This chapter introduces extruded features, a concept associated with basic modeling fundamentals. Within Pro/ENGINEER, the Extrude option is common among the Protrusion and Cut commands. Additionally, this chapter will introduce the Redefine command, feature modification techniques, and datum construction. Upon finishing this chapter, you will be able to

- Model solid features as extruded protrusions.
- Remove material from features using extruded cuts.
- Modify feature dimension values using the Modify command.
- Modify feature definitions using the Redefine command.
- Create datum planes.

**DEFINITIONS**

**Base feature** The first geometric feature created in a part. It is the parent feature for all other features.

**Child feature** A feature whose definition is partially or completely referenced to other part features. A feature referenced by a child feature becomes a parent of this feature.

**Cut** A negative space feature created from a sketched section.

**Definition** A parameter of a part. An example of a definition of a hole feature would be the depth of the hole.

**Protrusion** A positive space feature created from a sketched section.

**Negative space feature** A feature created by removing material from a model. Examples of negative space features include holes, cuts, and slots.

**Parent feature** A feature referenced by another feature.

**Positive space feature** A feature created by adding material to a model. Examples of positive space features include protrusions and ribs.

**FEATURE-BASED MODELING**
Parametric design packages are often referred to as feature-based modelers. A feature is a subcomponent of a part that has its own parameters, references, and geometry. Geometry (see Figure 2–1) is the graphic description of a feature. Geometry can be sketch-defined or predefined. Sketch-defined features consist of sketched sections that are protruded or cut to form either positive or negative space. Predefined geometry has a common section such as a hole, round, or chamfer. Parameters are the dimensional values and definitions that define a feature. A hole may have a diameter of 1 inch and can be extruded completely through all existing features. The diameter is a parameter, as is the through-all definition. Parametric modeling packages allow users to modify parameters after the feature has been modeled. This is one of the unique properties that separates parametric modelers from Boolean-based modelers. References are ways that features are related to other features in a part or assembly. Examples of references include axes, sketch planes, placement planes, reference planes, and reference edges. The surface of one feature may serve as the sketch plane for a second feature. The edges of the first feature may also serve as reference lines for parameters defining the second feature. In both examples, the first feature is a parent of the second feature.

**Parent-Child Relationships**

Parametric models are composed of features that have established relationships. Features build upon other features in a way that resembles a family tree, hence the phrase parent-child relationship. Actually, a history tree of the relationships between features in a Pro/ENGINEER model resembles a web. The first feature created in a part is the center of the web and is the parent feature for all features. Child features branch off from the base feature and themselves become parent features. Unlike a typical family tree, a child feature may have several parent features.

Parent-child relationships can be established between features implicitly or explicitly. Implicit relationships can be established through the adding of a numeric equation using the Relations option. An example of this would be making two dimensions of equal value. In this process, one dimension governs the value of another. The feature with the governing dimension is the parent feature of the feature with the governed dimension. Care should be taken when modifying a feature that has a dimension that governs another. If a parent feature is selected for deletion, Pro/ENGINEER will provide an error message requesting an action to be accomplished to satisfy the void relationship. The user has the option of deleting, modifying, redefining, or rerouting the relationship.

Explicit relationships are created when one feature is used to construct another. An example would be selecting a plane of one feature as the sketch plane for a second feature. The new feature will become a child of the feature being sketched upon. Another similar example of an explicit relationship would be using existing feature edges within the sketcher environment to create a new feature. By specifying references while sketching, these selected references will create a relationship between the feature being sketched and the existing feature being referenced. The new sketched feature becomes a child of any referenced feature.

**Protrusions and Cuts**

The procedures for performing a Protrusion and Cut in Pro/ENGINEER are virtually identical. The primary difference between the Protrusion command and the Cut command is that a protrusion is a positive space feature, while a cut is a negative space feature. When you protrude a feature, you actually create a solid object. With the Cut command, an extruded feature removes material from existing features.

The menu structure for both commands is similar. For both, the following options exist:

**Extrude**
The Extrude option sweeps a sketched section along a straight trajectory. The user draws the section in the skinner environment and then provides an extrude depth. The section is protruded the depth entered by the user.

**REVOLVE**

The Revolve option sweeps a section around a centerline. The user sketches a profile of the revolved feature and a centerline to revolve about. The user then inputs the degrees of revolution.

**Sweep**

The Sweep option protrudes a section along a user-sketched trajectory. The user sketches both the trajectory and the section.

**Blend**

The Blend option joins two or more sketched sections. The trajectory may be straight or revolved.

**Use Quilt**

Quilts are patchworks of surfaces. The Use Quilt option turns a quilt into a solid feature.

**Advanced**

Common modeling options under the Advanced menu include Variable Section Sweep, Swept Blend, and Helical Sweep.

Shown in Figure 2–2 is an illustration of how one section can be used to create an Extrude, Revolve, Sweep, or Blend feature.

**Solid Versus Thin Features**

When creating a Protrusion or Cut, Pro/ENGINEER gives the option of choosing either a solid feature or a thin feature. Solid features are objects that are completely enclosed with material. Thin features are often confused with surfaces. In Pro/ENGINEER, surfaces are quilts with no defined thickness, whereas thin features are actually solids with a user-defined thickness. As shown in Figure 2–3, when the section is extruded as a solid, the section’s feature is completely enclosed with material. When the section is extruded as a thin feature, the walls of the section are protruded with the provided wall thickness only.

Thin features can be used with all forms of the Extrude, Revolve, Sweep, and Blend options under the Protrusion and Cut commands. An example of an extruded thin cut is shown in Figure 2–4. The Thin option may be used with the Protrusion command for the base or secondary feature of a part or with the Cut command for secondary features.

**Extruded Features**

The following section will explore options available within extruded Protrusions and Cuts.

**Extrude Direction**

When sketching on a plane, Pro/ENGINEER, by default, specifies an extrude direction. When sketching on a datum plane, this direction is in the positive direction. When sketching on an existing feature, a Protrusion, as shown in Figure 2–5, will be extruded away from the feature. Since the objective of a Cut is to remove material, a Cut will be extruded toward the feature.
The Extrude option gives the user the option of flipping the direction of extrusion or specifying an extrusion in both directions. The Both Sides selection protrudes a section outward from the sketch plane in both directions. If the extrude depth is input to be 1.00 inch, the total extrusion will be 1 inch, not 1 inch in both directions. A typical Both Sides extrusion will divide the specified depth and extrude equally on both sides of the sketching plane. The 2 Side Blind depth option, though, allows the user to input unequal extrusion distances on both sides of the sketch plane.

**DEPTH OPTIONS**

For Extruded Protrusions and Cuts an important parameter is the distance of extrusion. Pro/ENGINEER provides eight basic ways to specify an extrusion’s depth. Four common depth options are shown in Figure 2–6. The depth for an extrusion is entered for a feature after exiting the sketching environment.

**BLIND**

Blind is the simplest and most basic of the depth options. The Blind option allows a user to input an extrusion distance. It is the most common option for extruded base features.

**2 SIDE BLIND**

A 2 Side Blind is used with a Both Sides direction option only. This depth option will allow the user to enter separate extrude depths for both sides of the sketch plane.

**THRU NEXT**

The Thru Next option extrudes a feature to the next part surface. Part geometry must exist prior to using this option.

**THRU ALL**

Thru All is one of the most common depth options for cut features. It extrudes a feature through the entirety of a part. The design intent of many material removal features (such as a Cut or Hole) is to cut completely through a part. Entering a blind depth that will extrude through the part may not be adequate if the part thickness changes. The Thru All option adjusts for changing part dimensions. This option is available for parts with existing features and is not available for surface features.

**THRU UNTIL**

The Thru Until option extrudes a feature until a user-selected surface. The surface can be any geometry, but cannot be a datum.

**PNT/VTX**

The Pnt/Vtx option extrudes a feature up to a selected datum point or vertex.

**UPTO CURVE**

The UpTo Curve option extrudes a feature up to a selected edge, axis, or datum curve.

**UPTO SURFACE**

The UpTo Surface option extrudes a feature up to a selected surface.

**OPEN AND CLOSED SECTIONS IN EXTRUSIONS**
Extruded sections may be sketched opened or closed. With the obvious exception of a base feature, many sections for an extruded Protrusion or Cut will suffice with an open section. The following are guidelines to follow when considering an open or closed section.

- Sections may not branch, and they can have only one loop. As shown in Figure 2–7, when sketching a section aligned with the edges of an existing feature, it often is not necessary to sketch over the existing geometry. Aligning the required sketch with the existing geometry will usually create a successful section. If Pro/ENGINEER is not sure which side of the section to protrude or cut, it will require the user to select a side (see Figure 2–8).
- Thin feature sections may be open or closed.
- For thin features, sections can be open when not aligned with existing geometry.
- Multiple closed sections can be included in a sketch. As shown in Figure 2–9, when a section is included within another, the inside section creates negative space.

**MATERIAL SIDE**

Two definitions are associated with Material Side. The most common is the Material Removal Side. Within the Extrude option, the Material Removal Side definition is relevant only to the Cut command. The Material Removal Side definition is used to specify what side of a section that material will be removed from. By default, the removal side is toward the inside of a section. The user has the option of flipping the direction. Material Side definitions can be used with Protrusions when a sketch is an open section. Often, Pro/ENGINEER cannot determine which side of the sketch should be extruded. When this situation occurs, the user has to input the material side.

**DATUM PLANES**

Datum planes are used as references to construct features. Datum planes are considered features, but they are not considered model geometry. When a datum plane is created through the use of the Datum Toolbar, the datum plane will show as a feature on the model tree. A datum plane can be created and used as a sketch plane where no suitable one currently exists. As an example, a datum plane can be constructed tangent to a cylinder. This will provide a sketching environment that can be used to construct an extruded feature through the cylinder (Figure 2–10). When a feature is created on a datum plane, the datum plane is considered a parent feature.

A datum plane continues to infinity in all directions, with one side yellow and the opposite side red. Protrusions and orientations occur initially toward the yellow side of the datum plane. By default, datum planes are named in sequential order starting with DTM1. As a note, template files may have datum planes that have been renamed (e.g., Front, Top, and Right) with the Set up >> Name option.

Many times, a datum plane would be useful, but cluttering the model with additional features could be detrimental or confusing. One solution to this problem is to make a datum plane that is used for single feature creation only. To do this, datum planes can be created on-the-fly using the Make Datum option. Datum planes created on-the-fly belong to the feature undergoing creation. These datum planes do not show on the model tree and become invisible after the feature is created.

New to Release 2000i2 of Pro/ENGINEER, a datum plane can be created at almost any point in the modeling process, including during the middle of a sketching environment. Traditionally, datum planes are created exclusively from other features. Starting with 2000i2, this new form of an on-the-fly datum still creates datum planes as features, but allows for flexibility during the creation process.

**CREATING DATUM PLANES**
A datum plane can be created at any point in the modeling process. One of the primary uses of a datum plane is as a sketching surface. Datum planes can be used as a mirror plane within the Copy command, or they can be used as references when sketching a feature. Datum planes are powerful features for aligning and mating parts within Assembly mode. Additionally, datum planes can be combined with other features using the Group option and then patterned.

Creating datum planes is a vital skill needed by all Pro/ENGINEER users. Several constraint options exist for datum plane definition. Some constraint options are stand-alone, while some are not. Paired constraint options are not stand-alone. They require two or more constraint definitions during the datum plane construction process. As an example, the Angle option requires the selection of an existing plane from which to reference the angle, then a Through constraint option to pass the datum plane through an axis or edge. Stand-alone constraint options require one option only.

**STAND-ALONE CONSTRAINT OPTIONS**

The following options only require one constraint to define a datum plane:

**THROUGH >> PLAN**

The Through >> Plane constraint option creates a datum plane that passes through an existing part plane.

**OFFSET PLANE**

The Offset Plane constraint option creates a datum plane that is offset from an existing plane. The user selects the plane from which to offset. Two offset options are available:

- **Point** Select a point to pass the datum plane through. The plane will be created through the point and parallel to the existing part plane.
- **Enter value** Enter an offset distance value. When selecting this option, the user is prompted to enter the offset value. An arrow in the graphics screen shows the default direction of the offset. To offset in the opposite direction, enter a negative value. The plane will be created offset from the reference plane at the value entered.

**OFFSET/COORD SYS**

The Offset Coordinate System constraint option creates a datum plane offset from the coordinate origin and normal to a selected coordinate axis. A coordinate system has to exist prior to the use of this option.

**BLEND SECTION**

The Blend Section constraint option creates a datum plane through a section used to create a feature.

**PAIRED CONSTRAINT OPTIONS**

The following options require two or more constraints to define a datum plane.

**THROUGH >> AXISEDGECURV**

This option is similar to the Through >> Plane option, except this option places a plane through an axis, edge, or curve. An axis, edge, or curve selected with the Through option will not fully constrain a datum plane; hence, a second constraint option is required.

**THROUGH >> POINT/VERTEX**

This option places a datum plane through a point or vertex. Similar to the Through >> AxisEdgeCurv option, this option will not fully define a datum plane and needs an additional constraint option.

**NORMAL >> AXISEDGECURV**
This option places a datum plane perpendicular to an axis, edge, or curve. As with the Through >> AxisEdgeCurv option, an additional constraint is needed.

**Tangent >> Cylinder**

This constraint option places a datum plane tangent to a hole or cylindrical surface. This is an extremely useful option since it allows a feature to be constructed on the surface of a cylinder. This constraint option is often paired with the Normal >> Plane option or the Angle >> Plane option.

**Angle >> Plane**

This option places a datum at an angle to an existing plane. It is often paired with the Through option, or the Tangent >> Cylinder option.

**Modifying Features**

What separates parametric modeling packages, such as Pro/ENGINEER, from boolean-based modeling packages are their feature modification capabilities. Features created within Pro/ENGINEER are composed of parameters. Examples of parameters include parametric dimensions, extrude depth, and material side. Parameters such as these are established during feature construction. These feature parameters, and other feature definitions such as a section’s sketch and sketch plane, can be modified later in the part modeling process.

As shown in Figure 2–11, a variety of feature modification options can be found under the Modify menu.

**Dimension Modification**

Parametric dimensions are used to define a feature. They can be modified at any time. Modifying a dimension value is the most common dimension modification function, but other modification tools exist. The number of decimal places in a dimension can be modified along with the tolerance format. The following are dimension modification techniques that are available within Pro/ENGINEER.

**Modifying a Dimension Value**

The value of a parametric dimension value can be modified. To modify a dimension, select the Value option from the Modify menu, then select the feature associated with the dimension to be modified. Select the dimension to modify and enter a new value. Modifying a dimension value requires the regeneration of the part. Select Regeneration from the Part menu.

**Modifying a Tolerance Mode**

Dimension tolerances can be displayed in a variety of modes. To modify the tolerance mode for individual dimensions, access the Dimension Properties dialog box by selecting Dimension under the Modify menu. As shown in Figure 2–12, an option exists on the Dimension Modification dialog box that allows for the changing of a tolerance mode.

**Modifying Tolerance Values**
The Value option under the Modify menu allows for the modification of tolerance values. As an example, if a dimension is set to Limits as the tolerance mode, either the upper or the lower dimension value can be changed. A problem with this approach is that the nominal value of the limit dimension cannot be modified. Another approach is to modify the nominal value and/or the tolerance values with the Dimension Properties dialog box (Figure 2–12). To access this dialog box, select Dimension from the Modify menu.

**Dimension Decimal Places**

Initially, dimension decimal places are set to two. This value can be changed permanently with the configuration file option `default_dec_places`. To change decimal places for individual dimensions, access the Dimension Modification dialog box by selecting Dimension from the Modify menu. As shown in Figure 2–12, an option exists for changing the decimal places of a dimension.

**Cosmetic Dimension Modification**

As shown in Figure 2–11, the DimCosmetics option under the Modify menu provides tools for modifying dimensions. A dimension’s cosmetics are the way that the dimension appears on the object. Modifying a dimension’s cosmetics does not modify the value of the dimension. The following are techniques that the DimCosmetics option provides for modifying dimensions.

**Tolerance Format**

A tolerance’s format mode can be changed using the Format option found under the DimCosmetics menu. Select the dimensions for format change, then select the desired format to use.

**Number of Digits of a Dimension**

The number of decimal places displayed by a dimension can be changed using the Num Digits option found under the DimCosmetics menu. To change the number of decimal places for dimensions, select Num Digits, then enter the number of significant digits. Follow this by selecting the dimensions to change.

**Adding Text Around a Dimension**

Text can be added around a dimension value. A dimension’s value is shown in a dimension note with the symbol @D. This symbol must remain in the dimension note. To add text around a dimension value, select Text from the DimCosmetics option then select the dimension to modify. Follow this by entering lines of text.

**Changing Dimension Symbols**

Dimensions within Pro/ENGINEER can be displayed in two ways. The first way is by showing the actual dimension or tolerance value. The second way is by showing the dimension symbol. As shown in Figure 2–13, when a dimension is created, it is provided with a dimension symbol. For the first dimension created on a part, the default symbol is d0. This number increases in sequential order for every new dimension. To change a dimension’s symbol to make it more descriptive, utilize the Symbol option from the DimCosmetics menu.

**Redefining Features**

Features are composed of parameters. Parameters can be modified and changed through the Redefine command or through the Modify menu. Many different varieties of parameters exist. The following is a partial list.

- A feature’s section.
- The sketch plane for a feature.
• The depth option for a feature.
• The material removal side.
• The extrude direction of a feature.
• The trajectory of a swept feature.
• The value of a dimension.

Figure 2–14 shows an example of a Feature Definition dialog box for an extruded Protrusion. This dialog box is accessible at the end of the construction of a part feature or it can be accessed through the Feature menu’s Redefine command. Perform the following steps to redefine a feature parameter.

**STEP 1:** Select FEATURE >> REDEFINE.

**STEP 2:** On the work screen or on the Model Tree, select the feature to be redefined.

After selecting the feature to redefine, the Feature Definition dialog box associated with the feature will appear (Figure 2–14). This dialog box is composed of two columns. The first column displays the definition’s name, while the second column displays the definition’s current value.

**STEP 3:** On the Feature Definition dialog box, select the element to redefine.

Select the definition’s name displayed in the first column of the dialog box.

**STEP 4:** Select DEFINE on the dialog box.

After selecting Define, Pro/ENGINEER will step you through the remodeling of the parameter.

**STEP 5:** Redefine the parameter according to Pro/ENGINEER modeling procedures.

**STEP 6:** View the feature’s new parameters by selecting PREVIEW on the dialog box.

**STEP 7:** Select OKAY on the dialog box.