- **PDM ONLY**

- **Designated ONLY**
- **Type**
- **Access**

Note: The **Sub Items** option is available only on selecting **Type** and **Access**.

You can hide or show the filter by clicking **Edit > Hide Filter** or **Edit > Show Filter**, respectively.

Searching Parameters

Use  to search for parameters either by name or value. You can select the following parameter types and values in the **Search** dialog box:

- **Name**
- **Type**
- **Value**
- **Unit**
- **Designated**
- **Access**
- **Source**
- **Description**
- **Restricted**

You can search for parameter types by typing the first letter or the first few letters of its value followed by an asterisk (*). For example, `NAME = len*` displays parameter names starting with the letters `len` in the **Designate** dialog box.

Selecting Parameters

You can select multiple parameters and perform the following operations:

Operation	Condition	Result
Delete	Not applicable if a parameter is locked.	Deletes all selected parameters.
Designate	Applicable if at least one or more parameters are not designated.	Designates all selected parameters.
Undesignate	Applicable if at	Undesignates all

	least one or more parameters are designated.	selected parameters.
--	--	----------------------

Units in Parameters and Relations

About Units in Parameters

You can assign units to new user-defined parameters of the type Real Number. Existing parameters are considered unitless.

Note: You cannot change a unitless parameter into a parameter with units.


You can show units in the BOM and Pro/REPORT by adding units to the report table. Parameter units for each designated parameter are available to the PDM system such as Pro/INTRALINK, Windchill, or PDMLink.

Consider the following information on the use of units:

- Units are not supported for layout and drawing parameters.
- The system parameters as well as alternate and reported parameters are derived from the current model units.

To Specify Units for a Parameter

By default, a new parameter is created unitless.

1. In the **Parameters** dialog box, click  to add a parameter.
2. If the **Units** column is displayed, click the cell under **Units** and select a unit from the list.

Tip: To select from a refined list of units, do the following:

- a. Right-click in the **Parameters** dialog box and click **Insert Units** from the shortcut menu.
- b. From the **Select Units** dialog box, select the measure type under **Type**, select the unit from the filtered list, and then click **Select**.

About Changing Parameter Units

You can change the units of a parameter if access to that parameter is not locked. To change the units of a newly created parameter, select the required units from the **Units** list for the same type of measure. For example, you can change cm to ft . Use the **Changing Parameter Units** dialog box to specify if you want to convert or interpret the parameter value. **Interpret Value** is the default.

Note: When changing the unit of a parameter, click the **In the future, do not show this dialog** check box in the **Changing Parameter Units** dialog box if you do not want the dialog box to open again in that session of parameter modification.

You can specify the general rule for handling parameter values by selecting one of the following options in the **Changing Parameter Units** dialog box.

- **Convert Value (for example 1 "becomes 25.4mm)**—Change the parameter unit and scale the parameter value automatically.
- **Interpret Value (for example 1 "becomes 1mm)**—Change the parameter unit without scaling the value.

Effect of Changing the Model System of Units

Changing the model system of units affects system, application, and user-defined parameters that use model units. When you change the system of units, parameter values are not scaled automatically.

You can specify the general rule for handling parameter values by selecting one of these options in the **Model** tab of the **Changing Model Units** dialog box:

- **Convert dimensions (for example 1 "becomes 25.4mm)**—Change the parameter units and scale the values.
- **Interpret dimensions (for example 1 "becomes 1mm)**—Change the parameter units without scaling the values.

In addition, you can set the rule individually for each parameter by clicking the **Ignore** option. Clicking **Ignore** leaves both the parameter units and its value unchanged.

About Units in Relations

By default, relations are unit-sensitive. Unit-sensitive relations behave as follows:


- If you mix parameters with different units in a relation, the system converts them in order to solve a relation. For example, if the system units are mm, and you have a parameter *B* with the value of 1 inch, the relation $d1=B$ is evaluated as $d1=25.4\text{mm}$.
- You can mix unitless parameters and parameters with units in the same relation. For example, if you have a unitless parameter *A* and parameter *B* with units, you can have a relation: $d3=B*A$. In this case, *A* is used as a scalar value. You can also include a unitless parameter and specify units directly in the relation. For example, $d3=A[\text{mm}] + B$.

Note: Relations created in Pro/ENGINEER Wildfire and previous releases are evaluated as unitless. After you change relations, they become unit-sensitive.

Checking for Units in Relations

You can set the configuration option `relations_units_sensitive` to check for units when solving relations. This option has the following values:

- `yes` (default)—Solve relations taking units into account. If units are missing, the system issues a warning.
- `no`—Ignore units in relations.

You can control whether relations for the selected object (listed in the Relations editor) are unit-sensitive by clicking  or clicking **Utilities > Unit Sensitive** in the **Relations** dialog box.

Note: You can set the `relations_units_sensitive` configuration option to `no`, and then apply **Unit Sensitive**. This prevents the system from issuing error messages if units are missing.

The **Verify** command checks for validity of relations from the relations perspective and issues an error if units are contradictory. You can set the configuration option `relations_num_const_units` to issue a warning if units are missing in a relation. If the option is set to `yes`, the system checks for units, issues a warning, and prompts you to fix the relation by applying units.

Note:

- If you want to add a relation to nonsolid models, for example, bulks, you must always specify units for numeric constants.
- The setting of the `relations_num_const_units` configuration option is ignored for nonsolid models.

To Specify Units in Relations

You can specify units for numeric constants directly in the Relations editor of the **Relations** dialog box by enclosing the units within brackets. For example, $A=5[\text{mm}]$.

You can also use the **Specify Units** command to do the following tasks:

- Specify units for a new parameter by adding units directly in the Relations editor.
 - Define units for a numeric constant or unitless parameter.
1. In the **Relations** dialog box, type an equation in the editor. For example, $A=5$.
 2. Right-click and click **Specify Units** from the shortcut menu.
 3. Select the units from the **Select Units** dialog box. The units are added to the equation.

Example: Including Units with Relations

This example uses the following parameters and dimensions:

Parameter or dimension	Units
A	unitless
B	inch
d1	mm
d2	mm

The following table lists sample relations that use parameters with and without units.

Note: You can declare units on the left side of a relation only for newly created parameters.

Sample Relation	Verification status	Comments
$d1=d2$	OK	
$d1=d2*A$	OK	
$d1=B$	OK	Value of B is converted to mm. For example, if $d2=1$ and $B=1$, $d1=25.4$.
$d1=d2+B$	OK	
$d1=5$	Error	This relation is not treated as an error if <code>relation_num_const_units</code> is set to <code>no</code> .
$d1=5$ [mm]	OK	
$A=d1$	Error	This relation is not treated as an error if <code>relation_num_const_units</code> is set to <code>no</code> .
A [mm] = $d1$	Error	A is a unitless parameter and cannot accept units on the left side of the equation.
$d1=A$	Error	This relation is not treated as an error if <code>relation_num_const_units</code> is set to <code>no</code> .
$d1=A$ [mm]	OK	
$d1=A$ [inch]	OK	A is converted to mm. If $A=1$, then $d1=25.4$.
$d1=d2+5$	Error	This relation is not treated as an error if <code>relation_num_const_units</code> is set to <code>no</code> .
$d1=d2+5$ [mm] + .2 [cm]	OK	Units are converted to mm.
$A=d1+d2$		This relation is not treated as an error if

Sample Relation	Verification status	Comments
		relation_num_const_units is set to no.

Restricted Value Parameters

About Restricted Value Parameters

You can create restricted value parameters that use definitions from a restriction definition file. This file defines the parameter name and type, and optionally, the value of the parameter or a range of values for the parameter, or a list of values for the parameter. After you create restricted value parameters, their definitions exist in the model independently of the parameter file.

If the restriction definition file changes, you can generate a report to investigate if parameter definitions in the model are different from those in the restriction definition file. You can then update parameter definitions in the model by using the modified restriction definition file.

Note: Drawing parameters can also be of restricted value type.

To create restricted value parameters, click **Tools > Parameters**. You can also add or edit restricted value parameters directly on the Model Tree.

When using the **Parameters** dialog box, you can create a restricted parameter in one of the following ways:

- Type the appropriate parameter name. Pro/ENGINEER recognizes that parameter as a restricted value parameter.
- Select a parameter under **Restricted** to indicate that the parameter is restricted, and select a parameter name from the list or type a name.
- Type the first letter or first few letters of the parameter name. All parameters from the parameter table that match the typed letter or letters are displayed in the **Name** box. Select the required parameter name and press ENTER. You are prompted to define the value of the parameter.
- Type the first letter or first few letters of the parameter name. If a match is not found for the typed letter, the restricted parameter value list is not available and you are notified that it is an invalid parameter name when you press ENTER.
- Define the parameter value. As you type the first letter of the parameter name, the first available value for the defined restricted parameter that matches the typed letter is displayed in the **Value** box. Select the required parameter name and press ENTER. A default value is assigned to the parameter. You can change the parameter value, if required.

After you create a restricted value parameter, you can change its value by selecting a different value from the predefined (allowed) values. If you enter an incorrect value, you get an error message that also gives the valid range for this parameter.

Setting up Pro/ENGINEER Environment for Restricted Value Parameters

To load the restriction definition file, specify the path to that file by setting the `restricted_val_definition` configuration option.

Note: The restriction definition file is loaded only when Pro/ENGINEER starts.

Applications of the Restricted Value Parameters

You can use restricted value parameters in relations.

Note: If a value of a restricted value parameter changes and becomes invalid, the relation where this parameter is used reports an error. If the assigned value does not satisfy the parameter restriction, the parameter value is not changed.

You can include a restricted value parameter in Family Tables, an Inheritance feature, a user-defined feature, an Annotation feature, Program, and Data Sharing Features.

Note: In case of a conflict in definitions between the UDF and the model, the newly placed UDF takes on the model's definition, but retains its present value. If the parameter brought in by the UDF is marked as invalid, you can edit the value of the new parameter and set it to a valid value defined by the current model.

If a restricted value parameter is defined as Designated, it is available to the PDM system. Only the parameter name, type, and its current value are passed to Pro/INTRALINK. A list of valid parameter values is not passed to Pro/INTRALINK.

Properties of the Restricted Value Parameters

When working with parameters, you can use the **Restricted** column in the **Parameters** dialog box to identify the restricted value parameters. The type and optionally the value of the parameter or a range of values for the parameter, or a list of values for the parameter are assigned based on the name.

Note: The names of the restricted value parameters are reserved for restricted value parameters only and cannot be given to regular parameters.

The restricted value parameters can be of the following types:


- String
- Integer
- Real Number
- Boolean

For Real Number and Integer parameter types, you can select a parameter value from a range of values.

The restricted value parameters can have units or be unitless. You can also include comments with the parameter definition.

Checking the Parameter Status Using the Restricted Column

The **Restricted** column can have three states:

- Empty—Not restricted.
- Check mark—The current value complies with the restriction.
- Check mark with —The current value is invalid.

About a Restriction Definition File

The restriction definition file defines parameters using the following format:

```
{Name=<name>
Type=<type>[quantity_type]
[Default=<value>]
[Range=<range> | Enum=<enum>]
[Access=<access>]
}
```

Note: All data in brackets [] are optional. The vertical bar indicates a choice between two or more variants.

When you apply definitions, the system updates model definitions using the restriction definition file.

To apply definitions, specify the directory path to restriction definition file by setting the configuration option `restricted_val_definition`.

Note: The restriction definition file is loaded only at Pro/ENGINEER startup.

Syntax for Creating a Restriction Definition File

A restriction definition parameter file has the following format:

```
ND_ParamDefArr_K01 = {
    <definition>
    [, ...]
}
```

where:

[, ...]—Optional continuation of the list.

Format for <definition> is:

```
{
```

```

Name = <name>

Type = <type> [<quantity type>]

[Default = <value>]

[Range = <range> | Enum = <enum>]

[Access=<access>}

}

```

Note: All data in brackets [] are optional. The vertical bar | indicates a choice between two or more variants.

where:

<name>—Any valid name of a Pro/ENGINEER parameter.

<type>—Uses the following format:

```
integer | real | string | boolean
```

<quantity type>—Either a name of any supported by Pro/ENGINEER basic quantity types (physical dimensions), for example, Length or Force, or an expression composed from such names with operations of multiplication (*), division (/), or power (^).

Format for <value> is:

```
<value_int> | <value_real> | <value_string> | <value_boolean>
```

where:

<value_int>—An integer.

<value_real>—Any real number valid in Pro/ENGINEER relations, optionally followed by a <unit>.

<value_string>—Any string enclosed by double quotes. **Note:** A quote in a string must be presented by the sequence \' and a backslash must be presented by \\\).

<value_boolean>—True | False.

<unit>—Either a name of any supported by Pro/ENGINEER measurement unit (for example, mm, lb, or N) or an expression composed from such names with operations of multiplication (*), division (/). Units of all values must match the quantity type of a parameter definition in which they occur as default or restricted values.

Format for <range> is:

```
[<value>,<value> | <value>, | ,<value>]
```

where:

variant <value>—A range with a minimum limit only.

variant ,<value>—A range with a maximum limit only.

Note: Use [] or () to define the range:

[]—The corresponding minimum or maximum value is included in the range.

()—The corresponding minimum or maximum value is excluded from the range.

Format for <enum> is:

```
{
    <value>
    [, ...]
}
```

Format for <access> is:

```
full | limited | locked
```

where:

full—Indicates full access user-defined parameters that can be modified anywhere.

limited—Indicates parameters cannot be modified by a relation. Restricted parameters can be modified only by Family Tables and Program.

locked—Indicates parameters were created by an external application (Data Management System, Analysis features, relations, Program, or Family Tables). Locked parameters can be modified only from within the external application.

Note:

- Spaces and new lines are optional. They are allowed everywhere with the exception of keywords and values.
- Comments starting with either // or /* and ending with a new line are allowed at any place of the file.

Example: Sample Restriction Definition File

The following parameter file list1.lst defines parameters with the names `paint`, `color_1`, `color_2`, `num_of_coats`, `grade`, `num_1`, `num_2`, `Bool_1` (the names are shown in bold in the file).

```
ND_ParamDefArr_K01 = {
    { Name = color_1
      Type = string
      Default = 'blue_new'
      Enum = { 'red_new', 'green_new', 'blue_new' }
    },
    { Name = color_2
```

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```
Type = string
Default = 'white_new'
! DEFAULT IS WHITE_NEW
Enum = { 'yellow_new', 'white_new', 'black_new' }
},
{ Name = num_of_coats
  Type = integer
  Default = 2    ! correct?
  Range = [ 0 , 9 ] // 0<=Value<=9
},
{ Name = grade
  Type = integer
  Default = 35   ! correct?
  Range = [ 26 , 50 ] // 26<=Value<=50
},
{ Name = num_1
  Type = real
  Default = -5.0
  Range = [ -10.0 , 0. ]
},
{ Name = num_2
  Type = real
  Default = 22.5
  Range = [ 21.00 , 30.0 ]
},
{Name=num_3
Type=integer
Default=75
Enum={25,50,75,100}
},
{Name=num_4
```

```

Type=real
Default=50.5
Enum={25.1,50.5,75.45,100.123}
Access=full
}
{ Name = Bool_1
  Type = boolean
},
!!! ADD MORE RESTRICTED VALUE PARAMS HERE
}

```

To Create a Restricted Value Parameter

1. Specify the location of the restriction definition file by setting the `restricted_val_definition` configuration option.
2. Click **Tools > Parameters**. The **Parameters** dialog box opens.

3. Click .

4. Locate the column **Restricted** and click the cell.
5. Click the **Name** cell and either type a name or select a name from a list of predefined parameter names. The type of the parameter is automatically selected.

Note: If you add a parameter whose name matches the name of a parameter in the table-restricted parameter table, a warning message appears and the parameter is not added.

6. Specify the parameter value by typing a valid value or selecting a value from a list.

Tip: If the parameter has a range of values, you can investigate the valid range either by entering any value in the **Value** box and obtaining the message for the valid range, or by clicking **Properties** and then clicking the **Definition** tab in the **Properties** dialog box.

To Update Parameter Restriction Definitions in the Model

1. Ensure that the path to the restriction definition file is specified as the value of the `restricted_val_definition` configuration option before starting the Pro/ENGINEER session.
2. Click **Tools > Parameters**. The **Parameters** dialog box opens.

3. Click **Tools > Conflict Report**. The report identifies the difference between the definition of the restricted parameters in the model and the restriction definition file.
4. If a conflict is reported, click **Tools > Update Restriction Definition**.
In case of no conflicts, you are notified that no conflicts were found between the model definition and restriction definition file.
5. Select the check box in the confirmation message when prompted for confirmation that updating restriction definitions may cause some existing parameter values to become invalid and click **Apply** to update parameter restriction definitions.
6. If required, click **Report** to view the conflict report.

Generating Reports on Restricted Value Parameters

You can generate two types of reports: Conflict report and Invalid Values report.

Conflict Report

The Conflict report checks whether the restricted value parameters in the model are in agreement with an external file. If the external file has changed and the definitions of the parameters in the model are old, the system identifies the parameters in conflict.

To generate the Conflict report, click **Tools > Conflict Report** in the **Parameters** dialog box.

Invalid Values Report

The Invalid Values report identifies parameters with invalid values and generates two lists:

- Results of comparing the parameter values with parameter definitions in the model.
- Results of comparing the parameter values with parameter definitions in the restriction definition file.

To generate the Invalid Values report, click **Tools > Invalid Values** report in the **Parameters** dialog box.

Syntax for Creating an External Definition File for Table-restricted Parameters

The external definition file uses the following format to define parameters:

```
{Name=<name>
```

```
Type=<type>[quantity_type]
```

```
[Default=<value>]
```

```

Access=<access>
TABLE=<table-name_1>,<table-name_2>,...,<table-name_n>
}

```

Note: All data in brackets ([]) are optional. The vertical bar (|) indicates a choice between two or more variants.

When you apply definitions to parameters, Pro/ENGINEER uses the external definition file to update the parameters of the model. To apply definitions, specify the path to the restriction definition file by setting the `restricted_val_definition` configuration option.

Note: The external definition file is loaded only at Pro/ENGINEER startup.

Sample External Definition File

The following parameter file, `list1.lst`, defines parameters with the names `string_par1`, `int_par1`, `real_par1`, `real_par2`, `bool_par1`, respectively.

```

ND_ParamDefArr_K01 =
{ Name = string_par1
  Type = string
  // this is double slash comment
  Table = Table1
},
{ Name = int_par1
  Type = integer
  ! this is an exclamation mark comment
  Table = Table1
},
{ Name = real_par1
  Type = real
  Table = Table1
},
{ Name = real_par2
  Type = real
  Table = Table1
},
{ Name = bool_par1

```

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```

Type = Bool
Table = Table1
}

!!! ADD MORE RESTRICTED VALUE PARAMS HERE

ND_ParamTable_K01 = {

```


NAME =	TABLE-1					
HEAD =		string_par_1,	int_par1,	real_par1,	real_par2,	bool_par1
ROWS =	't1_row1- m4-120',	'M4',	120,	1.1,	2.3,	true
	't1_row2- m5-130',	'M5',	"(, 200) 130",	"[12.25,] 22.65",	"[12.25,] 24.65",	false
	't1_row3- m6-135',	'M6',	"(-100, 300) 135",	"[123.55, 225.5) 200.50",	"[123.55, 225.5) 202.50",	TRUE
	't1_row4- m8-140',	'M8',	"[-200,] 140",	"(2234.579) 1555.75",	"(2234.579) 1557.75",	FALSE
	't1_row5- m9-150',	'M9',	"[-300, 500) 150",	"(- 12345.50, 22345.56] -175.0",	"(- 12345.50, 22345.56] - 173.0",	true
	'T1_Row6- M10-155',	'M10',	155,	FREE 5.675,	FREE 7.675,	false
	'T1_ROW7- M12-6',	'M12'	FREE 6,	NULL,	NULL,	NULL
RELATIONS = int_par1 > 1 real_par2 > real_par1 + 1.1						

```

}
```

To Create Table-restricted Value Parameters

1. Click **Tools > Parameters**. The **Parameters** dialog box opens. You can choose to select a restricted value parameter or a table restricted parameter.

2. Click  to insert a restricted value parameter.
 - a. Under the **Restricted** column, click the cell for a parameter to make the parameter a restricted parameter.
 - b. Click the **Name** cell and either type a name or select a name from a list of predefined parameter names. The type of the parameter is automatically selected. The list contains the following:
 - Parameter names in the restriction definition file that are not table-restricted.
 - Table names (set labels) that are declared in the restriction definition file.
 - c. Select a table name from the list. The **Select Parameter Set** dialog box opens displaying the contents of the selected table.

Alternatively, click **Parameters > Add Restricted Parameter Set** in the **Parameters** dialog box to insert a table restricted parameter set directly. The **Select Parameter Set** dialog box opens displaying the contents of the selected table.

3. Select any cell in the restricted parameters table to add the corresponding parameter set. All parameters that are defined for the selected set or row are automatically created and assigned the corresponding values.

If you select the parameter name, the table is sorted based on the values of the corresponding parameter. If the parameter values are defined by a range, the values are sorted by the default value.

4. Click **Apply** to add all defined set parameters and set labels with values defined in the selected set or row.

Note: The parameters with a null value are also added.

5. Click **OK**.

Note: You can use the external parameter definition file to automatically create Annotation Element parameters. The Annotation Element parameters can either be regular or restricted parameters if they are defined in the restriction definition file.

To Delete Table-restricted Parameters

1. Select a table-restricted parameter or any set label parameter that you want to delete and click **Parameters > Delete Parameter(s)**. The **Modify Parameter** dialog box opens.
2. Click **Yes**. The selected table-restricted parameter or all parameters within the set are deleted.

To Modify Table-restricted Parameters

To Modify a Table-restricted Parameter Whose Value is not Defined by a Range or a Set Label

1. Click **Tools** > **Parameters**. The **Parameters** dialog box opens.
2. Select the table-restricted parameter from the parameter set that you want to modify.
3. Right-click and click **Open Table**. The **Select Parameter Set** dialog box opens and the corresponding parameter of the restriction table is automatically selected or highlighted.
4. Select another parameter set and click **Apply**.

To Modify a Table-restricted Parameter Whose Value is Defined by a Range and the New Value is Within the Same Range

Select the respective parameter cell under the **Value** column in the **Parameters** dialog box and enter a new value.

To Modify a Table-restricted Parameter Whose Value is Defined by a Range and the New Value is not Within the Same Range

1. Select the table-restricted parameter from the parameter set that you want to modify.
2. Right-click and click **Open Table**. The **Select Parameter Set** dialog box opens and the corresponding parameter of the restriction table is automatically selected or highlighted.
3. Select another parameter set with the required range and click **Apply**. The **Select Parameter Set** dialog box closes.
4. Select the respective parameter cell under the **Value** column in the **Parameters** dialog box and type a new value.

To Modify a Set Label Parameter

1. Select the table-restricted parameter from the parameter set that you want to modify.
2. Click the arrow adjacent to the parameter cell under the **Value** column in the **Parameters** dialog box. A list of set label values with the **TABLE** option appears. The current value of the set label is highlighted in the list.
3. If you select **TABLE**, the **Select Parameter Set** dialog box opens.
4. Select another set label value from the list and click **Apply**. The values of all the associated parameters are updated accordingly.
5. Select another parameter set.

Note: If some parameters existed in the original parameter set but not in the newly defined parameter set, these parameters in the original parameter set are retained but their value is changed to null.

To Show Restricted Parameters

1. Click **Tools > Parameters**. The **Parameters** dialog box opens.
2. Click **Show > Restriction Tables**. The **Restriction Tables** dialog box opens and lists parameters defined in the restriction definition file.

Tip: Converting Restricted Parameters to Table-Restricted Parameters

While converting a restricted parameter to a table-restricted parameter in a model, consider the following:

- If a feature contains a restricted parameter, then you can add the same parameter with a new table-restricted definition to a different feature.
- If a feature contains a restricted parameter and you add a new table-restricted parameter with the same name, a message appears stating that a parameter with that name is already defined in the feature. To add a new table-restricted parameter, you must either delete the original restricted parameter or update the restriction definition file.
- If you update the restriction definition file in a model that has original restricted parameters, the **Select Parameter Set** dialog box opens and you are prompted to select a parameter set. All parameters in the selected table-restricted parameter set are automatically added to the parameters table.

Annotation Element Parameters

About Annotation Element Parameters

You can define parameters for Annotation Elements of an Annotation feature. Annotation Element parameters can be of the following types:

- String (user-defined or restricted value)
- Real number (user-defined or restricted value)
- Integer (user-defined or restricted value)
- Boolean (user-defined or restricted value)

You can list Annotation Element parameters in the Model Tree just like other model parameters.

Use one of the following methods to create Annotation Element parameters:

- Add Annotation Element parameters by specifying parameter properties in the **Parameters** dialog box. You can access the **Parameters** dialog box from the **ANNOTATION FEATURE** dialog box. In addition, you can edit Annotation

Element parameters by clicking **Tools > Parameters** and changing the selection in the **Look in** box of the **Parameters** dialog box to **Annotation Element**.

- Use an external file to create Annotation Element parameters automatically each time you create an Annotation Element.

Note:

- You can select and highlight the Annotation Element parameters and its values in the parameter table.
- You can move the defined Annotation Element parameters and its values vertically. Use the SHIFT or CTRL key to select and highlight the Annotation Element parameters and move one or more rows in the up or down direction by clicking the arrows located at the right of the parameters table in the **Parameters** dialog box.
- After repositioning specified parameters, you can save the newly ordered parameter list for the table for every session, and when saving the model. The parameter position information is retained only on saving the model.
- You cannot reorder parameters in a filtered state.
- The parameter order is propagated and maintained when you propagate an Annotation Feature (AF) through a UDF or a Copy or Paste operation.
- The parameter order is propagated and maintained when you propagate an Annotation Element through the Annotation Element Propagation Feature or through a DSF.
- If an Annotation Element is dependent, then parameter reordering is not allowed for child Annotation Elements.
- Parameter reordering is applicable for the following parameter items:
 - Main
 - Alternate Mass Properties
 - Reported Mass Properties

Including Annotation Element Parameters in a Note

To include the Annotation Element parameters in a note, use the following syntax:

```
&<param_name>:NID_<annotation_element_name>
```

Simultaneous Editing of Multiple Annotation Element Parameters

You can simultaneously create, edit, or delete common parameters for multiple Annotation Elements.

To select objects for the simultaneous parameter editing, select several Annotation Elements, and then use the **Edit Parameters** command on the shortcut menu.

About Automatic Creation of Annotation Element Parameters

You can automatically create Annotation Element parameters for each new Annotation Element by using an external parameter file. The external file follows format similar to that of the restriction definition file.

To set up the automatic parameter creation, set the configuration option `auto_ae_param_file` by specifying the directory path to the external file. Thereafter, the Annotation Element parameters are added automatically each time you create an Annotation Element.

Note: You cannot define parameters as restricted value parameters using automatic parameter creation for Annotation elements. To define parameters as restricted, use a restriction definition file.

The external file defines the following parameter properties:

- Required properties:
 - Name
 - Type
 - Default
- Optional properties:
 - Unit—If not specified, the parameter is assumed unitless.

Note: Units can only be defined at a parameter creation. You cannot add units later.
 - Access—If not specified, access is assumed Full.

The following properties cannot be specified in the parameter file:

- Description
- Owner
- Restriction

Note: If a parameter description is in conflict with the restrictions, the parameter is not created.

Example: Automatic Creation of Annotation Element Parameters

Create a text file with the following content:

```
!Default parameters for Annotation Elements /this is a comment/

ND_ParamDefArr_K01 = {
  { Name = Version
    Type = integer
    Default = 2
  },
  { Name = Color
    Type = string
  }
}
```

```

        Default = 'green'
    },
    { Name = Ratio
      Type = real
      Default = 0.5
    }
}

```

Save the file with the name `auto_params.lst` in the folder `/shared_data/annotations`.

Set the configuration option as follows:

```
auto_ae_param_file /shared_data/annotations/auto_params.lst
```



Now, every time you create an Annotation Element, the system automatically adds to it the following parameters:

Parameter Name	Type	Value
Version	Integer	2
Color	String	green
Ratio	Real	0.5

You can later manipulate these parameters like any regular parameters: search for them, modify their values individually or simultaneously for multiple Annotation Elements, and so on.

To Define Annotation Element Parameters

You can define Annotation Element parameters while defining Annotation Elements. You can also add parameters later by editing the definition of the Annotation feature. Alternatively, you can define Annotation Element parameters by accessing them directly through the **Parameters** dialog box.

1. In the **Annotation Feature** dialog box, click an Element for which you want to define a parameter.
2. Click **Parameters**. The **Parameters** dialog box opens.
3. Click  to add a parameter. After you have defined parameter properties, click **OK**.
4. Click **Tools > Parameters**. The **Parameters** dialog box opens.
5. To define the object type, click **Annotation Element** under **Look in**.
6. Select an Annotation Element for which you want to define parameters.
7. Click  to add a parameter. After you have defined parameter properties, click **OK**.

Family Tables

Basics of Family Tables

About Family Tables

Family Tables are collections of parts (or assemblies or features) that are essentially similar, but deviate slightly in one or two aspects, such as size or detail features.

For example, wood screws come in various sizes, but they all look alike and perform the same function. Thus, it is useful to think of them as a family of parts. Parts in Family Tables are also known as table-driven parts.

The following figure shows a family of bolts. The generic is at the top of the figure, and its instances are underneath. The generic is the parent.



Using Family Tables, you can:

- Create and store large numbers of objects simply and compactly
- Save time and effort by standardizing part generation
- Generate variations of a part from one part file without having to re-create and generate each one
- Create slight variations in parts without having to use relations to change the model
- Create a table of parts that can be saved to a print file and included in part catalogs

Family Tables promote the use of standardized components. They let you represent your actual part inventory in Pro/ENGINEER. Moreover, families make it easy to interchange parts and subassemblies in an assembly, because instances from the same family are automatically interchangeable with each other.

Family Table Structure

Family Tables are essentially spreadsheets, consisting of columns and rows. They consist of the following three components:

1. The base object (generic object or generic) on which all members of the family are based.
2. Dimensions and parameters, feature numbers, user-defined feature names, and assembly member names that are selected to be table-driven (hereafter referred to as items).
3. Names of all family members (instances) created by the table and the corresponding values for each of the table-driven items.

Rows contain instances of parts and their corresponding values; columns are used for items.

The column headings include the instance name, and the names of all of the dimensions, parameters, features, members, and groups that were selected for the table. Dimensions are listed by name (for example, `d9`) with the associated symbol name (if any) on the line below it (for example, `depth`). Parameters are listed by name (dim symbol). Features are listed by feature number (for example `F107`) with the associated feature type (for example `[cut]`) or feature name on the line below it.

The generic model is in the first row in the table. The table entries belonging to the generic can be changed only by modifying the actual part, suppressing, or resuming features; you cannot change the generic model by editing its entries in the Family Tables.

Note: Family Table names are not case-sensitive. Therefore, any subsequent references to inserted names show them in uppercase letters.

For each instance, you can define whether a feature, parameter, or assembly name is used in the instance either by indicating whether it is present in the instance (`Y` or `N`) or by providing a numeric value (in the case of a dimension). All dimension cells must have a value, either a number or asterisk (`*`) to use the generic's value.

All aspects of the generic model that are not included in the Family Table automatically occur in each instance. For example, if the generic model has a parameter called `Material` with a value `Steel`, all instances will have the same parameter and value.

You can scroll horizontally through a Family Table to see additional information. The Instance Name column remains visible as you scroll.

Family Table functionality varies with your Pro/ENGINEER module licenses.

Module	Available Family Table Functionality
Basic Pro/ENGINEER	Create table-driven parts by adding dimensions to the Family Table
Assembly	Create table-driven assemblies by adding to the Family Table subassembly and part names, as well as assembly dimensions.
Pro/FEATURE	Create table-driven user-defined groups whose group feature dimensions can be table-driven,

	invariable, or variable. Add table-driven groups to a part Family Table.
--	---

About Multi-Level Family Tables

A Family Table is not restricted to a single level of a table. You can create instances of instances by selecting an instance and creating a new table for it, using **Insert > Instance-Level Table** in the **Family Table** menu bar. When you must do this depends on what you intend to model using Family Tables. The Pro/ENGINEER BASIC Library is itself constructed of several levels of tables.

You can also add new features or components directly to an instance. This facilitates the construction of multi-level Family Tables (families with subfamilies or instances with instances). The new feature or component appears in the instance in which it is added and in all subsequent instances of that instance. The feature or component is suppressed in other instances at the same level of the table and in generics at higher levels.

1. Create a generic model with the desired base features.
2. Create a Family Table with instances designating the subfamilies to be created.
3. Regenerate an instance and add features unique to its family. Repeat this until all levels of the family are completely defined.

When Pro/ENGINEER creates a family instance, only those features that were created in a direct line from the top-level generic may be present. This means that some features may need to be re-created, because the instance is unaware that similar features exist in another (parallel) subfamily of the table.

Features from a parallel subfamily can be temporarily resumed within a particular instance and used, for example, as a guide to sketch a similar feature. Upon regeneration, however, the Family Table resuppresses the resumed features. Similarly, suppression of a feature only lasts until the next regeneration, when the Family Table resumes it.

When you create a feature that refers to the geometry of a resumed feature used in another instance, the new feature belongs to the other instance. It is best to create the sections of features that are common to different branches of the family in Sketcher mode and store the sections for later use.

You cannot redefine features added to an instance. If you cannot achieve the desired feature shape by modifying dimensions, you must delete the feature and start over. Modifying or deleting a feature affects all children of the instance, regardless of the instance from which the feature was modified or deleted.

About User-Defined Feature Family Tables

User-defined feature (UDF) Family Tables differ slightly from part or assembly tables. The following restrictions apply when working with UDF Family Tables:

- You can not retrieve instances of a UDF by themselves unless you are placing the UDF on a part. In that case, when you select the UDF generic name, the **Select Instance** dialog box opens. You can then select an instance to use.
- You can create a UDF Family Table at the time you define or modify the UDF, using the definition dialog box for the new UDF.
- You cannot retrieve a UDF instance into a separate window.
- You cannot verify a UDF instance.
- When you place a UDF defined with a Family Table, any suppressed features in the UDF instance are not placed on the part. These features are in effect deleted as the UDF group is placed.
- When you place a legacy UDF having local groups and a Family Table defined in it, the geometry may differ if the Family Table contains group members in the item list. To avoid this, update the Family Table to remove the suppressed members of the group from the Family Table items list. You can then add the group header as a new column.


Note: In Wildfire 4.0, you can suppress individual group members without suppressing the entire group. You can place the UDF instance such that only the specified member of a given group is suppressed.

Creating Family Tables

To Create a Family Table

1. Create a generic model that will serve as an original.
2. Click **Tools > Family Table**. The **Family Table** dialog box opens.

The model should have no rows for design variations.

3. Click . The **Family Items** dialog box opens.
4. Under **Add Item**, click the object type you want to add to the new variation.
5. Select the specific object you want to add using the **Select** dialog box.
6. Click **Done Sel**. The object is added under **Items**. Make sure the **Filter** box is checked to show the item.
7. Add all items by selecting them from the generic model. Click **OK**. You are returned to the Family Table. A "home" row is added containing the original object; new columns are added for each item you added.

8. Click **Insert > Instance Row**, or click . A new row is added for the first new instance.

9. Place the cursor in the empty cells and add the new values. You are in effect creating a new part with slightly different properties than the original. Cells for features may be set to Y or N , to indicate whether or not the instance contains the feature. If you want to inspect the instance, put the cursor anywhere in the instance row and click **Open**.

After an instance has been entered in the table, it can be retrieved and used like any other model.

To Create a Multi-Level Family Table

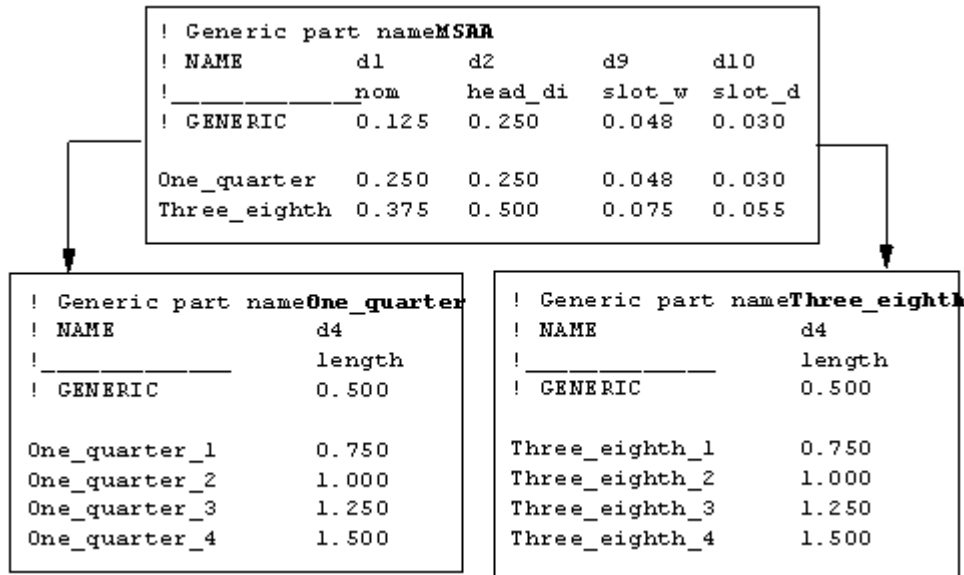
You can create a Family Table from a selected instance in an existing Family Table. The selected instance becomes the generic in the new Family Table.

1. Open a model that contains a Family Table.
2. Click **Tools > Family Table**. The **Family Table** dialog box opens.
3. Select the instance you want to use to create the new Family Table.
4. Click **Insert > Instance-Level Table**. A new Family Table is created. The selected instance becomes the generic of the new table.
5. Immediately add instance rows and columns to the new Family Table. If you click **OK** to close the Family Table without having created instance rows or columns, the table creation is cancelled.
6. Click **OK** when you are done creating the new Family Table. The system closes all Family Tables that were open at the time you created the new table.
7. To see how your original (top-level) Family Table was modified, reopen the **Family Table** dialog box. The instance upon which the new Family Table is based contains a folder icon in the **Type** column of its instance row.

To Modify a Family Table of an Instance

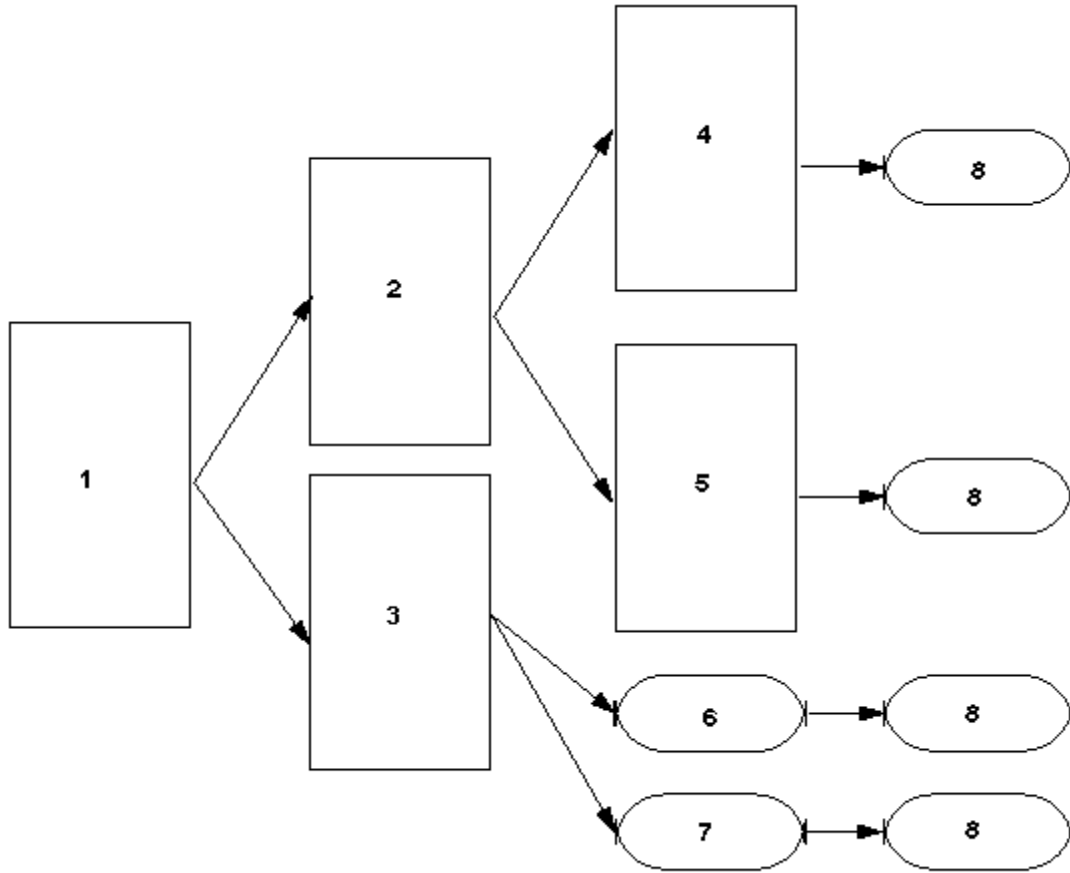
1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Do one of the following:
 - Under **Look In**, select an instance that has its own Family Table. The Family Table of the selected instance is displayed.
 - Under **Look In**, type the name of any instance. If the instance has a table of its own, this table appears in the dialog box. If the instance does not have a table of its own, then a table to which this instance belongs appears in the dialog box, with the instance selected.
3. Modify the current Family Table as desired, then click **OK** to close the Family Table and accept your changes.

Example: Several Levels of Family Tables



Example: Slot Added as a Feature

In the following figure, the slot is added as a feature to the F-Slotted instance of the fillister head screw. The slot feature appears in the Fillister Family Table and is automatically suppressed in both the Fillister and F-Phillips instances. Similarly, when the roundhead feature is added to the Round instance, it is placed in the generic Family Table, but is automatically suppressed in both the generic and the Fillister models.



1 Generic

2 Fillister

3 Round

4 F-Slotted

5 F-Phillips

6 R-Slotted

7 R-Phillips

8 Lengths

1 shows the top table level

2 and 3 show table level 2

4, 5, 6, and 7 show table level 3

8 shows table level 4


To Create or Modify a UDF Family Table

You can create a User-Defined Feature (UDF) Family Table when defining a new UDF or modifying a UDF.

1. Click **Tools > UDF Library**. The **UDF** menu appears.
2. Click one of the following:
 - **Create**—Creates a UDF and adds a Family Table (See *To Create a UDF in Part Modeling*)
 - **Modify**—Modifies or adds a Family Table in a UDF


The **UDF: <udfname>** dialog box opens.

3. After creating or selecting a UDF, click **Family Table** and then **Define** in the **UDF: <udfname>** dialog box.
 - **When** defining a new UDF or modifying an existing UDF that does not contain a family table, the **Family Table:<udfname>** dialog box opens prompting you to add a column or row.
 - **When** modifying a UDF that contains a Family Table, the **Family Table:<udfname>** dialog box opens with the existing Family Table information.

4. Click  to add table columns. The **Family Items, Generic: <udfname>** dialog box opens.
5. Under **Add Item**, select the type of item you want to add to the table and then select the respective item in the Pro/ENGINEER graphics window or on the Model Tree. The selected items appear under **Items**.

If you select **Parameter**, the **Select Parameter** dialog box opens.

1. Under **Look In**, select the type of object for which you want to add parameters.
2. Select the respective item in the Pro/ENGINEER graphics window or on the Model Tree, the item name and parameters appear in the **Select Parameter** dialog box.
3. Select a parameter you want to add and click **Insert Selected**. This parameter is added under **Items** in the **Family Items, Generic: <udfname>** dialog box and removed from the parameters table in the **Select Parameter** dialog box.
4. Click **Close** to close the **Select Parameter** dialog box.
6. After all items are added, click **OK** in the **Family Items, Generic: <udfname>** dialog box. The **Family Table:<udfname>** dialog box opens.


7. Click  to add instance rows and informational or parameter columns as necessary.

- Click **OK** to close the **Family Table:<udfname>** dialog box. The family table is defined or modified.
- Click **OK**. The **UDF: <udfname>** dialog box closes.


Adding Items to Family Tables

To Add an Assembly Member to a Family Table

You must have a license for Assembly to create Family Tables for assemblies and to use assemblies.


- Click **Tools > Family Table**. The **Family Table** dialog box opens.
- Click . The **Family Items** dialog box opens.
- Under **Add Item**, click **Component**. The **Select** dialog box opens.
- Select the components (parts and subassemblies) that you want to display in the Family Table. These components must include those that can be omitted or replaced in one or more instances of the family. All other components are included in every instance.
- Click **OK**.

To Add a Merge Part to a Family Table

- Click **Tools > Family Table**. The **Family Table** dialog box opens.
- Click . The **Family Items** dialog box opens.
- Under **Add Item**, click **Merge Part**. The **References** dialog box opens with a list of merge parts needed for the model.
- Select the merge part or parts from the list and click **Ok**.
- Click **OK** to close the **Family Table** dialog box.

Adding Dimensions to Family Tables

To Add a Dimension to a Family Table

- Click **Tools > Family Table**. The **Family Table** dialog box opens.
- Click . The **Family Items** dialog box opens.
- Under **Add Item**, click **Dimension**.
- Select the dimension you want to add to the Family Table. You can add regular dimensions, pattern dimensions, and geometric tolerances.

5. Click **OK**.

Note: You can scroll horizontally through a Family Table to see additional items. The Instance Name column remains visible.

Tip: Selecting Dimensions in a Family Table

Dimensions appear in the Family Table in the order in which they are selected.

If there are multiple features in the model, select all the dimensions from one feature before proceeding to the next. This makes it easy to add instance values and to maintain the table.

Note: If you cannot see all the columns in the Family Table, you can scroll horizontally through the table. The Instance Name column remains visible.

Tip: Entering Negative Dimension Values in a Family Table

When you enter a negative dimension value in a Family Table, Pro/ENGINEER interprets the negative value as a flip of the direction each time the feature is regenerated.


This is because the dimension value is considered to be "relative" by default. To avoid this, you must change the column to be considered as "absolute" values. You can switch the column between relative and absolute by selecting a cell in the column and choosing **Tools > Absolute / Relative Dimension Values** in the **Family Table** dialog box. The dimension name at the top of the column will be preceded by a dollar sign (\$) when the column is absolute.

To Switch Between Absolute and Relative Dimension Values in a Family Table

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Select the dimension column whose values you want to change from absolute to relative, or vice versa.
3. Click **Tools > Absolute / Relative Dimension Values**. The dimension values change from absolute to relative, or vice versa. The dimension column heading contains a dollar sign (\$) when the column values are absolute.

Note: The value of dimensions is considered to be relative by default.

To Add a Tolerance to a Dimension in a Family Table

1. Click **Tools > Environment**. The **Environment** dialog box opens.
2. Under **Display**, select **Dimension Tolerances**.
3. Click **Tools > Family Table**. The **Family Table** dialog box opens.
4. Click . The **Family Items** dialog box opens.
5. Under **Add Item**, click **Dimension**.

6. Select the dimensional tolerance you want to add to the Family Table. You must explicitly select each dimensional tolerance to add it to the Family Table.
7. Click **OK**.

Note: You can scroll horizontally through a Family Table to see additional columns. The **Instance Name** column remains visible.

Adding Parameters to Family Tables

About Adding Parameters to a Family Table

You must set the values of parameters for a Family Table by using either table-driven values or relations. However, ensure that you do not use both methods simultaneously for any single parameter.

For example, you can use a relation to establish the width of a model and a table-driven value to determine its length, but neither parameter must be determined by both a relation and a table-driven value.

If a parameter in a generic model is table-driven and evaluated by a relation, the value produced by the relation overwrites all values for that parameter in the Family Table. For family instances, however, the table-driven values for the parameter are not overwritten by the relation value until the instances are regenerated.

If you must use a table-driven parameter in a relation, ensure that you keep the table-driven value on the right side of the operator to keep its value from being reevaluated and overwritten by the relation.

You can add table-restricted parameters to Family Tables. On selecting a parameter, the entire parameter set is automatically selected. A Family Table can add and drive any parameter within the set, based on the conditions defined in the restriction definition file.

Prefixes and Hierarchy

A top-level parameter (one related to a part or assembly) is stored under its own name only, without any prefixes or suffixes, in the Family Table (for example, `parameter_name`).

A lower-level parameter (one related to a component or feature) is stored under its own name followed by `:FID_` and the feature ID or name of its parent (for example, `parameter_name:FID_fid`). For example:

- In the model (part or assembly), the model parameter called ABC would be stored under the name ABC.
- In an assembly, a particular component has a feature ID of 10. The associated component parameter DEF would be stored under the name `DEF:CID_10`.
- In the model (part or assembly), a particular feature has a feature ID of 23. The associated feature parameter HIJ would be stored under the name `HIJ:FID_23`.

- In the model (part or assembly), a particular feature has a feature name MYHOLE. The associated feature parameter KLM would be stored under the name KLM:FID_MYHOLE.

Note: In relations, if you want to add parameters of a model, features, components, or other items such as quilts, curves, edges, inheritance, or annotation elements using the **Relations** dialog box, click **Insert > From List**. The **Select Parameter** dialog box opens. Select the required parameter and click **Insert Selected**. The **Select Parameter** dialog box closes and the parameter with a correct suffix is displayed under **Relations**.

To Add a Parameter to a Family Table

1. Click **Tools > Family Table**. The **Family Table** dialog box pens.

2. Click . The **Family Items** dialog box opens.

3. Under **Add Item**, click **Parameter**. The **Select Parameter** dialog box opens.

Note: You can use the **Other** option to add lower-level parameters for entities such as edges, curves, quilts, annotation elements, surface, composite curve, or material. For example, to add a surface parameter to the family table, type the parameter name as PARAM1:SID_THESURF.

4. Under **Look In**, select one of the object types. The **Select** dialog box opens. Select the components or features whose parameters you want to add. As each one is selected, Pro/ENGINEER checks if the component or feature has any user-defined parameters that have not been added to the Family Table. If Pro/ENGINEER finds any such parameters, they are listed on the **PARAMETER** menu.

5. Select the parameters that you want to add and click **Insert Selected**.

Note: If you select a table-restricted parameter, the entire parameter set is selected. The set label parameter value determines the value of parameters in the set. Changing the set label parameter value may change the value of other parameters in the set for the Family Table. When adding a parameter to a Family Table, if the parameter is part of a set label parameter, it is good practice to add the set label parameter to the Family Table.


6. Click **Close** when you finish selecting the parameters. The **Select Parameter** dialog box closes.

7. Click **OK** in both the **Family Items** and **Family Table** dialog boxes.

To Add System Parameters to a Family Table

You can add system parameters to a Family Table. These parameters are standard parameters such as mp_density, provided by Pro/ENGINEER for part density.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.

2. Click . The **Family Items** dialog box opens.
3. Under **Add Item**, click **Parameter**. The **Select Parameter** dialog box opens.
4. Click the list above **Close** to select **Alternate Mass Properties** or **Reported Mass Properties**. **Main** is the default option. The corresponding system parameters are listed in the **Select Parameter** dialog box.
5. Select the parameters that you want to add and click **Insert Selected**.
6. Click **Close** when you finish selecting the parameters. The **Select Parameter** dialog box closes.
7. Click **OK** in both the **Family Items** and **Family Table** dialog boxes.

Adding Features to Family Tables

About Adding Features to Family Tables

Keep in mind the following when adding features to Family Tables:

- If you give the feature a NOTE name, the name appears at the top of that feature column in the Family Table.
- When a feature is omitted, all its dimensions and children are omitted or ignored. If a feature is present, all its parents are present. In cases of conflict, omitted features supersede present features.
- In a multi-level Family Table, you cannot add a feature to an instance Family Table if that feature is already added as a column in the parent Family Table. The system displays the message Cannot add a parameter existing in a parent table of this object.
- When dealing with patterns, the system automatically records the parent feature. Also, note that you can include the pattern instance parameter in the table. You can have zero instances by omitting the pattern completely.
- When you place a UDF defined with a Family Table, any features in the UDF instance which have been suppressed are not placed on the part. They are in effect deleted as the UDF group is placed.

The following information is helpful when adding a UDF to a Family Table:

- When you place a UDF from a Family Table on a part, any features in the UDF which have been suppressed are not placed on the part. They are in effect deleted as the UDF is placed.
- When a UDF contains a Family Table within it, you cannot suppress and resume features within that UDF by adding instance names of the UDF to the part Family Table. This is because when UDF is placed, any feature that is suppressed in the UDF Family Table will not be placed at all. Therefore, when adding that same UDF group to a part Family Table, any non-placed features cannot be resumed, and

likewise, any placed feature cannot be suppressed when creating a new part instance.


Adding a Feature to a Family Table By Using Suppression

In some cases, you must suppress some features before making and including others in the table. This makes it possible to include features that may conflict if they are present at the same time, such as a round and a chamfer on the same edge.

For example, suppose you have an edge that can be filleted, chamfered, or left square depending on the instance.

In this case, you would have to add both the fillet and the chamfer features to the Family Table; however, you cannot have both features on the model simultaneously. You can, however, create the fillet, add it to the Family Table, and then suppress it. You could then create the chamfer and add it to the table later.


To Add a Feature to a Family Table

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Click . The **Family Items** dialog box opens.
3. Under **Add Item**, click **Feature**. The **SELECT FEAT** menu appears and the **Select** dialog box opens.
4. Select the feature that you want to add graphically and click **OK** in the **Select** dialog box.
5. Click **Done** on the **SELECT FEAT** menu.
6. Click **OK** in the **Family Items** dialog box.

Note: In a multi-level Family Table, you cannot add a feature to an instance Family Table if that feature is already added as a column in the parent Family Table. The system displays the message `Cannot add a parameter existing in a parent table of this object.`

To Add a User-Defined Feature (UDF) to a Family Table

Only table-driven UDFs (generic or instance) that have been placed on the model can be added to the model family table.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Click . The **Family Items** dialog box opens.
3. Under **Add Item**, click **Group**. A menu appears and lists the UDFs contained in the model.
4. Select a UDF from the list, and then click **OK**.

Note: You cannot add a feature to an instance Family Table if that feature is already added as a column in the parent Family Table. The system displays the message `Cannot add a parameter existing in a parent table of this object.`

Tip: Adding UDFs to Family Tables

When adding a UDF to a Family Table, keep in mind that only table-driven UDFs (generic or instance) that have been placed on the model can be added to a family table. Adding UDFs to Family Tables do not work under the following circumstances:

- When a model contains a local group or UDF group definition, the definition itself is not a valid group to add to the Family Table. You must first define the group, designate it as table-driven, and then place it on the model before you add it to the Family Table.
- When a simple UDF is placed on a model, the UDF itself is not a valid group to add to the Family Table. You must first designate it as table-driven and then place it on the model before you add it to the Family Table.

Tip: Adding Features to a Family Table Using Suppression

In some cases, you must suppress some features before making and including others in the table. This makes it possible to include features that may conflict if they are present at the same time, such as a round and a chamfer on the same edge.

For example, suppose you have an edge that can be filleted, chamfered, or left square depending on the instance.

In this case, you would have to add both the fillet and the chamfer features to the Family Table; however, you cannot have both features on the model simultaneously. You can, however, create the fillet, add it to the Family Table, and then suppress it. You could then create the chamfer and add it to the table later.

Tip: Adding Features in Multi-Level Family Tables

In a multi-level Family Table, you cannot add a feature to an instance Family Table if that feature is already added as a column in the parent Family Table. The system displays the following message:

`Cannot add a parameter existing in a parent table of this object.`

Adding Reference Models to Family Tables

To Add a Reference Model to a Family Table

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.

2. Click . The **Family Items** dialog box opens.

3. Under **Add Item**, click **Ref Model**. The **REFERENCES** menu appears with a list of reference models.

4. Select the reference model or models from the list and click **DONE**.
5. Click **OK**.

Tip: Adding Reference Models

It is useful to add external references to the family tree of a model for a family of assemblies when a component refers to the generic assembly or another component in the generic assembly.

Changes in the reference assembly or component can cause changes to the geometry of the component with the external reference.

To ensure that the geometry of the component with the external reference is updated, create a Family Table for the component with the external reference in which each instance has an external reference to the appropriate assembly instance. The assembly Family Table must also have a corresponding item for the component with the external reference.

Example: Adding a Reference Model to a Family Table

This example shows a Family Table that includes an external reference. It uses two parts (A and B) and an assembly (AB).

The hole in part B is dimensioned to the outside edge of part A. In different instances of assembly AB, the thickness of part A varies, and so the hole in part B must be located differently. However, since part B has an external reference, its geometry can only be updated if it knows which assembly instance it needs to reference.

Part A has a Family Table that controls the values of its thickness. Assembly AB has a Family Table in which each instance of AB includes a different instance of part A and part B, Part B has a Family Table in which the external reference to assembly AB is established.

Family Table for Part A (5 instances)

Instance	d2	d3
Generic	3.00	3.00
A01	1.00	1.00
A02	2.00	2.00
A03	3.00	3.00
A04	4.00	4.00
A05	5.00	5.00

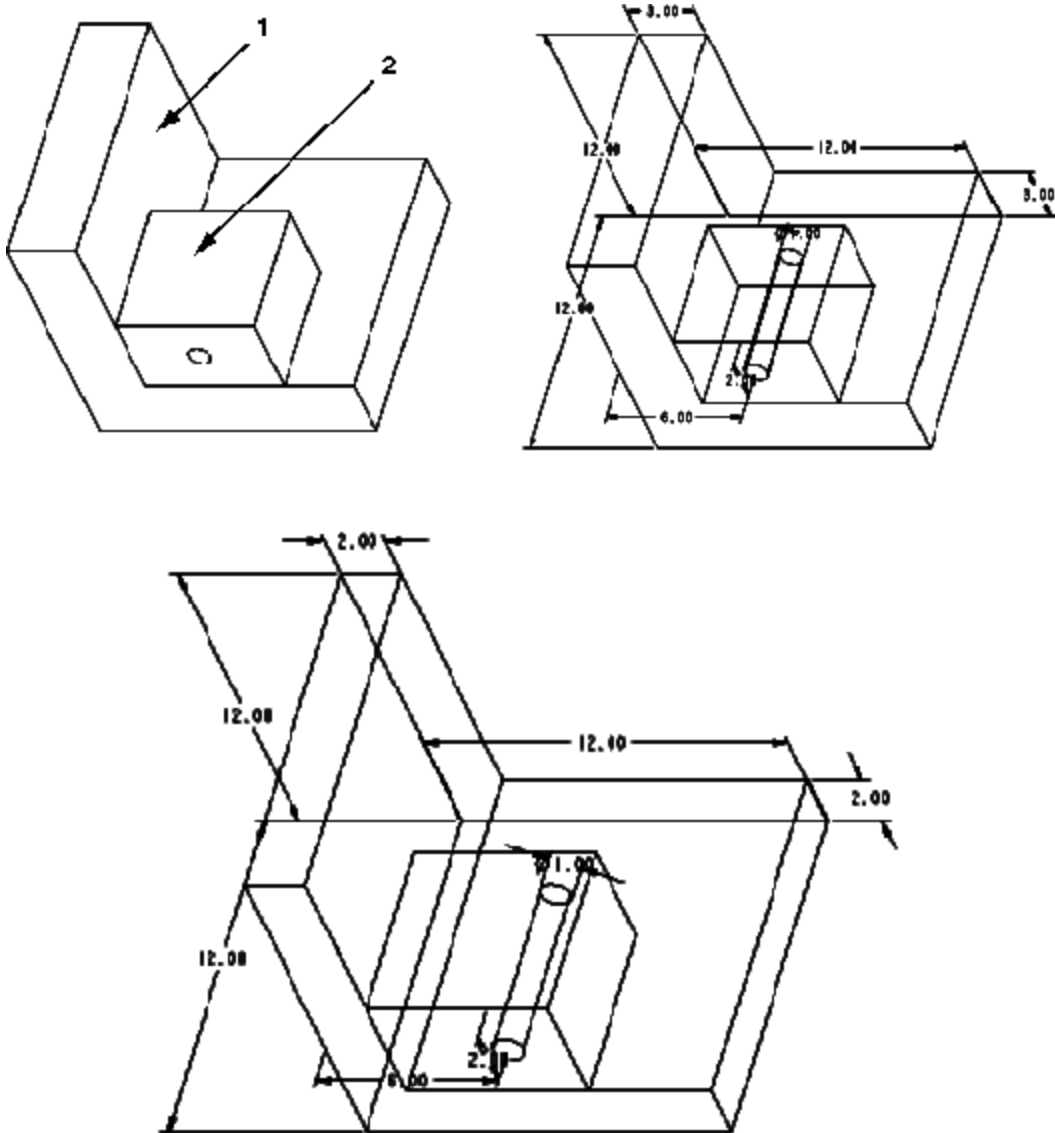
Family Table for Part B (5 instances)

Instance	External Reference 0
Generic	AB

B01	AB01
B02	AB02
B03	AB03
B04	AB04
B05	AB05

Family Table for Assembly AB (5 instances)

Instance	Component	Component
Generic	A	B
AB01	A01	B01
AB02	A02	B02
AB03	A03	B03
AB04	A04	B04
AB05	A05	B05



1 Part A

2 Part B

The generic of Assembly AB, in the top part of the diagram, shows Part A and Part B joined together and then dimensioned. An Instance of Assembly AB, shown dimensioned at the bottom of the diagram, reveals that the hold in Part B has an external reference to Part A.


Adding Patterning to Family Tables

To Add a Pattern Table to a Family Table

Pattern tables can be included in Family Tables, enabling different instances of a family to use different pattern tables to drive the same patterned feature. The generic model must contain all of the pattern tables required by the family instances.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.



2. Click . The **Family Items** dialog box opens.
3. Under **Add Item**, click **Pattern Table**.
4. Select the patterned feature on the model that corresponds to the pattern table you want to add to the Family Table. The name of the current pattern table is automatically entered for the generic in the table. The instances also have the current pattern table by default.

Example: Patterning

A simple example of a two-direction pattern is a bolt, where there are four different nominal diameters, and for each diameter there are five lengths. These bolts can be created using several levels of Family Tables or using patterning. With patterning, all instances can be quickly created in the same table.

To pattern this bolt, make sure that the parameter **LENGTH** is in the table. Then, follow these steps:

1. Select the parameter **NOM** as the first direction pattern. Specify a quantity of 4 and an increment of 0.125.
2. Select the parameter **LENGTH** as the second direction pattern. Specify a quantity of 5 and an increment of 1.0.

A duplicate seed instance is created with a new name, and every instance added after it is incremented by one.


```
!Generic part name: MSAA
!NAME          d1          d2          d9          d10         d4
!              nom          head_di    slot_w     slot_d     Length
!GENERIC      0.125      0.250      0.048      0.030      1.000
MSAA          0.250      0.250      0.048      0.030      1.000
```

```
!Generic part name: MSAA
!NAME          d1          d2          d9          d10         d4
!              nom          head_di    slot_w     slot_d     Length
!GENERIC      0.125      0.250      0.048      0.030      1.000
MSAA00        0.250      0.250      0.048      0.030      1.000
MSAA01        0.250      0.250      0.048      0.030      2.000
MSAA02        0.250      0.250      0.048      0.030      3.000
MSAA03        0.250      0.250      0.048      0.030      4.000
MSAA04        0.250      0.250      0.048      0.030      5.000
MSAA05        0.375      0.250      0.048      0.030      1.000
MSAA06        0.375      0.250      0.048      0.030      2.000
MSAA07        0.375      0.250      0.048      0.030      3.000
```

Note: There are five lengths for each nominal diameter added to the table, as shown in column d4 above.

To Pattern a Family Table Instance

Before you attempt to pattern an instance, you must create a seed instance and to make sure that all parameters to be patterned are present.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Select an instance, and then click **Edit > Copy with Increments**, or click . The **Patternize Instance** dialog box opens with Direction1 under **Pattern Tree** highlighted by default and a quantity of 1.
3. Specify a new quantity and select one parameter to increment.
4. Click on the double arrows to accept the parameter.
5. Enter an increment.
6. You may select additional parameters and use the double arrows to add and remove them. Enter an increment for each parameter added.
7. When you are finished, select **OK** to complete the pattern or add more directions by clicking on the **Add** (plus sign) button.
8. Click **OK** when you are done in all directions.

Note: You may go back and add or remove directions at any time.


Creating Instances of Parts or Assemblies in Family Tables

To Create an Instance of a Part or Assembly

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.

2. Click **Insert > Instance row**, or click . A new row for a new instance is added to the table.

Instance names can be up to 31 characters long. All instance names must be unique, and you must not use the same instance name in different families.

3. Click . The **Family Items** dialog box opens.
4. Under **Add Item**, select the type to add.
5. Click **OK** when you are finished adding items. The Family Table is filled in with values for the generic model and asterisks for the instances.
6. For each instance, type the table-driven value of each item in the table.
7. Click **OK** to store the table.

Creating Instances by Patterning

Patterning an instance automatically creates other instances that have incremental values for selected parameters or dimensions. The instance used for the patterning does not become a parent; it is used, in effect, as a seed to generate the new Family Table members.

Features, component members, and groups in a Family Table cannot be patterned, but instances to which they belong can. You can then edit the table to change these items.

Note that patterning an instance does not replace the original seed instance with the pattern instance, but instead duplicates the seed and the first instance. You must delete the original instance manually if you do not want it in the table.


Patterning an instance can be done with several directions. The first direction is a set of parameters or dimensions that you want incremented together, the second direction is actually a patterning of the first, where you select a parameter that is incremented a certain number of times for each of the first direction instances.

The third direction is a patterning of the second, where you select a parameter that is incremented a certain number of times for each of the second direction instances, and so on. Using patterning can propagate instances quickly.

To Add an Instance With No Table-Driven Features

Add the table-driven features to the Family Table.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.

2. Click **Insert > Instance Row**, or click . Asterisks appear in all cells for the new instance row.
3. Select each cell in the row and type N . For this instance, none of the table-driven features will appear.
4. Click **OK**.

Working with Instances in Family Tables

About Instance Operations

Instance operations include tasks like verifying, unlocking, renaming, and retrieving instances, creating new instances, and creating and updating instance indexes and accelerator files. You can use **File > Instance Operations > Update Index** or **File > Instance Operations > Accelerator Options** (which opens the **Instance Accelerator** dialog box) to perform most instance operations. You can also use the **Family Table** to perform certain instance operations.

Retrieving Instances

When you retrieve an instance from disk, the system first checks to see if an instance accelerator file exists for the instance.

- If the instance accelerator file exists, the system retrieves the generic and the accelerator file. If the generic has not changed since the instance was last saved, the system immediately proceeds as if the instance were a separate model. Otherwise, if the generic has changed, the instance is regenerated before displaying it.
- If the instance accelerator file does not exist, the system retrieves the generic and regenerates the instance.

Verifying Instances

When you verify an instance, you regenerate the instance. You can use **Verify** to facilitate the process of bringing newly created instances into session so that they can be added to the instance index file. Saving the generic after verification causes the names of all instances that were successfully regenerated to be added to the instance index file for that directory.

Copying an Instance to the Generic

You can copy the values of an instance to the generic model. However, you must remember that the generic establishes the default configuration (absence or presence of features, members) and dimensions for all instances. Therefore, replacing the values for the generic with the values from an instance may change instances where you used an asterisk to accept default values.

When you copy an instance to the generic, the original generic is deleted. If you want to keep the generic model, create an instance identical to it.

Deleting Instances from a Family Table

To delete an instance, delete the row that contains the instance. If the instance is in memory, Pro/ENGINEER warns you that the instance is no longer table-driven.

After you delete an instance, you cannot undo the deletion. To retrieve the instance without creating it anew, you can erase the model from the current memory and close the window. This works only if you have not saved the model. If you have saved the model, you can retrieve an earlier version in which the instance still exists.

About Verifying Instances

When you verify an instance, you regenerate the instance. You can use **Verify** to facilitate the process of bringing newly created instances into session so that they can be added to the instance index file. Saving the generic after verification causes the names of all instances that were successfully regenerated to be added to the instance index file for that directory.

To verify instances, you can click **Tools > Verify**, or the **Verify** toolbar button, both located in the **Family Table** dialog box. Both commands open the **Family Tree** dialog box, which lists and selects all instances for verification. You can verify all instances, or only selected instances.

As each instance is regenerated, a red arrow icon is placed next to its name (this can happen very quickly). After an instance is regenerated, the status of the regeneration (**SUCCESS** or **FAILURE**) is displayed under the **Verify status** column of the dialog box.

The results of the verification are written to a file named `modelname.tst`, where `modelname` is the name of the current generic model. This `.tst` file records the instance name, the name of the generic, and the regeneration status of each verified instance.

Note:

If a family tree has multiple branches (as in a multi-level Family Table), the verification process regenerates all instances in the tree.

After each instance is regenerated, it is erased from memory. However, those instances that were already in memory (such as instances retrieved using **Instance**) remain available in memory.

To Create a Real Model from an Instance


1. Open the instance.
2. Click **File > Save** or **File > Save a Copy**.
3. Type a name for the new model. The instance is saved as a new model with no relationship to the generic.

Note:

- Suppressed features are deleted from the model.
- Dimensions are set with the current instance values.

To Verify an Instance

When you verify an instance, the system regenerates the instance and checks for failures.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. In the dialog box, click  or **Tools > Verify**. The **Family Tree** dialog box opens with all the instances of the family selected.
3. Click **VERIFY** to regenerate each instance in sequence from the first to the last or select one or more instances and click **VERIFY** to regenerate only the selected instances.

The selected instances are regenerated. If the family tree contains multiple branches, all selected instances in the tree are verified. The verification (regeneration) status (SUCCESS or FAILURE) is displayed under **Verification Status**.

Note: If an instance fails regeneration, Pro/ENGINEER does not automatically enter the Resolve environment. You must investigate the instance row in the Family Table to find any problems with the failed instance.

To Copy an Instance to the Generic Model

Note: This procedure permanently deletes the generic and replaces it with the values of the instance; you can not undo the change. If you click **Cancel**, the original generic is not restored.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Select an instance name.
3. Click **Edit > Copy Instance to Generic**.

To Rename an Instance

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Under **Instance Name**, select the instance you want to rename and type the new name.
3. Click **OK** to close the **Family Table** and accept your changes.

Note: You cannot rename the generic (located in the first row).

To Save Instances of Parts and Assemblies

1. Click **File > Instance Operations > Accelerator Options**. The **Instance Accelerator** dialog box opens.
2. Under **Creation Options**, select one of the following options:
 - **None**—Pro/ENGINEER does not save an instance by saving the generic model and its Family Table.

- **Always**—Pro/ENGINEER always saves the accelerator file of an instance both when you explicitly save the instance itself and when you save the instance through a higher-level object (for example, when you save the entire model).
- **Explicit**—Pro/ENGINEER saves an instance accelerator file only when you retrieve the instance into the Pro/ENGINEER session and then explicitly save the instance.
- **Saved Objects**—This is the default. This option is similar to **Always** except that Pro/ENGINEER updates the accelerator files whenever you modify or verify an instance and the instance verification status is not set to Failed.


In the **Instance Accelerator** dialog box, one of the options is highlighted to reflect the current setting which was loaded from a configuration file or selected previously from this dialog box.

3. Click **Close**.

Alternatively, you can save instances of parts and assemblies by setting the `save_instance_accelerator` configuration option to `None`, `Always`, `Explicit`, or `Saved Objects` by clicking **Tools > Options**.

To Lock or Unlock an Instance

An instance of a Family Table can be locked so that table-driven characteristics of the instance cannot be modified. A lock icon is contained in the locked instance row in the **Type** column.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Select the names of the instances that you want to lock. Press and hold the CTRL key to select more than one instance.
3. Click **Tools > Lock / Unlock Instance**, or click . A lock icon is displayed next to each instance name.

Note: The instances are locked only when you click **OK** to save the table, or when you click **Open** to open the instance. You can make changes to the file, but when you close the file, the locked values are restored.

Follow the same procedure to unlock instances.

Working with Instance Index Files

About Instance Index Files

Instance index files enable you to retrieve instances by entering their instance name only. An instance index file displays the total number of instances, followed by a column of instance names on the left and a column of their corresponding generic names on the right. The instance index file is named `directoryname.idx` (where `directoryname` is the directory in which the file is saved).

- Saving any generic model automatically creates a new instance index file, or updates the existing file. This file lists the names of all the regenerated instances for every generic regenerated and saved in the same directory as the current generic.

The instance names that are listed in the instance index file are only those instances that were regenerated when their generic model was saved. The regenerated instances did not have to be in memory when the generic was saved. For example, the instances could have been retrieved, then erased from the session using the Erase command on the **File** menu. (The exception to this is for Pro/LIBRARY objects, such as parts from the Basic Library).

Storing the generic part automatically adds the names of all of its instances to the instance index file, without the need to regenerate them. This includes the names of instances in subfamilies, such as instances that have instances, as well.

Instance index files are cumulative, so instances that are retrieved in later sessions can be added to the list. To add new instance names to the file, you must save the generic after those instances have been regenerated; the instance index file is then updated, including the new instances.

- You can also create or update an instance index file by clicking **File > Instance Operations > Update Index**.

This creates or updates an instance index file for a specified directory (the current working directory is the default). **Update Index** creates the instance index file for all generics saved in the specified directory, not for objects that are in session or active in the Main Window.

- You can use the **Tools > Verify** command in the **Family Table** to facilitate the process of bringing instances into session so that they can be added to the instance index file. Saving the generic after verification causes the names of all instances that were successfully regenerated to be listed in the instance index file for that directory.
- The `.idx` file is a text file and can be edited outside of Pro/ENGINEER, within any text editor. However, if you edit this file outside of Pro/ENGINEER, Pro/ENGINEER will not recognize the file. For instance, when you click **File > Open** to open the model that contains the edited `.idx` file, its instance names do not display in the **File Open** dialog box.

To Create an Instance Index File

1. Save the generic model by clicking **File > Save**. The system automatically adds the names of all of its instances to the instance index file, without the need to regenerate them. This includes the names of instances in subfamilies, such as instances that have instances.

Note: To add new instance names to the file, you must save the generic after those instances have been regenerated; the instance index file is then updated, including the new instances.

2. Click **File > Instance Operations > Update Index**. The system creates or updates an instance index file for a specified directory (the current working directory is the default).

Note: Update Index creates the instance index file for all generics saved in the specified directory, not for objects that are in session or active in the Main Window.

3. In the **Family Table** dialog box, click **Tools > Verify** to facilitate the process of bringing instances into session so that they can be added to the instance index file. Saving the generic after verification causes the names of all instances that were successfully regenerated to be listed in the instance index file for that directory.

Tip: Updating the Instance Index File

Whenever you move a generic model from one directory to another using operating system commands, update the instance index file by clicking **File > Instance Operations > Update Index** in the source and destination directories. This ensures that the instances of that generic can be retrieved from their new directory.

Note: Do not edit an instance index file from outside Pro/ENGINEER. Pro/ENGINEER does not recognize the file if it is edited outside of the application.

Working with Instance Accelerator Files

About Instance Accelerator Files

You can reduce the time that Pro/ENGINEER takes to retrieve an instance of a part or assembly from disk by saving the instance in a special file called an instance accelerator file. Each instance is saved in its own unique accelerator file, named `instancename.xpr` for part instances and `instancename.xas` for assembly instances.

The system behavior in saving instances of parts and assemblies is controlled by the `Save Instance Accelerator` variable. You can set up or modify this variable in two ways as follows:

- By setting the `save_instance_accelerator` configuration option. You can select one of the following values:
 - `None`—Pro/ENGINEER does not save an instance by saving the generic model and its Family Table.
 - `Always`—Pro/ENGINEER saves an accelerator file when you explicitly save the instance. You can save an instance by opening the instance in a new Pro/ENGINEER window using **Open** in the **Family Table** dialog box, and then saving it as a separate model file using **File > Save** or **File > Save a Copy** in the **Pro/ENGINEER** menu bar. You can also save the instance through a higher level object, that is, when you save the entire model file using **File > Save** on the **Pro/ENGINEER** menu bar.

- `Explicit`—Pro/ENGINEER saves an accelerator file only when you explicitly save the instance (see the preceding paragraph for a description of saving an instance).
- `Saved Objects`—This is the default. This option is similar to **Always** except that Pro/ENGINEER updates the accelerator files whenever you modify or verify an instance and the instance verification status is not set to Failed.
- By clicking **File > Instance Operations > Accelerator Options**. The **Instance Accelerator** dialog box opens. The **Creation Options** area contains the same options as the `save_instance_accelerator` configuration option:
 - None
 - Always
 - Explicit
 - Saved Objects (default)

One of these options is highlighted to reflect the current setting that was loaded from a configuration file or selected previously using this dialog box.

Using the **Instance Accelerator** dialog box, you can also:

- Create an instance accelerator file.
- Save instances of parts and assemblies.
- Update the generic and all instance accelerator files.
- Delete invalid instance accelerator files.

To Create an Instance Accelerator File

1. Click **File > Instance Operations > Accelerator Options**. The **Instance Accelerator** dialog box opens.
2. Under **Creation Options**, select the desired option, that is, one of **None**, **Always**, **Explicit** or **Saved Objects**. One of these options is automatically selected to reflect the current setting that was loaded from a configuration file or selected previously using this dialog box.
3. Click **Update** to create a new instance accelerator file.
4. Click **Close**.

Alternatively, set the `save_instance_accelerator` configuration option to `None`, `Always`, `Explicit`, or `Saved objects` by clicking **Tools > Options**.

To Delete Invalid Instance Accelerator Files

1. Click **File > Instance Operations > Accelerator Options**. The **Instance Accelerator** dialog box opens.

2. The current instance appears under **Instance**. To delete the invalid instance accelerator files for the current instance, click **Purge**. To delete the invalid accelerator files for another instance, navigate to the instance whose accelerator files that you want to delete, and then click **Purge**.

Note: Purge (in-session generic only) checks the validity of all instance accelerator files on disk and deletes them if they are not current with the generic.

To Update the Generic and All Instance Accelerator Files

1. Click **File > Instance Operations > Accelerator Options**. The **Instance Accelerator** dialog box opens.
2. Under **Instance**, click **Update**. **Update** updates the generic file or files and all existing instance accelerator files. In addition, it creates instance accelerator files for all Family Table instances of the generic that are listed in the instance index file and which do not currently have them. This has the same effect on the generic as clicking **File > Save**.

If the `save_instance_accelerator` configuration option is set to `None` when you choose this command, a **Confirmation** dialog box opens and Pro/ENGINEER prompts you as to whether you still want to update the accelerator files.

3. Click Yes to update the files, or click No to cancel the update.

Note: The system regenerates each instance accelerator file during the update process. It deletes any file that fails the regeneration.

Retrieving Instances

About Retrieving Instances

When you retrieve an instance, Pro/ENGINEER opens the specified instance in a new window. When you retrieve a generic by using **File > Open**, Pro/ENGINEER retrieves the instance rather than the generic into the main window. Pro/ENGINEER regenerates the instance automatically during retrieval.

You can retrieve an instance of a family of parts or assemblies using one of the following methods:

- With the generic already open and active, select the instance from the Family Table and click **Open**.
- Open a file that contains a Family Table, select an instance from the **Select Instance** dialog box, and click **Open**.
- Click **File > Open**. All instances for the model are listed as parts unless you explicitly set the `menu_show_instance` configuration option to `no`. Select the required instance and click **Open**.

Retrieving Assembly Instances

When you retrieve an assembly instance, Pro/ENGINEER retrieves and assembles all the necessary components.

All components of the generic assembly must be accessible, that is, they must either be stored locally, or they must reside along a search path.

When you retrieve an assembly instance, Pro/ENGINEER does not prompt for an instance name of the generic components. It places the exact component (whatever is written in the component entry) at the instance line in the Family Table.


If you set the `sel_insts_on_comp_retrieval` configuration file option to `yes`, then when you retrieve a generic, Pro/ENGINEER prompts you to specify an instance of that generic.

Set the `retrieve_instance_dependencies` configuration option to an appropriate value to determine what set of objects are loaded into the session when retrieving Family Table instances with dependencies. On setting the `retrieve_instance_dependencies` configuration option to:

- `instance_deps_only`—you can only retrieve an instance assembly with instance dependencies and the generic with no dependencies. The generic dependencies are never retrieved even if there are some references to them from the instance components.
- `Instance_req_generic`—You can retrieve the instance and the required generic models along with their dependencies. Generic dependencies such as components excluded from the instance are not retrieved.
- `instance_and_generic_deps`—This is set by default. You can retrieve instances with its dependencies and all dependencies of the generic.

To Preview an Instance

Follow this procedure if you want to view a smaller version of a selected instance without having to open it first.


1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Select the instance you want to preview (select any cell in the instance row).
3. In the **Family Table** dialog box, click **Tools > Preview**, or click . A picture of the selected instance opens in a small **PREVIEW** window.

To Open an Instance from Within the Family Table

You can open an instance by selecting it from the Family Table. The instance opens in a separate Pro/ENGINEER window.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens and lists all the instances of the current generic part.
2. Select any cell in the row containing the instance you want to open.
3. Click **Open**. The instance opens in a new Pro/ENGINEER window.


To Retrieve the Generic During File/Open

1. Open a model that contains a Family Table. The **Select Instance** dialog box opens.
2. Click the **By Name** tab.
3. Select the generic from the displayed list.
4. To preview the generic without opening it in a new Pro/ENGINEER window, click **Tools > Preview**, or click . To open the generic in a new Pro/ENGINEER window, click **Open**.


To Retrieve an Instance During File/Open

You can retrieve an instance by name or by parameter.

To Retrieve an Instance by Name

1. Open a model that contains a Family Table by clicking **File > Open**. The **Select Instance** dialog box opens.
2. Click the **By Name** tab if it is not already active. Then, select the name of the instance from the displayed list. (You can also type the name of the instance.)
3. To preview the instance without opening it in a new Pro/ENGINEER window, click **Tools > Preview**, or click . To open the instance in a new Pro/ENGINEER window, click **Open**.

To Retrieve an Instance by Parameter

1. Open a model that contains a Family Table by clicking **File > Open**. The **Select Instance** dialog box opens.
2. Click the **By Parameter** tab. Then, select a parameter from the **Parameter** list and a value (if there is more than one) from the **Value** list. The name of the instances that contain the specified parameter and value appear in the **Name** box.
3. Select the name of the desired instance from the **Name** box.
4. To preview the instance without opening it in a new Pro/ENGINEER window, click **Tools > Preview**, or click . To open the instance in a new Pro/ENGINEER window, click **Open**.

Note: To restore the entire list of instance names, click **Restore**.

Tip: Speeding Up Instance Retrieval

Pro/ENGINEER retrieves an instance from disk using the following process:

1. Retrieves the generic

2. Copies the generic in memory
3. Changes that copy according to the parameters of the Family Table instance
4. Regenerates the copy

The generic and the instance are then both held in memory.

If, however, you have saved a particular instance as a separate model (using **File > Save** or **File > Save a Copy**), then Pro/ENGINEER retrieves it directly without going through the regeneration process—at a considerable savings in time (especially for complex models with many features and/or components). However, the instance no longer has any ties to its former generic.

Searching in a Family Table

About Searching in a Family Table

You can search through the current level of a Family Table for instances that contain a common element or elements; for example, you can search for all instances that contain a specified parameter, character string, or *yes/no* variable.

To perform the search, use the **Search** dialog box through the Family Table dialog box. The **Search** dialog box is identical to the one you use to perform Model Tree searches. The procedure for performing a search in a Family Table is the same as that for performing a search in the Model Tree, except that the options presented for types and values for the search are different for a Family Table than those for the Model Tree. As with Model Tree searches, you can perform single condition or multiple condition searches in a Family Table.

Type allows you to search for instances by specifying the type of information you want to find. The following type choices are available:

- **Table Item**—This is the default type choice. An item can be a dimension, a parameter, a feature number, a user-defined feature name, or an assembly member name that is selected to be table-driven. You can search for instances by specifying a common item.
- **Model Name**—Use this type when you want to search for an instance by its instance name. A list of available instance names appears in the **Value** list box.
- **Expression**—Use this type when you want to search for instances using a common string variable, *Yes/No* variable, or number variable.

About Using Expressions When Searching a Family Table

An expression is a method by which classes of parameters can be specified in a search. By defining an expression involving a string, *yes/no* variable, or number variable, groupings of parameters can be found with a single preparation of the search tool.

The basic form for an expression is as follows:

```
<parameter> ==, !=, <, > <variable>
```

where `parameter` is the name of the parameter, `==`, `!=`, `<`, and `>` are individual operations, and `variable` is the name of the search variable.


Sample Expressions

A model and Family Table can contain any parameter(s) you want it to contain. For example, assume a model with the following model parameters:

Parameter	Possible Sample Parameter Values
name	'bolt' or 'washer'
generic	'yes' or 'no'
level	a number between 1 and 9

You can use these sample parameters and values in search expressions in the **Search** dialog box, when searching in the Model Tree or in a Family Table.


To Perform a Single-Condition Search in a Family Table

1. In the **Family Table** dialog box, click **Edit > Find**, or click . The **Search** dialog box opens.
2. From the **Type** list, select an information type, and then select corresponding values from the **Value** list. For predefined information types, predetermined items appear in the **Value** list on the right. Refine your criteria by selecting `=` (equals) or `!=` (does not equal).

Note: The parameter **Expression** refers to an expression that will be searched for in the parameter list.
3. Click **Add** to add the current definition of the criteria to the list of search criteria at the bottom of the dialog box.
4. Click **Find** to perform the search. Beginning from the top of the Family Table (or the current cell if one is selected), the search engine moves to the next item that meets the search criteria. Continue clicking **Find** to move to the next found item.
5. To remove a search condition and use a different one, select the condition in the list of search criteria and click **Remove**. The condition clears and a new condition can be used.
6. To alter a condition, select a condition and click **Change**. You can then redefine, as outlined in Steps 2 and 3.
7. Click **Close** when finished searching.

To Perform a Multiple-Condition Search in a Family Table

To perform more accurate searches, it is necessary to have multiple search parameters. The search tool allows for the easy setup and customization of such operations.

1. In the **Family Table** dialog box, click **Edit > Find**, or click . The **Search** dialog box opens.
2. From the **Type** list, select the required information type, and then select corresponding values from the **Value** list. For predefined information types, predetermined items appear in the **Value** list on the right. Refine your criteria by selecting = (equals) or != (does not equal).

Note: The parameter **Expression** refers to an expression that will be searched for in the parameter list.

If you select a model parameter from the **Table Item** type list, you can type a search value in the text box below the **Value** list.


3. Click **Add** to add the current definition of the criteria to the list of search criteria at the bottom of the dialog box.
4. As additional conditions are included in the search, they appear in the list of search criteria separated by a default **AND** operator. To change this operator, select it in the list box and then choose the option button **AND** or **OR** above the list box.

Items in parentheses are evaluated together, as in the following:

(conditional AND conditional) OR (conditional)

5. Click **Find** to perform the search. Beginning from the top of the Family Table (or the current cell if one is selected), the search engine moves to the next item that meets the search criteria. Continue clicking **Find** to move to the next found item.
6. To remove a search condition and use a different one, select the condition in the list of search criteria and click **Remove**. The condition clears and a new condition can be used.
7. To alter a condition, select a condition and click **Change**. You will then be able to redefine, as outlined in Steps 2 and 3.
8. Click **Close** when finished searching.

To Perform a Search for Instances in a Family Table

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. In the **Family Table**, click **Edit > Find**, or click . The **Search** dialog box opens.
3. Perform a single-condition or multiple-condition search by specifying search criteria and adding them to the list at the bottom of the dialog box.
4. Click **Find** to perform your search. The system selects the first found instance in the table. To step through all the found instances, continue clicking **Find**.

Example: Using Expressions to Perform a Family Table Search

The following three examples illustrate how to use model expressions to search in a Family Table.

For example, assume a model with the following model parameters:

Parameter	Possible Sample Parameter Values
name	'bolt' or 'washer'
generic	'yes' or 'no'
level	a number between 1 and 9

These parameters and values are used in the following sample procedures.

Example 1: To Find Instances of a String Variable with Expression

To find all instances of the **name: "bolt"** parameter, you would perform the following procedure:

1. Perform a single-condition search using the **Search** dialog box.
2. Select **Expression** as your information type.
3. In the box below the **Value** list, enter the following:

```
name = "bolt"
```

Note:

In a string search, you must include quotation marks around the name of the string variable.

The only permissible operators in a string search are = and !=.

4. Click **Add** to include this expression in the list of search criteria.
5. Click **Find** to begin the search, or complete steps 2 and 3 again to add other parameters.

Example 2: To Find Instances of a Yes/No Variable with Expression

To find all instances of the **generic: yes** parameter:

1. Perform a single-condition search using the **Search** dialog box.
2. Select **Expression** as your information type.
3. In the box below the **Value** list, enter the following:

```
generic = yes
```

Note:

In a yes/no search, quotation marks are unnecessary.

The only permissible operators in a yes/no search are = and !=.

Click **Add** to include this expression in the list of search criteria.

Click **Find** to begin the search, or complete steps 2 and 3 again to add other parameters.

Example 3: To Find Instances of a Number Variable with Expression

To find all instances of the **level** parameter with a value of greater than **4**:

1. Perform a single-condition search.
2. Select **Expression** as your information type.
3. In the box below the **Value** list, enter the following:

```
level > 4
```

Note:

You need not put the variable in quotation marks.

Any of the operators (=, !=, <, or >) can be used.

Decimals are permissible.

Deleting Items from Family Tables

About Deleting Items from Family Tables

Instances and other items can be deleted from Family Tables. Doing so removes them from the control of the table and causes all instances to reflect the corresponding changes. All the information about an instance is contained in a single row of the Family Table.

To delete an instance, you delete the row that contains the instance. If the instance is in memory, Pro/ENGINEER warns you that the instance is no longer table-driven.

Note: Once you delete an instance or other item, you cannot undo the deletion. If you have deleted an instance or other item in error, you can erase the model from the current memory and close the window if you have not already saved the model. If you have already saved the model, you can close it and retrieve an earlier version. The deleted instance(s) still exist in earlier versions of the model.

To Delete an Item from a Family Table

1. With the generic model active, click **Tools > Family Table**. The **FAMILY TABLE** dialog box opens.
2. Navigate to the appropriate Family Table.
3. Select a cell in the column for the item you want to delete.
4. Do one of the following:

- Click **Edit > Delete Columns**.
 - Right-click and select **Delete Columns** from the shortcut menu.
5. Confirm the deletion. Once you delete an item, you cannot undo the deletion. If you have not saved the model, you can erase the model from the current memory and close the window. If you have saved the model, you can close it and retrieve an earlier version in which the item still exists.

To Delete an Instance from a Family Table

1. With the generic model active, click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Navigate to the Family Table that contains the instance(s) you want to delete.
3. Select the instance(s) you want to delete. To select multiple instances, hold the CTRL key down as you select each instance.
4. Click **Edit > Delete Rows**. The selected instance rows are deleted permanently.

Note: You cannot undo the deletion.

Saving Family Table Information

About Saving Family Tables

When you save a model or one of its instances by clicking **File > Save** on the Pro/ENGINEER menu bar, Pro/ENGINEER automatically saves Family Table information for that model, including the generic part, features, and dimensions found in the Family Table. Pro/ENGINEER saves the Family Table information separately from the model file by creating an external generic file named `modelname.ptd`. When you save the model, the `modelname.ptd` file contains the same Family Table information as is stored internally within the part database. Every time you click **File > Save**, Pro/ENGINEER updates the `.ptd` file.

When you save the generic, the instances in the Family Table are not saved to the `.ptd` file and you can retrieve the instances only through the generic; that is, only by opening the generic model. However, you can set the `save_instance_accelerator` configuration option to save instances automatically every time you click **File > Save**. Instances are saved to an `.xpr` instance accelerator file for part instances and `.xas` instance accelerator file for assembly instances. You can set the `save_instance_accelerator` configuration option to the following values, this controls whether the instances are saved or not saved during the **File > Save** operation:

- If the `save_instance_accelerator` configuration option is set to `None`, the instances are not saved and can be retrieved later only through the generic.
- If the `save_instance_accelerator` option is set to `Explicit`, `Always`, or `Saved Objects` the instances are saved in an Instance Accelerator File.

Saving Family Table Information from Within a Family Table

You can also save the contents of a Family Table separately from saving the model. To do so, use **File > Export Table** in the Family Table. You can save a Family Table as a text file or as a Pro/TABLE file. If you save a Family Table as a regular text file, you can edit the table information within an external text editor, such as Microsoft Word. If you save it as a Pro/TABLE file, the information is not editable. In either case, the file extension of the saved text file is `.ptd`.

Once you have exported Family Table information, you can retrieve it into another Family Table using **File > Import Table** in the Family Table.

To Save Family Table Information

There are two methods to save information contained in a Family Table:

Save the generic model

Export the Family Table information to a text file or to a Pro/TABLE file.

The following procedures describe each method.

Procedure 1:

1. Make sure you have closed the Family Table before you save it.
2. On the Pro/ENGINEER menu bar, click **File > Save**. The generic model is saved, and all Family Table information is saved in a file named `modelname.ptd`.

Note: Whether Family Table instances are saved depends upon the current setting for the configuration option `Save Instance Accelerator`. For more information on setting this option and what it means, refer to the Pro/HELP section on configuration options.

Procedure 2:

You can save Family Table information as an editable text file, or as a Pro/TABLE file. In either case, the file is saved with the extension `.ptd`.

To Export a Family Table to a text (.ptd) File

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. In the **Family Table** dialog box, click **File > Export Table > Textual Family Table**. The **Save** dialog box opens.
3. Click **Save** to save the Family Table information in the working directory and in the default file name, or enter a new file name and navigate to the directory in which you want to store the file. The system saves the file with the `.ptd` extension.

To Export a Family Table to a Pro/TABLE File

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.

2. In the Family Table dialog box, click **File > Export Table > PRO/TABLE file**. The **Save** dialog box opens.
3. Follow step 3 in the preceding procedure. The system saves the file with the .ptd extension.

To Import a Family Table into a Model

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Click **File > Import Table**. The **File Open** dialog box opens.
3. Navigate to and select the Family Table file you want to import. Family Table files contain the extension .ptd.
4. Click **OK**. The contents of the selected Family Table text file are inserted into the current Family Table.

Note: During import, the generic in the imported Family Table does not get imported into the current Family Table.

Working with Family Tables

About Modifying Family Tables

There are several points to remember when modifying family members. The generic serves as the template for all instances; all of its features, dimensions, and characteristics are copied each time an instance is activated (though some may be suppressed).

Whenever an instance is regenerated, the system uses the generic to determine the basic parameter values and the status of all features. It then incorporates the changes listed in the Family Table. Any feature or parameter that already exists in an instance can be suppressed, resumed, or modified.

Observe the following rules when you are creating or using Family Tables:

- You can modify any common dimension, in the generic or any instance, even if other members of the family are in use. To modify the dimension, right-click a feature in the Model Tree and click **Edit** on the shortcut menu. The updated dimensions are propagated to the other family members after the active one is regenerated. The geometry in the other family members is updated after they are regenerated.
- You can modify a table-driven dimension in any instance either by editing the Family Table or double-clicking the table-driven dimension. The **CONFIRMATION** menu appears. Click **Confirm** to change the dimension and update the Family Table. In either case, only the particular family member is affected.
- You can add a feature or component to a generic model at any time. It is added as a common feature or component to all family members. If an instance is in use at the time, it is updated after regeneration.

- You can delete a feature or component from the generic model at any time. It is also then deleted from all other family members. If an instance is in use at the time, it is updated when its window becomes active.
- You can add a feature or component to an instance at any time. It is added as an item to the Family Table. It is enabled (resumed) in the particular instance and is suppressed in the generic and in all the other instances.
- You can delete a feature or component from an instance at any time. However, in actuality, you are only suppressing it. If the feature or component was not previously in the Family Table, the system now adds it to the table. It is suppressed in the particular instance and enabled (resumed) in the generic and in all the other instances.
- Suppressing or resuming a feature or component in an instance, however, has only a temporary effect and the feature resumes its regular status after the next regeneration. Further, you cannot resume the feature or component of an instance even temporarily if it is suppressed in the Family Table.
- You can suppress or resume a feature or component in the generic model at any time. If the feature or component is table-driven, its status in the Family Table is updated immediately. If it is not table-driven, any active instances in session are updated accordingly after regeneration.
- You can add or delete parameters to or from the model from either the generic or an instance. You can also modify or designate common parameters from either the generic or an instance. You can modify a table-driven parameter in the generic or an instance, either by editing the Family Table or using the **Type** list in the **Family Table** dialog box. However, only the particular family member and its children are affected.

You can modify any common parameter, in the generic or an instance, through the **Type** list, even if other members of the family are in session. The updated parameters are propagated to the other family members, and they are all automatically regenerated.

- You can change the name of a feature, datum, dimension, or other items in the generic only. The change is immediately propagated to the other family members in use.
- You can add reference dimensions, geometric tolerances, or surface finish information to any member of the Family Table. All members, including the parent, are updated.
- In most cases, relations for an assembly family instance are copied from the generic model. However, if a generic relation refers to a model that is replaced by a different family member in the assembly instance, the instance relation references the interchanged model.

For example, if the generic assembly BLENDER contains the relation $d0:6=d1:5*2$, where 5 is the coding symbol of a generic part COVER, and this part is replaced in the assembly instance BLENDER1 by an instance COVER1 (coding symbol 8), the relation for BLENDER1 is $d0:6=d1:8*2$.

- If you want to customize an instance, you can always create a duplicate part with a different name by choosing **Save a Copy** from the **File** menu, and then work on the copy.

To Add Comments in a Family Table

You can add comments in any location within a Family Table. This is useful if you want to include important notes about the table in general, other tables that are associated with the current table, or instances in the current table.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Select an instance row below which you want to insert the comment row.
3. Click **Insert > Comment Row**. A comment row is inserted below the selected instance row and the **Comment Row** icon displays in the **Type** column. If you have not selected a row, the new comment row is inserted below first row in the table (the first row contains the generic).
4. Add comments in the row as desired, and then press ENTER. Click **OK** to close the **Family Table** and accept your changes.

To Select an Entire Row or Column in a Family Table

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. To select an entire row, select a cell within the row, and then click **Edit > Select Entire Row**.
3. To select an entire column, click on its header, or select a cell within the column and then click **Edit > Select Entire Column**.


To Hide or Show Family Table Columns


1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. To hide a column or columns, do one of the following:
3. Click any cell in the column(s) you want to hide. The columns you select do not have to be contiguous.

Click **Edit > Hide Columns**. The selected columns are hidden from view.

4. To show all hidden columns, click **Edit > Unhide All Columns**. The hidden columns appear in the Family Table.

To Cut, Copy, and Paste Items in a Family Table

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Select the cells that you want to cut or copy. You must select a rectangular region of cells; you cannot select a noncontiguous group of cells.
3. Click **Edit > Cut Cell**, or click .

4. Navigate to the location where you want to paste the cut or copied cells, and then click **Edit > Paste Cell**, or click . You need select only the cell in which you want to paste the first (upper left) cell in the rectangular region. The cells are pasted in their original relative positions.

To Print a Family Table

You can save a Family Table in an output file suitable for printing. The output file is in text format and is laid out in the same format as you see on the screen.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Click **File > Export Table > Textual Family Table**. The **Save** dialog box opens.
3. Type the output file name, then click **Save**. The dialog box closes. The file is saved as `filename.ptd`, where `filename` is the name that you typed.
4. Open the file and print it as you would any other text file.

To Highlight Family Table Items in the Graphics Window

While working with a Family Table, you can show table-driven dimensions and highlight table-driven features and assembly members in the graphic window.

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. In the Family Table dialog box, select the cells in the columns that you want to highlight.
3. Click **Edit > Highlight on Screen**. The selected items are highlighted (redrawn in a different color) in the graphics window.

To Change Column Location in a Family Table

1. Click **Tools > Family Table**. The **Family Table** dialog box opens.
2. Select the two columns you would like to swap. The columns must be contiguous.
3. Click **Edit > Swap Two Columns**. The two columns swap positions.

Creating a Drawing of a Family Member

You can create a drawing of the generic model, as well as any member of its Family Table. You must enter the name of the instance when adding the first view to the drawing. The instance must be in use or listed in an instance index file. If you do not know the instance name, retrieve the instance using standard table retrieval methods and look at the name in the lower-right corner.

Note: The drawing has the instance and generic name stored with it, and is able to retrieve it as long as the generic model is in the current directory or search paths.

When you create part simplified representations, or assembly features that intersect a part, instances are created that do not necessarily appear on the part Family Table. The following operations cannot be performed when a hidden instance is in use:

- Feature Redefine
- Feature Reorder
- Feature Del Pattern
- Feature Reroute
- Feature Delete/Suppress/Resume (has temporary effect)
- Feature Group Unpattern
- Component Package (applicable only to affected subassemblies)
- Component Copy (applicable only to affected subassemblies)
- Relations Add/Edit
- Program Edit/Instantiate
- Edit Explode (applicable only to affected subassemblies)
- Declare
- Offset X-Section Modify
- Assembly Feature Intersect (applicable only to affected subassemblies)
- Component Replace (does not allow components to be replaced if they are intersected by or referenced by assembly features).
- When adding a dimension to a Family Table of a subordinate table-driven user-defined feature (UDF), the symbols of the Family Table items are driven by the original model for the subordinate UDF.

Renaming Family Tables, Assemblies, and Drawings

Renaming Family Table Files

- If you rename an object that resides in the current directory and is not an instance of a table-driven family, all versions of the object are instantly renamed.
- If you rename an object that is the generic of a table-driven family, the instance index file of the directory is updated to reflect the change.
- If you retrieve an object from a non-current directory, and then rename it, only the object in memory is renamed. If you later save the renamed object, it is saved in the current working directory. If it is the generic of a table-driven family, the instance index file of the directory is updated to reflect the addition of the renamed object and its instances.
- You can rename an instance of a table-driven object only by editing the Family Table of its generic either while the generic is in session (and then saving the generic) or after you retrieve the instance into Pro/ENGINEER.

- If you retrieve an instance from the current directory and then rename it, its name is placed immediately in the Family Table of its generic and in the instance index file of the directory.
- If you retrieve an instance from a non-current directory and then rename it, only the instance in memory is renamed. If you later save the renamed instance, its generic and the Family Table of its generic both are updated in the directory that contains them. The instance index file of that directory is also updated appropriately.

Renaming Assembly Components and Drawing Models

If an assembly component or a model is renamed, it may cause failures in assemblies and drawings that reference it, because Pro/ENGINEER may not be able to locate the (original) file. If the assembly or drawing that contains the part being renamed is active in memory, it is updated automatically.

To save the updates, however, you must save the assembly or drawing after renaming it. Alternatively, you can set the configuration option `rename_drawings_with_object` to `part`, `assem`, or `both`.

When you copy an assembly, you can rename any or all of the components that make up the assembly.

To Delete a Family Table

1. Open the generic model.
2. Click **Tools > Family Table**. The **Family Table** dialog box opens.
3. Click **Edit > Delete Entire Table**.
4. Confirm the deletion. Pro/ENGINEER deletes all of the Family Table contents except the instance name and type for the generic.

Note: You cannot undo the deletion once it has been confirmed.

Tip: If You Delete a Family Table Inadvertently

After you delete a Family Table, you cannot undo the deletion. If you delete a Family Table inadvertently, the following are some suggestions for restoring it.

- If you have not saved the model since deleting the Family Table, you can erase the model from current memory, close the window and then retrieve it. The Family Table will be intact.
- If you have saved the model since deleting the Family Table, you can retrieve an earlier version of the model.
- If you created the Family Table using Excel, you can link back to the original Excel file.
- Before you delete the table, you can save it as a Pro/TABLE (.ptd) file using **File > Export Table > PRO/TABLE File**. Then, delete the table and close the file.

When you reopen the file, you can then import the `.ptd` file using **File > Import Table**.

To Edit a Family Table Outside of Pro/ENGINEER

When you save a generic part and its features and dimensions in a Family Table by choosing **Save** from the **File** menu, the system creates or updates an external generic `modelname.ptd` file. At that moment, the `modelname.ptd` file contains the same Family Table information as is stored internally (within the part database).

You can edit a Family Table outside of Pro/ENGINEER, using any text editor from the operating system level to edit the `.ptd` file. Any modifications you make to the `.ptd` file are not automatically copied to the Family Table within Pro/ENGINEER. For example, if you delete an instance during the `.ptd` file edit, when you later retrieve the model in Pro/ENGINEER, the system detects that the table is missing an entry and asks you whether you want to similarly modify the Family Table in Pro/ENGINEER.

You can also edit the `modelname.ptd` file in a different window outside of Pro/ENGINEER while working with the part containing the Family Table in Pro/ENGINEER. In this case, if you save the part by choosing **Save** from the **File** menu, the internally stored Family Table takes precedence over the external `modelname.ptd` file, if that external file exists in the current directory.

When you retrieve a part containing a Family Table while an external `modelname.ptd` file exists in the current directory, the external file takes precedence over the internally stored table.

Pro/ENGINEER no longer reads and writes to `.ptd` files by default. You can now use the configuration file option `read_famtab_file_on_retrieve` to choose whether you want Pro/ENGINEER to read and write to `.ptd` files.

To Edit a Family Table Using Microsoft Excel

You can edit Family Table information in Microsoft Excel. This functionality is available only if you are using a Windows environment and have Microsoft Excel installed.

1. With the generic model active, click **Tools > Family Table**. The **Family Table** opens.
2. Navigate to the appropriate **Family Table**.
3. Click **File > Edit with Excel**. Pro/ENGINEER opens the Family Table in Microsoft Excel.
4. Edit the Family Table. In Microsoft Excel you can:
 - Insert new rows.
 - Create additional instances.
 - Reorder items or instances.

- When finished, click **File > Update Pro/E**. Microsoft Excel checks the Family Table for acceptable values, names, and so forth. If an error is found, Microsoft Excel prompts you to fix the error. If no errors are found Microsoft Excel saves the changes to the Family Table and returns you to the **Family Table** in Pro/ENGINEER.

Note: To exit Microsoft Excel without saving the changes to the family table, click **File > Close**. If Microsoft Excel prompts you save the changes, click **No**.

Glossary

Glossary for Fundamentals

Term	Definition
annotation orientation	Refers to a plane or a parallel plane in which the annotation lies, the viewing direction, and the right direction or text rotation.
annotation plane	A datum plane that defines the orientation of the annotation item with respect to the 3D model. The annotation item is not actually placed on the Annotation plane, but on an imaginary plane parallel to the Annotation plane, and facing in the direction of the front of the Annotation plane. The actual location of this imaginary plane with respect to the model is determined by the first selected attachment for the annotation.
chain	A series of multiple edges or curves that you can connect by a common vertex or by tangency.
child	An object, such as a part or a feature, that is dependent on another object for its existence.
clip	The process of removing portions of lines or surfaces that reside outside the view volume.
clipping plane	A trimming plane. Objects or parts of an object are visible on one side of the plane, while invisible on the other side.
closed loop	The condition required for a group of edges to form a face (surface) on an object. The edges must be connected end to end, forming a continuous path around the perimeter of the face.
collector	An interface tool for collecting single or multiple references when defining geometry.
cross section	The section of an objection as it would appear if cut through by an intersecting plane.

Term	Definition
dashboard	A context-sensitive area at the bottom of the Pro/ENGINEER window that guides you through the modeling process. As you select geometry in the graphics window and set your preferences, the dashboard narrows your available options, so you make only targeted modeling decisions. The dashboard consists of the dialog bar, slide-up panels, message area, and the control area.
datum	Datums are features without geometry which are displayed in the Model Tree. You can create datums as points, curves, lines, planes, coordinate systems and use them for placing components or dimensioning.
depth cue	Quick method of adding a feel of space to the scene or realistic depth effects.
dimension bound	When you design a model, you expect the actual part dimensions to be within certain predetermined limits of size. In Pro/ENGINEER, these limits of size, that is, upper and lower dimension boundaries, are defined as "Dimension Bounds."
dimension line	A thin, solid line that shows the extent and direction of a dimension. Dimension lines are paired with the number representing the physical size of the feature, and extension lines are used to associate the dimension to the corresponding feature on the drawing.
dimension witness line	Thin, solid lines that extend dimensions beyond the outline of a view so that you can read the dimensions easily. These lines are perpendicular to the dimension line, that is, the dimension leader arrows, and indicate which feature is associated with the dimension.
driven dimension	Dimensions in a drawing file derived from the 3D model. Driven dimensions reflect changes to part or assembly geometry, but you cannot modify their values in the drawing. The association is one-way; from the model to the drawing.
driving dimension	A dimension that when modified in the drawing is also modified in the 3D model from which it originated.
dynamic preview	A high-quality preview of geometry that dynamically simulates the result of direction manipulation of the model.
face	Generically refers to a planar surface of a geometric model, or a specific side of a geometric model which is oriented in a particular direction. For example, a machining tool or a

Term	Definition
	viewer.
Family Tables	<p>A collection of parts (or assemblies or features) that are essentially similar, but deviate slightly in one or two aspects, such as size or detail features.</p> <p>For example, wood screws come in various sizes, but they all look alike and perform the same function. Thus, it is useful to think of them as a family of parts. Parts in Family Tables are also known as table-driven parts.</p>
feature	Any self-contained set of elements that can be defined and stored with the model and which has an impact on the geometry. A feature can be physical (hole, draft), procedural (for example, machining strategy in Expert Machinist), or computational (analysis). Features are listed on the Model Tree.
generic model	The lead model that owns a family table, consisting of columns of table items (dimensions, feature, and so on) for which each member of the family may have a differing value.
graphics area	The portion of the Pro/ENGINEER window used to display the model.
instance	An instance of an object, is the same as an exact duplicate of the object. An instance is really an object that shares the exact same properties as the original. It will always stay the same as the original regardless of what changes are made to the original.
instance accelerator file	<p>A special file containing saved instances of a part or assembly, that helps reduce the time required to retrieve an instance of a part or assembly from disk by eliminating the need to first retrieve the generic into memory, selecting the instance according to the family table, and then regenerating.</p> <p>Instant accelerator files that store part instances are saved as <code>instancename.xpr</code> and that store assembly instances are saved as <code>instancename.xas</code>.</p>
layer	User-defined logical subdivisions of data in a CAD/CAM database that may be viewed on the graphics screen individually or overlaid and viewed in groups.
layout	Functionality allowing for the creation of a 2D notebook that contains conceptual sketches to begin the design

Term	Definition
	process and maintain design intent.
line	A geometric element that connects two points in space. Although a line itself is 2D in nature, it may connect points in 3D space. Lines are typically classified as either straight (linear) or curved. Lines are the most prominent element in technical drawings, defining edges of objects, indicating symmetry, relating text elements to geometric elements, creating borders, and so on.
locked instance	An instance that cannot be deleted or modified.
loop	A closed chain of edges on a part surface. For example, the contour of a hole or slot.
mirror	Duplicate by symmetry about an axis.
Model Tree	A Model Tree is a list of every feature in a part file including the datums and coordinate system. Pro/ENGINEER displays the model structure in a hierarchical tree format with the current part or assembly as root object at the top of the tree. The subordinate parts or features are displayed below the root object.
pan/zoom state	The state in which you can change the position of or enlarge the model to view more details.
parametric	A term used to classify curves for which the path is described by a mathematical function rather than a set of coordinates. A parameter within the function (often specified as u or v) is varied from 0 to 1 to define all the coordinate points along the curve.
parent	An item in a model that has other items (children) dependent upon it for dimensions or shapes. If a parent is deleted, all dependent items (children) are deleted as well. See child
parent/child relationship	An association between the instances of one or more entity types in which one entity is dependent on the other. The entity that is dependent on the other is called the child entity while the entity on which the other entity depends is called the parent entity. The association between the parent entity and the child entity is called a parent/child relationship.
planar	Having a flat, two-dimensional characteristic.
plane	A region of space defined by a minimum of three noncoincident points in space. For the simplest type of

Term	Definition
	plane surface, all points can be described by two coordinate axes; that is, the plane has no curvature.
reference dimension	A reference dimension is a calculated size used to show the intended design size of a part. A numerical value, enclosed in parentheses, provided for information only and not used in the fabrication of the part. Drawings made to older standards may use REF placed next to a reference dimension, instead of using parentheses.
regenerate	Restore, as in function or nature, to a former or original state.
solid	3D volume that is filled with material. Not interrupted by a break. For example, a continuous line.
solid geometry	The geometry of 3D objects, such as cylinders, cubes, and spheres, and their relationships.
table-driven	Family Tables are collections of parts, assemblies, or features that are essentially similar, but deviate slightly in one or two aspects, such as size or detail features. These parts in Family Tables are known as table-driven.
tangent edge	A tangent edge is the intersection at which two surfaces become tangent.
target model	A model to which data that is measured or generated from the source model is to be sent.
tolerance stack-up	The cumulative tolerance that occurs when Pro/ENGINEER uses a series of dimensions with tolerances to dimension a certain feature in a part or an assembly member.
trajectory	The course of the tool and the table motion consisting of straight lines and circular arcs.
translate	To move an object in a linear fashion.

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