



MODELING FUNDAMENTALS

Modeling Fundamentals

Selection Methods

In order to make modifications to features, geometry, or components, users must first select those entities. The following selection methods are the most useful when working with models and drawings.

Direct Selection

Direct selection occurs when you directly place the cursor over the desired feature, geometry, or component and click it. Direct selection can be done through either the graphical interface or the model tree. Use the CTRL key while clicking to select multiple items and use the SHIFT key while clicking to select a range of items, just as you would with Microsoft Windows.

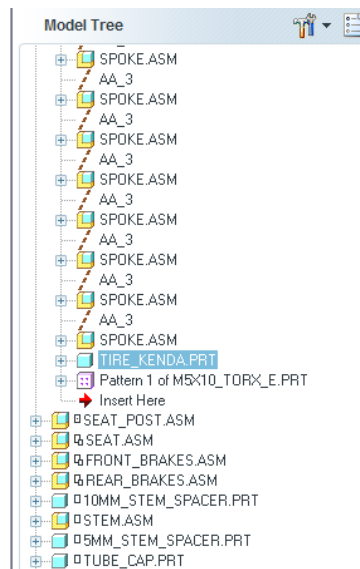


FIGURE N

Query Selection

Query selection enables users to select features, geometry, or components that may be hidden behind other features, geometry, or components. To use the query selection, place the cursor over the approximate area of the feature, geometry, or component, and use the right mouse button to toggle through each entity, almost like pushing your finger through the screen until you touch the object you want. The selections will pre-highlight in a cyan color and once the desired entity is pre-highlighted, simply left-click to select.

For example, the part called “tube.prt” needs to be selected to make a modification, but it is hidden by another part called “HT.prt”. This is a perfect example of where you would use the query selection tool to select tube.prt. Figure O, below, illustrates this selection process.



FIGURE O

Pick from List Selection

The “Pick from List” selection is similar to query selection, but it lists all available features, geometry, or components instead of querying for them. To use the Pick from List selection, place the cursor over the approximate area of the feature, geometry, or component, and *hold* the right mouse button down. A context-sensitive menu will display and the Pick from List selection can be chosen. This will display a dialogue box that the user will then be able to choose entities from (see below).

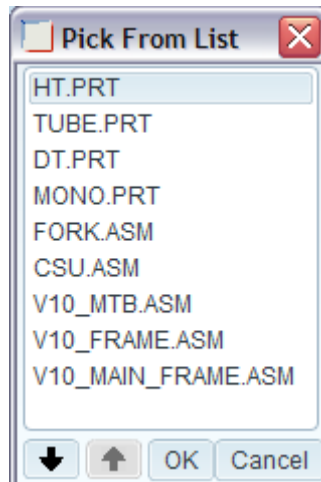


FIGURE P

Search Tool Selection

The Search Tool selection allows the user to filter what is being searched by any number of criteria. Example criteria include:

- **Look For:** Specifies the desired entity type, such as datum planes, components, axis, etc. for which the user wants to search.
- **Look By:** Specifies the desired type of item by which the user wants to search.
- **Name:** Specifies the full or partial name of the item for which the user wants to search.

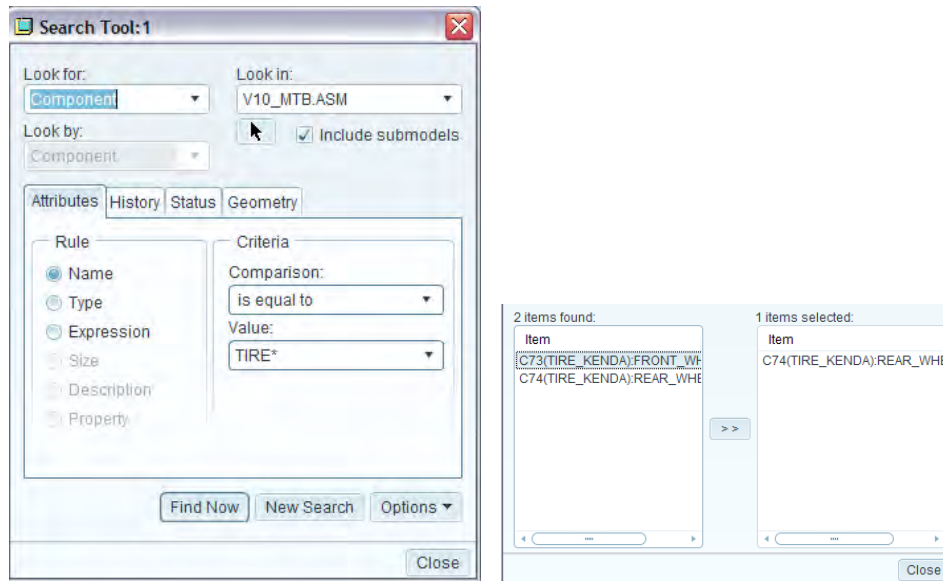


FIGURE Q

Search Filters

Search filters allow users to narrow down the type of item that is being selected. These filters are defined by and change based on filtering within a part, assembly, or drawing. By default, Pro/ENGINEER is set to the “smart filter”, where all items are selectable through a nested process. A nested process allows a user to select any geometry appropriate for a given entity.

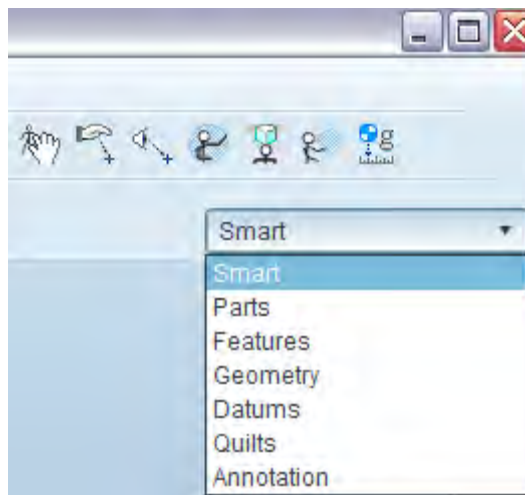


FIGURE R

De-selection Methods

Users can de-select features in one of three easy ways. 1) De-select specific geometry or components by using the CTRL key and selecting the item(s) in the Model Tree or in the model itself, 2) de-select all items by clicking in the graphics window background, or 3) de-select all by going to Edit > Select > Deselect All.

Editing Methods

Edit

The Edit command allows users to alter dimensions of a component or feature that has been selected. This command is available through the model tree, pop-up menu, or drop-down menu. After selecting the feature or component and selecting Edit, the associated dimensions will display in the graphics window. These dimensions can be easily edited directly from the model by double-clicking on the dimension. Once a new dimension has been entered, update the geometry with the regenerate command.

A second method for changing dimensions is to choose a value from the “Most Recently Used” drop down list. Pro/ENGINEER records the most recent values from the current session, as it assumes those values may be used again.

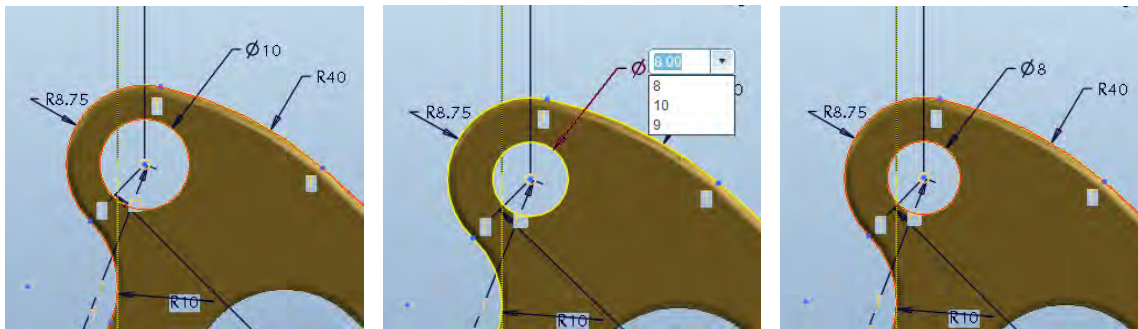


FIGURE 5

Edit Definition

Whereas Edit is limited to changing only dimensional values, Edit Definition allows you to modify anything about the feature, such as size, shape, location, references, or options. Models can be modified by using the following tools:

1. Dashboard: this is a graphical interface that allows users to change the type, size, shape, and location of the feature.
2. Drag handles: this is a tool that allows users to dynamically alter the feature or component in the graphical interface.
3. Context-sensitive right mouse menus from a drag handle. These menus are available through the right mouse button, which provides a list of available actions that can be taken by the user.

Regeneration

The Regenerate button updates the model after changes by recalculating the model's geometry. This is usually done immediately after editing the dimensional value(s) of a feature or component. One exception is with Edit Definition, which will regenerate the model automatically after completion.



Undo/Redo Functionality

You have the ability to Undo or Redo most steps in building a model (in the current session of Pro/ENGINEER). Steps can include creating, deleting, editing, redefining, suppressing, resuming, creating and deleting patterns, and reordering features or components in the model tree. These operations are stacked sequentially in memory, and will not be available after exiting Pro/ENGINEER.

Activating Models

To make changes to a component in assembly mode, the model must first be activated. Right-click on the component and select “activate”. This allows users to modify features of a component and visualize what affects it may have to the surrounding components.

- A green symbol designates an active component.
- All other non-active components are grayed out.
- Text in the graphics window will state the active component.



Deleting Items

If a feature or component is no longer needed, you simply delete it. Keep in mind, if you delete a feature that is a parent of other features, the delete function will need to delete children features as well.

Suppressing or Resuming Items

Suppressing a feature or component prevents it from displaying in the model and removes it from the regeneration cycle. Unlike a deleted item, a suppressed item can be restored by resuming it.

- A suppressed item is denoted by a black rectangular box next to the feature or component name in the model tree.
- Since suppressing items removes them from the regeneration cycle, regeneration speeds increase.
- To rename a suppressed item, right-click the object in the model tree and select resume.



FIGURE T

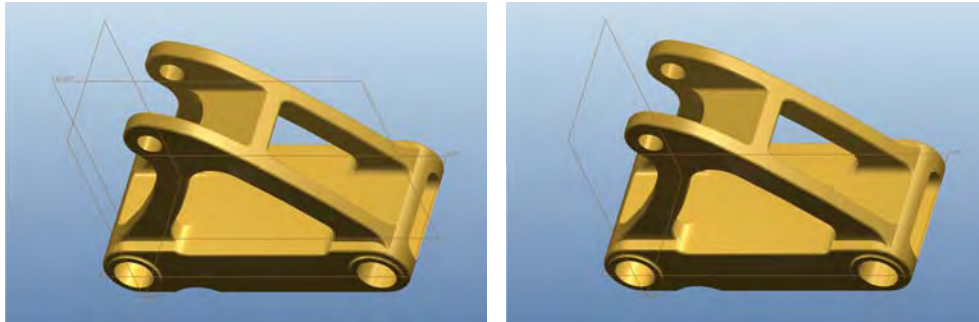
Renaming

Pro/ENGINEER gives all features a default name, so it is a good practice to rename features to help describe what they are. To rename, select a feature in the model tree or graphics window, right-click, and select rename.

Component Visibility

Hide/Unhide

The Hide tool enables users to temporarily control component or feature display. The Hide tool removes the display of non-solid geometry such as datum planes, axis, points, sketches, etc. from the graphics window to allow for easier selection and visualization while working with parts and assemblies. These items can then be unhidden using the Unhide tool to restore the non-solid geometry. Unlike suppress, resume, and delete tools, hide and unhide do not affect parent/child relationships.



FEATURE HIDE



COMPONENT HIDE

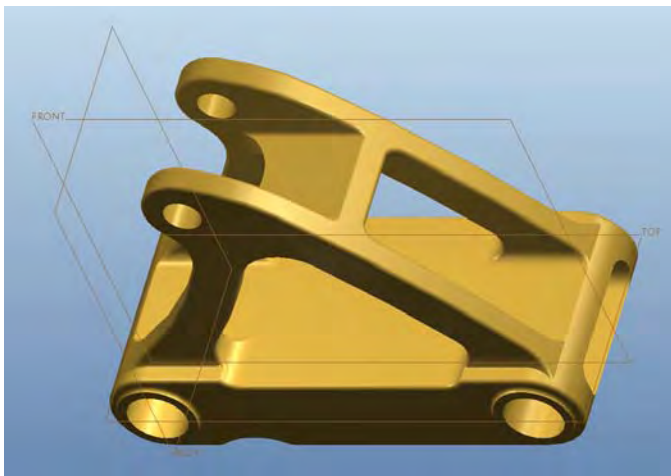
Layers

Layers allow users to organize features, datum planes, parts in assemblies, and other layers. By utilizing layers, users can perform actions collectively on those layers, such as hide/unhide, suppress/resume, or delete. This is a powerful productivity tool when working on large part and assembly files.

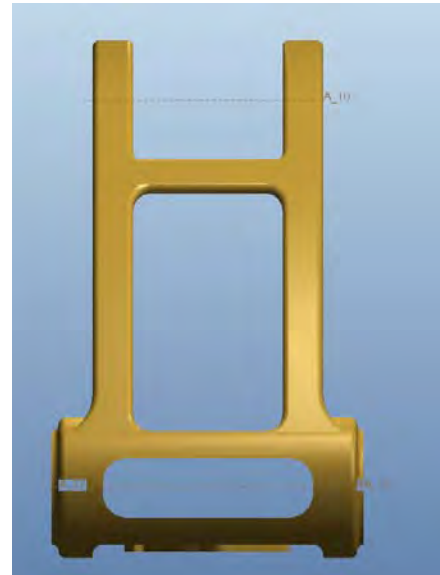
Datum Features

Datum features are required and used for dimensional references, feature placement, and assembly references. They are denoted by brown colored entities, which help users distinguish them from other features in a model or assembly. There are five types of datum features:

- Datum Planes
- Datum Axes
- Datum Points
- Datum Coordinate Systems
- Datum Curves



DATUM PLANES



DATUM AXIS

Basic Sketch Based Features

Sketch based features allow users to create features with irregular profiles. Sketch based features require a 2D cross section where dimensions and design intent are embedded. The following sections define some of the most commonly used sketch-based features.

Extrudes

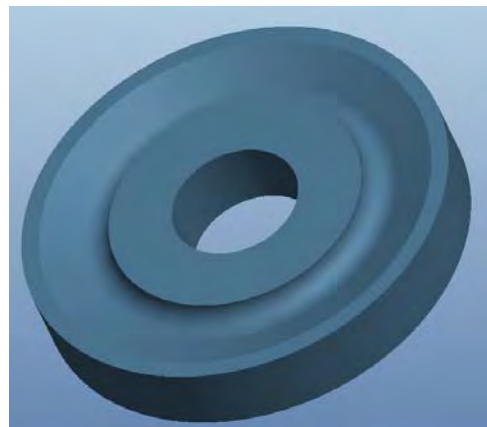
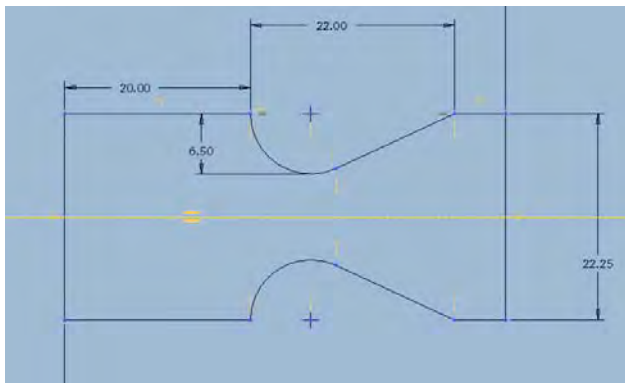
The extrude tool extrudes material perpendicular to the 2D section to create a protrusion, cut, or surface, allowing users to create or cut solid models. With the extrude tool, users can define the type of extrude (cut, surface, or protrusion), direction of extrude, and depth. It is a best practice to create closed-loop sketches for more robust models.



EXTRUDE

Revolves

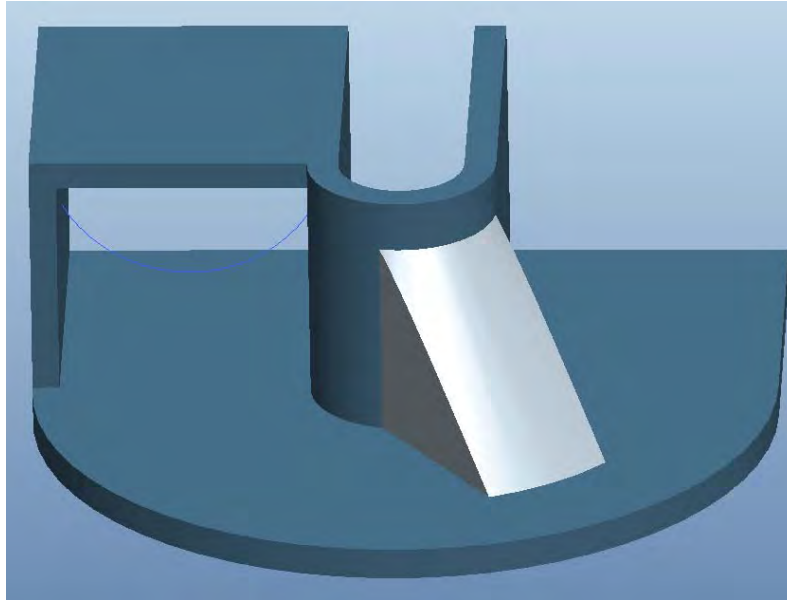
Similar to an extruded feature, a revolve feature is defined by its 2D section. However, a revolve feature rotates about a centerline or a datum axis. This tool can add or remove material and create surfaces. Users can specify an angular depth and define the direction of the revolve feature. It is a best practice to use closed-loop sketches here too for more robust models.



REVOLVE

Ribs

Rib geometry is used to strengthen parts. Ribs are similar to extruded features, but require the use of open-looped sketches that will conform to existing planar or cylindrical surfaces. Once the open-loop sketch has been chosen, the user can define the thickness of the rib and in what direction the material will be extruded. The user can choose from either side of the sketch or symmetric around the sketch.



RIB

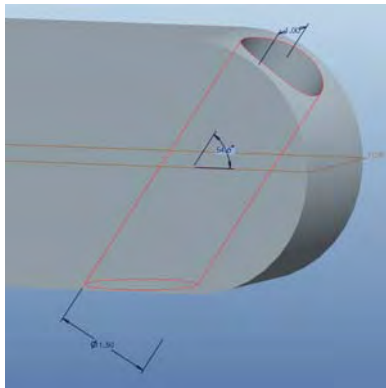
Direct Features

Direct features allows users to quickly add features, such as holes, rounds, chamfers, drafts, and shells on pre-defined edges and surfaces. Direct features are fast, because there is no sketch required to define the feature. Direct features not only use edges and surfaces, but can also use datum planes and axis as references.

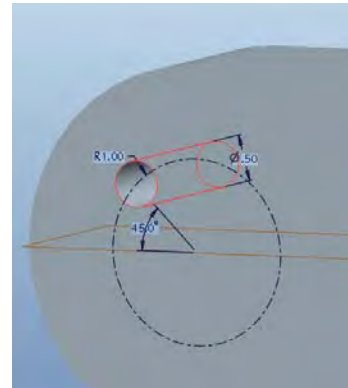
Holes

There are several different methods in creating holes:

Type	Placement Reference	Offset Reference	Number of Offset References
Linear holes	<ul style="list-style-type: none"> Datum Plane or surface 	<ul style="list-style-type: none"> Data plane or surface Edge Datum axis 	2
Coaxial holes	<ul style="list-style-type: none"> Datum Axis Surface or datum plane 	<ul style="list-style-type: none"> none 	0
Radial holes on a cylindrical surface	<ul style="list-style-type: none"> Cylindrical surface 	<ul style="list-style-type: none"> Datum plane or surface for offset Datum plane or surface for angle 	2
Radial or diameter holes on a planar surface	<ul style="list-style-type: none"> Datum plane or planar surface 	<ul style="list-style-type: none"> Datum axis Datum plane or surface for angle 	2



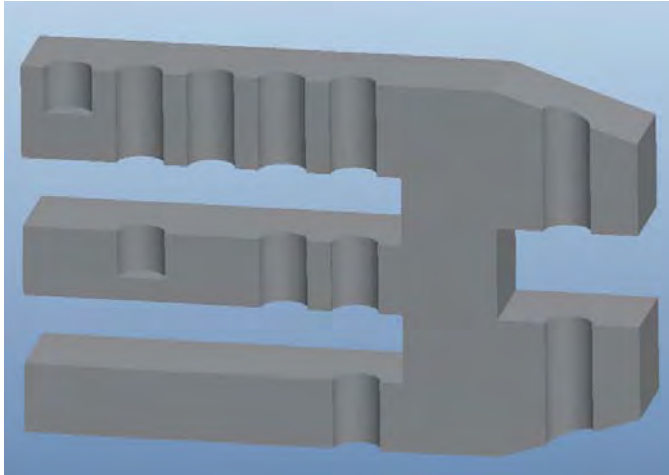
RADIAL HOLES ON A CYLINDRICAL SURFACE



RADIAL OR DIAMETER HOLES ON A PLANAR SURFACE

Users can specify the following depths for holes:

- Blind: user defined variable depth.
- Symmetric: hole will bore equally on both sides.
- To Next: hole will stop at the next surface.
- Through Until: hole will stop at a selected surface, where the hole must pass the selected surface.
- To Selected: hole will stop at a selected surface, where the hole does not have to pass the selected surface.
- Through All: the hole will bore through all material.
- Side 1/Side 2: hole depth can be independently controlled on both sides.



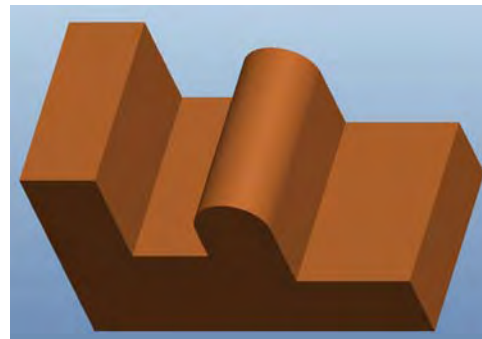
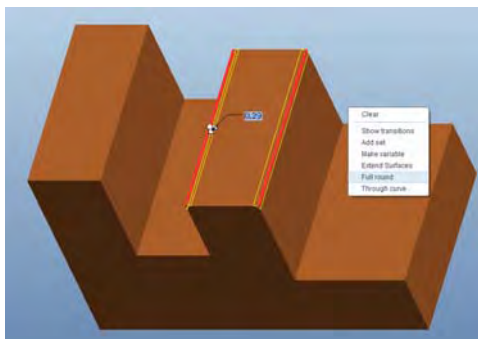
From left to right:

1. Blind = 0.5 unites
2. Blind = 2.25 unites
3. To Next
4. To Selected
5. Through All
6. Symmetric and through all for side 1 and side 2

Rounds

Rounds smooth transitions between existing geometry by adding and removing material. There are four types of rounds:

- Edge round: requires the selection of edge(s), which can be selected individually or by using a variety of edges.
- Surface – Edge round: requires the selection of a surface and an edge.
- Full round: requires a selection of two edges and then converted into a full round or two surfaces, with a third surface that will be replaced by the round.
- Surface – Surface round: requires the selection of two surfaces.

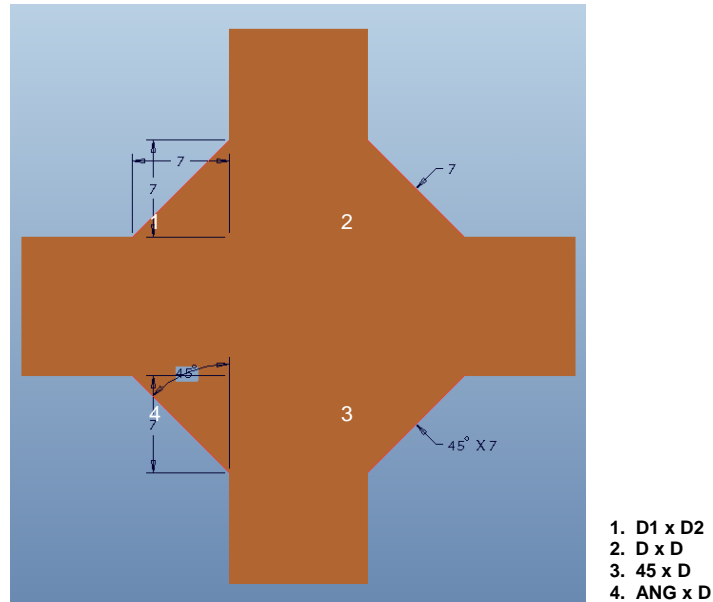


FULL ROUND

Chamfers

Very similar to rounds, chamfers create a beveled surface between surfaces or edges by adding or removing material. There are five types of chamfers:

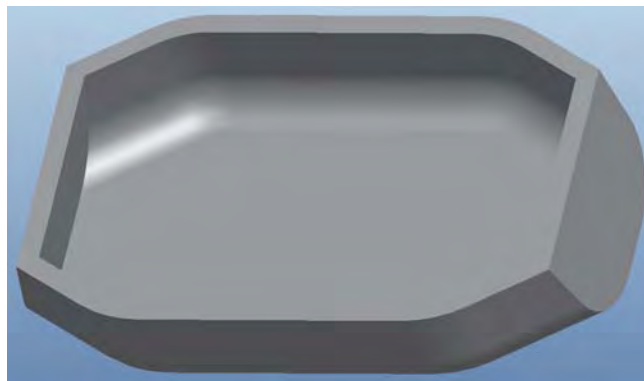
- D x D: the chamfer is driven by one dimension.
- D1 x D2: the chamfer is driven by two dimensions.
- ANG x D: the chamfer is driven by a linear and angular dimension.
- 45 x D: the chamfer is driven by a linear dimension at 45-degree angle. This chamfer style only works for perpendicular surfaces.



Shells

The shell tool hollows out the inside of a model, leaving a wall thickness to represent the model, which is very useful in the creation of molded or cast parts. There are two steps in creating a shell.

- Select a Surface for Removal: By default, when selecting the shell tool, it will automatically create a hollowed model that is enclosed by all surfaces. If you would like a surface removed, you must select a surface(s) for exclusion.
- Thickness: Specify the thickness of the wall. By default the system will assume that all wall thickness will be the same, but the user can choose to define non-default wall thicknesses.



SHELL

Assembly

Assemblies allow users to arrange parts and sub-assemblies, as most products are comprised of numerous components. New assemblies can be created by selecting File > New > Assembly. Components can be assembled by constraints or connections.

Constraints

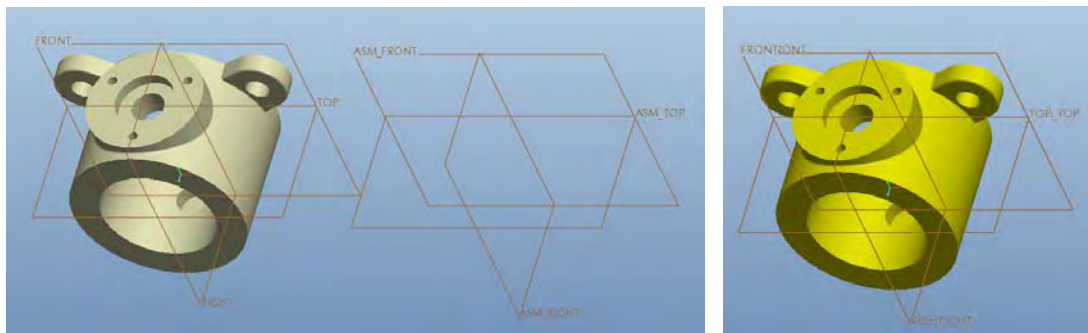
Constraints determine the location of each component placed in an assembly. Most constraints are defined by selecting a reference on the component and a reference on the assembly. These constraints are defined one at a time assuming that all degrees of freedom are locked. The active constraint tag is highlighted in yellow.

The most commonly used constraints are:

- Default Constraint
- Insert Constraint
- Mate Constraint
- Align Constraint
- Automatic Constraint

DEFAULT CONSTRAINT

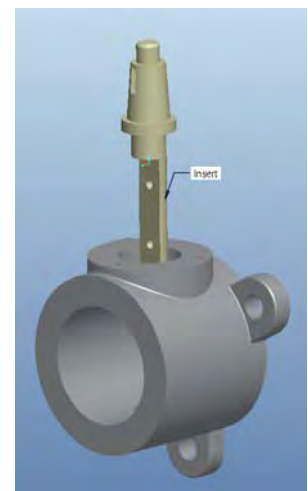
As a best practice, it is important to define the first component of an assembly with the default constraint. This enables users to align the assembly's internal system-created coordinate system with the component's internal system-created coordinate system.



DEFAULT CONSTRAINT

INSERT CONSTRAINT

The insert constraint is used to coaxially position two revolved surfaces when axes are not available for selection. For example, a bolt is constrained with an insert constraint into a bolt hole. Users can use cylindrical surfaces and conical surfaces to define an insert constraint. Note that the insert constraint will only constrain the references coaxially and does not slide the component into the other.



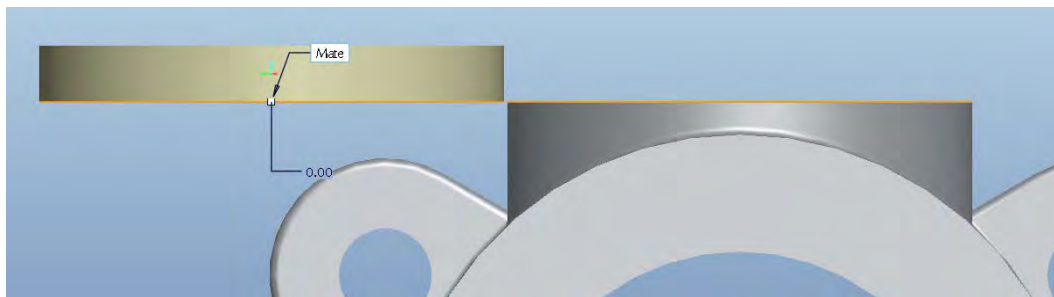
INSERT CONSTRAINT

MATE CONSTRAINT

The mate constraint positions two surfaces, datum planes, or conical surfaces on the same plane, facing the opposite direction.

The mate constraint can be defined by the following offset options:

- **Offset:** specifies an offset value between the surfaces or datum planes.
- **Oriented:** selected surfaces are kept parallel to each other facing the each other, but do not care about the distance between the two surfaces.
- **Coincident:** specifies an offset value of zero between the surfaces or datum planes.
- **Angle Offset (Align Angle):** specifies a rotational angle between the surfaces or datum planes.



MATE CONSTRAINT

ALIGN CONSTRAINT

The align constraint positions two surfaces or datum planes on the same plane, facing the same direction. In addition to surfaces and datum planes, users can use the align constraint on axes, points, or edges. However, to use the align constraint, the references both must be the same type, for example, surface to surface, point to point, axis to axis, etc.

The align constraint can be defined by the following offset options:

- **Offset:** specifies an offset value between the surfaces or datum planes.
- **Oriented:** selected surfaces are kept parallel to each other facing the same direction, but do not care about the distance between the two surfaces.
- **Coincident:** specifies an offset value of zero between the surfaces or datum planes.
- **Angle Offset (Align Angle):** specifies a rotational angle between the surfaces or datum planes.



ALIGN CONSTRAINT

AUTOMATIC CONSTRAINT

The automatic constraint allows Pro/ENGINEER to automatically determine what type of constraint will be used to assemble components based on the references selected and component location or orientation.

Mechanism Connections

Although similar to constraints, mechanism connections allow users to create dynamic connections that can help simulate motion between moving parts. The following table outlines the three most commonly used connections.

Connection Type	References	Rotational DOF	Translation DOF	Total DOF	Example
Pin	<ul style="list-style-type: none">• Axes or cylindrical surfaces to enable rotation.• Datum planes or planar surfaces to constrain translation.	1	0	1	Hinge on a door.
Slider	<ul style="list-style-type: none">• Axes or cylindrical surfaces to enable translation.• Datum planes or planar surfaces to constrain rotation.	0	1	1	Elevator door.
Cylinder	<ul style="list-style-type: none">• Axes or cylindrical surfaces to enable rotation.	1	1	2	Pen cap over pen.