

Chapter 8

Advanced Modeling Tools-I

Learning Objectives

After completing this chapter, you will be able to:

- *Create sweep features.*
- *Create features using sweep cut.*
- *Create parallel, rotational, and general blends.*
- *Use blend vertex in blend features.*
- *Create shell features.*
- *Create datum curves.*
- *Create draft features.*

OTHER PROTRUSION OPTIONS

The **Extrude** and **Revolve** buttons available in the **Base Features** toolbar were discussed in Chapter 4, and the **Sweep** and **Blend** options are discussed in this chapter. As mentioned earlier, Protrusion and Cut are the two basic options available in Pro/ENGINEER that are used to create a feature.



Note

*All options that are available for creating a cut are similar to those that are available for creating a protrusion. Remember that a cut is performed on an existing feature and therefore, the **Cut** option is available only when at least a base feature exists in the drawing area.*

In this chapter, you will also learn about the tools of solid modeling that make the creation of a complex model easy. In the next section, you will learn about sweep features.

SWEEP FEATURES

The **Sweep** option extrudes a section along a defined trajectory. The order of operation is to first create a trajectory and then a section. Trajectory is a path along which a section is swept. The trajectory for a sweep feature can be either sketched or selected. The **Sweep** option of protrusion is similar to the **Extrude** option. The only difference being that in the case of the **Extrude** option, the feature is extruded in a direction normal to the sketching plane, but in the case of the **Sweep** option, the section is swept along the sketched or selected trajectory. The trajectory can be open or closed. Normal sketching tools are used for sketching the trajectory. The cross section of the swept feature remains constant throughout the sweep.



Note

Some important points to remember while drawing a trajectory and a section for a sweep feature are discussed later in this chapter.

Creating Sweep Protrusions

The **Sweep** option can be used for adding material as well as for removing material, that is, for protrusion as well as for cut features. You can choose the **Sweep** option from the menu bar. In the **Insert > Sweep > Protrusion** option that is discussed here, material defined by the section is added in the specified path. The **SWEEP TRAJ** (Sweep Trajectory) menu appears, as shown in Figure 8-1. The options in this menu are discussed next.



Figure 8-1 The **SWEEP TRAJ** menu

Sketch Traj Option

The **Sketch Traj** (Sketch Trajectory) option is used when you want to sketch the trajectory for the sweep feature. This is the most commonly used option for defining the trajectory. As mentioned earlier, the trajectory can be open or closed. There are some limitations for using closed or open trajectory with closed or open section. These limitations are discussed in the next section. When you choose the **Sketch Traj** option, you are prompted to select a sketching plane. The sketching plane you select will be parallel to the screen when you sketch the trajectory. Figure 8-2 shows how the section is sweep along the sketched trajectory, and Figure 8-3 shows the shaded image of the sweep feature.

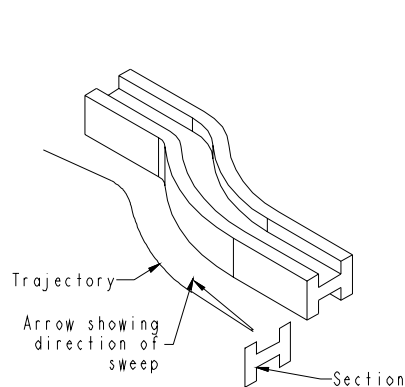


Figure 8-2 Sweep along the sketched trajectory

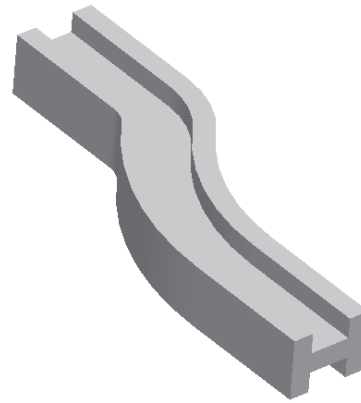


Figure 8-3 Shaded image of the sweep feature

The following points specify the combinations of trajectories and sections that are possible/not possible to create.

1. Open section and open trajectory are not possible.
2. Closed section and open trajectory are possible.
3. If the sketched trajectory is a closed loop then after you exit the sketcher environment, the **ATTRIBUTES** menu is displayed, as shown in Figure 8-4. There are two options available in this menu: **Add Inn Fcs** (Add inner faces) and **No Inn Fcs** (No inner faces). The **No Inn Fcs** option is chosen by default.



Figure 8-4 The **ATTRIBUTES** menu

By using the **Add Inn Fcs** option, only open sections are possible, as shown in Figure 8-5. The shaded image of the corresponding sweep feature is shown in Figure 8-6. These two figures explain the **Add Inn Fcs** option.

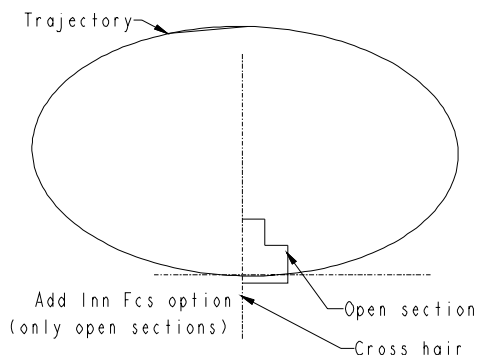


Figure 8-5 The open section on choosing the **Add Inn Fcs** option

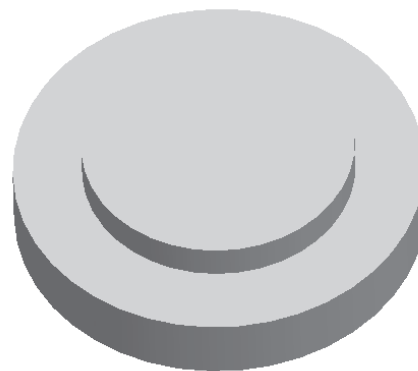


Figure 8-6 Shaded image of the sweep feature

By using the **No Inn Fcs** option, only closed sections are possible, as shown in Figure 8-7. The shaded image of the corresponding sweep feature is shown in Figure 8-8.

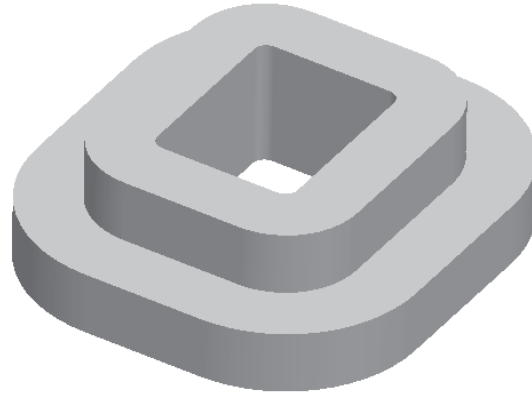
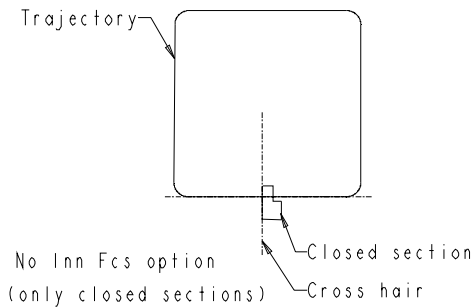


Figure 8-7 The closed section on choosing the **No Inn Fcs** option

Figure 8-8 Shaded image of the sweep feature

Select Traj Option

The **Select Traj** (Select Trajectory) option allows you to select a trajectory in the drawing area. The trajectory to be selected can be an existing edge or a datum curve. Creation of datum curves will be discussed later in the chapter. When you choose this option from the **SWEEP TRAJ** menu, the **CHAIN** menu is displayed. The **CHAIN** menu is also discussed later in this chapter.

Figures 8-9 and 8-10 show two examples of selecting the edges of the base feature and then using these as a trajectory to sweep. The corresponding sweep features are also shown.

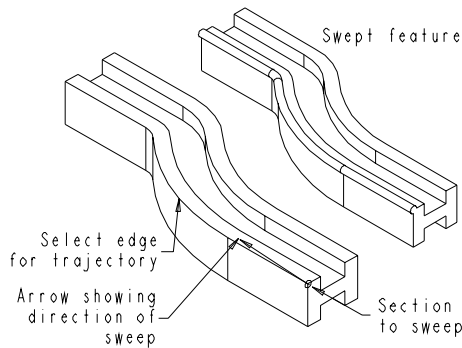


Figure 8-9 Sweep along the selected trajectory

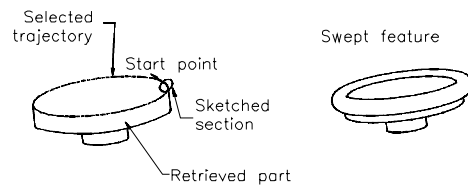


Figure 8-10 Sweep along the selected trajectory

Sketching a Trajectory Aligned to an Existing Geometry

When one end of the sketched trajectory is aligned to the adjacent geometry of the existing feature, Pro/ENGINEER provides two options. The first option is to merge the ends of the sweep feature with the adjacent geometry and the second option is to leave the ends of the

sweep feature free. These options are available in the **ATTRIBUTES** menu that is displayed after you complete the sketch of the trajectory and choose the **Done** button. The **ATTRIBUTES** menu is shown in Figure 8-11.

The **ATTRIBUTES** menu is displayed only when the trajectory is aligned to an edge or surface of the feature that already exists in the drawing area. This means that the **ATTRIBUTES** menu does not appear if the sweep feature you are drawing is the base feature of a model or, in other words, if there is no adjacent geometry to which the trajectory can be merged. The options available in the **ATTRIBUTES** menu are discussed next.



Figure 8-11 The ATTRIBUTES menu

Merge Ends

The **Merge Ends** option merges the end of a sweep feature to the surface to which the end of the trajectory is aligned. For this option the trajectory should be aligned to the adjacent geometry.

Free Ends

The **Free Ends** option leaves the sweep feature partially attached to the adjacent feature even if the end of the trajectory is aligned to the adjacent geometry.

Figure 8-12 shows the use of **Merge Ends** and **Free Ends** options. In the figure shown below, the trajectory is aligned with the adjacent geometry in both the cases. Figure 8-13 shows the corresponding shaded image.

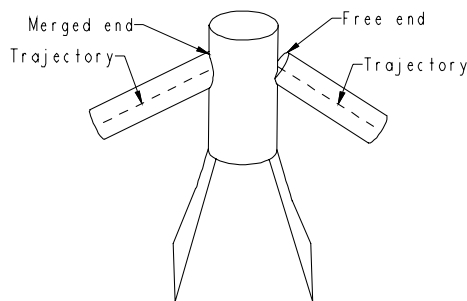


Figure 8-12 Model with merge and free ends options

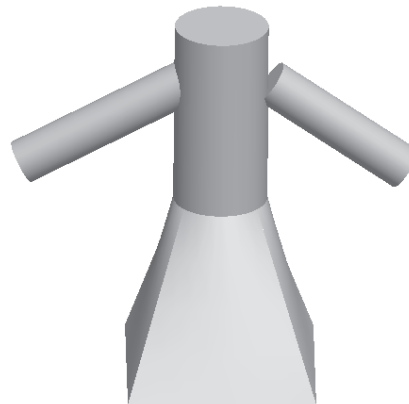


Figure 8-13 Shaded image

Creating Sweep Feature by Selecting a Trajectory

When you choose the **Select Traj** option from the **SWEEP TRAJ** menu, the **CHAIN** menu is displayed, as shown in Figure 8-14. You can use the **CHAIN** menu only if you have created a feature that will be used to select the trajectory. The options in this menu are used to select a trajectory. These options are discussed next.



Tip: The following points should be remembered while creating a sweep feature:

1. Similar to other sketched features, the trajectory of the sweep feature is also sketched after selecting a sketching plane.
2. The section for the sweep trajectory is sketched using the normal sketcher tools when the sketch trajectory option is selected.
3. At bends in a trajectory, the radius of the bend should be proportionate to the cross section to be swept to avoid overlapping. If the section size is large and the radius of the curve or bend is small, overlapping takes place and the sweep feature will not be created. Therefore, make sure that the ratio of the size of the section to the size of the trajectory is appropriate.

One By One

The **One By One** option of the **CHAIN** menu is selected by default. Using this option you can select an edge or curve individually, one by one. You have to hold the CTRL key while selecting the edges. When you select an edge, it is highlighted in red. Before selecting the edge, make sure that the **Select** option in the **CHAIN** menu is highlighted. The edge once selected and confirmed by choosing the **OK** button from the **Select** message box or by using the middle mouse button can also be unselected by choosing the **Unselect** option from the **CHAIN** menu.

Tangnt Chain

Using this option, you can select an edge or edges tangent to the selected edge. When you select an edge, all edges tangent to the selected edge are highlighted. If the selected edge is not tangent to any other edge then the function of this option is the same as that of the **One By One** option. The difference being that you can select only one edge in the case of the **Tangnt Chain** option.

Curve Chain

You can select a chain of curves by using the **Curve Chain** option.

Bndry Chain

The **Bndry Chain** (Boundary Chain) option is used only for surface features. You can define a chain by selecting a quilt and using its one-sided edges. If the quilt has more than one loop, select a specific loop to define the chain.

When you select the edge of the quilt, it is highlighted in blue and the **CHOOSE** menu is displayed with the options, as shown in Figure 8-15.



Figure 8-14 The **CHAIN** menu



Figure 8-15 The **CHOOSE** menu

Surf Chain

Using the **Surf Chain** option, you can define a chain by selecting a surface and using its edges. If the surface has more than one loop, then you are prompted to specify a loop to define the chain. When a surface is selected, the **CHAIN OPT** menu is displayed, as shown in Figure 8-16. Choose either **Select All** or **From-To** from the **CHAIN OPT** menu.



Figure 8-16 CHAIN OPT menu

Intent Chain

The **Intent Chain** option is used to select multiple edges. When a section is extruded, the edges formed by the extrusion consists of intent chains. The intent chains can either be the edges of the section or the edges of the extruded surface. The edges selected should form a closed loop.

Creating Thin Sweep Protrusion

The **Sweep > Thin Protrusion** option creates a thin sweep feature with a specified thickness. This option is similar to the **Extrude > Thin** option that was discussed in Chapter 4. In case of thin features, a certain thickness has to be specified. The thickness is specified on one side of the section or symmetrically to both the sides of the section. The resultant sweep is similar to the solid sweep created with a section comprising of two closed loops at some offset distance. Figure 8-17 shows the sections that can be used to create the model shown in Figure 8-18.

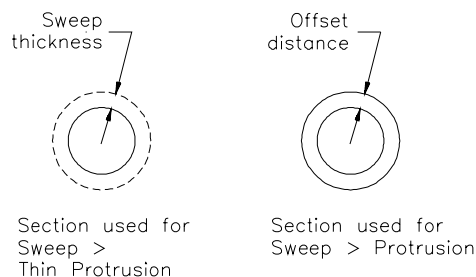


Figure 8-17 Two possible sections to create the same model

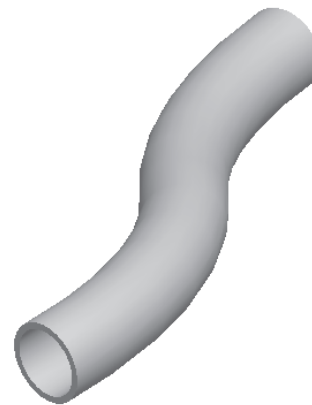


Figure 8-18 Same model created using different sections

Creating a Sweep Cut

To create a **Sweep Cut** feature, the procedure to be followed is the same as that in **Sweep Protrusion**. The only difference is that in case of cut features, the material is removed from an existing feature. The **Cut** option can be invoked by choosing **Insert > Sweep > Cut** from the menu bar. Cut can be a solid swept cut or thin swept cut. Figure 8-19 shows trajectories for the **Sweep Cut** feature. Figure 8-20 shows the shaded model with open and closed trajectory sweep cuts.

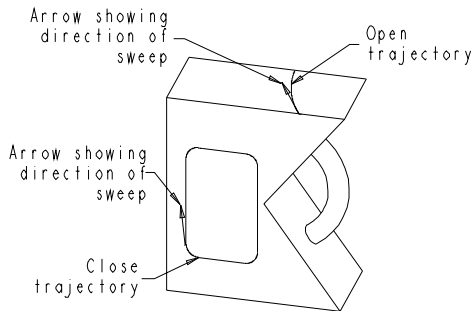


Figure 8-19 Trajectories for the Sweep Cut feature



Figure 8-20 Shaded model with open and close trajectory sweep cuts

BLEND FEATURES

Blend features are composed of two or more sections that are joined through transitional faces at their edges so as to form a continuous feature. The number of entities in each section that creates the blend feature should be the same. For example, you cannot blend a circle with a rectangle. This is because a rectangle is composed of four entities and a circle of one. It can be achieved only if the circle is divided into four entities.

In Pro/ENGINEER, the **Blend** feature is of two types, **Protrusion** and **Cut**. The **Blend** option is used where the feature to be created has varying cross sections. To invoke this option, choose **Insert > Blend > Protrusion** or **Insert > Blend > Cut** from the menu bar.

When you choose the **Blend > Protrusion** option, the **BLEND OPTS** submenu is displayed, as shown in Figure 8-21. The options in this menu are discussed next.

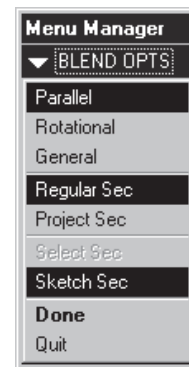


Figure 8-21 The **BLEND OPTS** submenu

Parallel Blend

Parallel blends have sections that are drawn parallel to each other with a specified distance between them.

After choosing **Parallel > Regular Sec > Sketch Sec > Done** from the **BLEND OPTS** submenu, the **ATTRIBUTES** menu is displayed, as shown in Figure 8-22. The options in this menu are discussed next.



Figure 8-22 The **ATTRIBUTES** menu

Straight Option

The **Straight** option is used to connect the vertices of all sections in a blend feature with straight lines.

Smooth Option

The **Smooth** option is used to connect the vertices of all sections in a blend feature with curves.

Figure 8-23 shows three sections that are used to create the blend feature. Figures 8-24 and 8-25 show the parallel blend features with straight edges and smooth edges respectively, created using the sections shown in Figure 8-23.

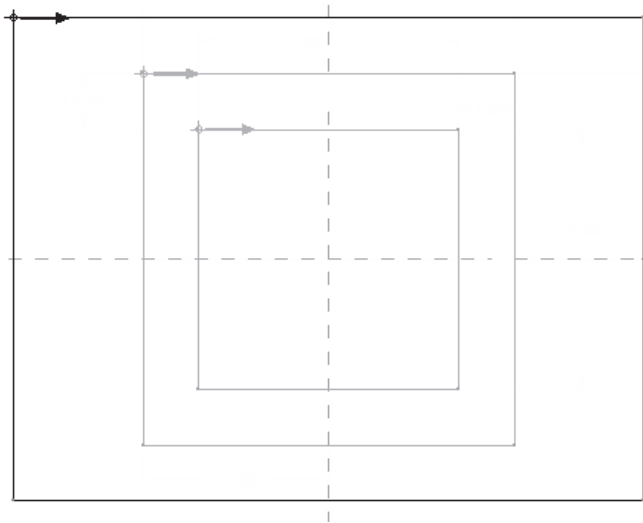


Figure 8-23 Three parallel sections

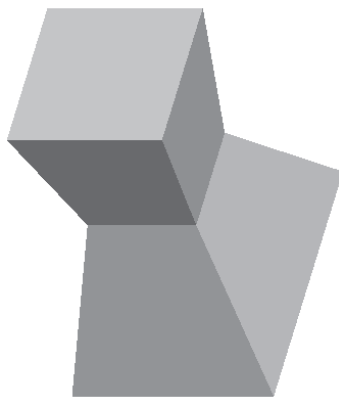


Figure 8-24 Parallel blend with straight edges



Figure 8-25 Parallel blend with smooth edges



Note

While drawing a section, the start point of all sections should be in the same direction in order to avoid twisted blend features.



Tip: The following points should be remembered while creating a **Parallel** blend feature:

1. After completing the first section, choose **Sketch > Feature Tools > Toggle Section** to proceed for drawing the second section. You can also hold down the right mouse button and choose the **Toggle Section** option from the shortcut menu that appears. If you choose the **Done** button before drawing the second section, the system prompts you to use the **Toggle Section** option to continue with the second section.
2. Active section appears in cyan color and the other sections appear in gray.
3. All sections in a blend feature must have the same number of entities. However, you can blend a point with any section irrespective of the number of entities.
4. System prompts for depth between subsequent sections after completion of all sections in the blend.
5. By default, the start point of any entity that is drawn to define a section is considered as the **Start Point** of the section. To change the **Start Point** of a section, select the point to be defined as the **Start Point** and hold down the right mouse button to display the shortcut menu. Choose the **Start Point** option from this shortcut menu to change the start point.
6. After defining the sections in a blend feature and before choosing the **Done** button, the sections can be modified by using the **Toggle Section** option.

Rotational Blend

The rotational blends have sections that are rotated about the Y-axis up to a maximum of 120-degree and the distance between two sections is measured from the coordinate system. Between each section, an angle called the **rotational blend angle** has to be defined. In this type of blend, each section has its own user-defined coordinate system. If the rotational blend angle entered between the two sections is equal to 0-degree, then the **Rotational** blend option functions the same way as the **Parallel** blend option.

Note that all nonparallel blends can be open or close. Therefore, after choosing **Rotational > Regular Sec > Sketch Sec > Done** from the **BLEND OPTS** menu, the **ATTRIBUTES** menu is displayed, as shown in Figure 8-26. The options in this menu are discussed next.

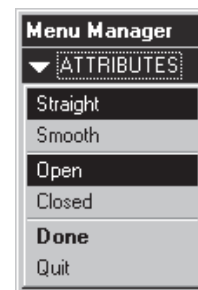


Figure 8-26 The **ATTRIBUTES** menu

Open Option

The **Open** option is used when the blend feature to be created has to be kept open.



Tip: It is recommended that the closed blend features should have at least three sections.

Closed Option

The **Closed** option is used to create a closed blend feature. In this type of blend feature, Pro/ENGINEER closes the feature by automatically blending the last section with the first section. Figure 8-27 shows the sections used to create a rotational smooth blend feature. The three default datum planes can also be seen. From Figure 8-27, it is evident that two sections are used to create the blend feature and that these sections are at an angle of 45-degree. It is also evident from the figure that the second section is dimensioned from the coordinate system that was defined in the first section. Figure 8-28 shows the shaded model of the same blend feature.

The following steps explain briefly the procedure of creating the blend feature shown in Figure 8-28.

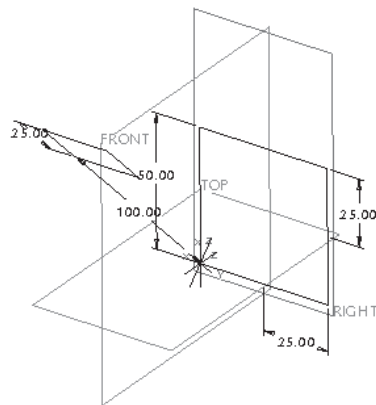


Figure 8-27 The two sections with dimensions and the default datum planes used to create the blend feature shown in the adjacent figure

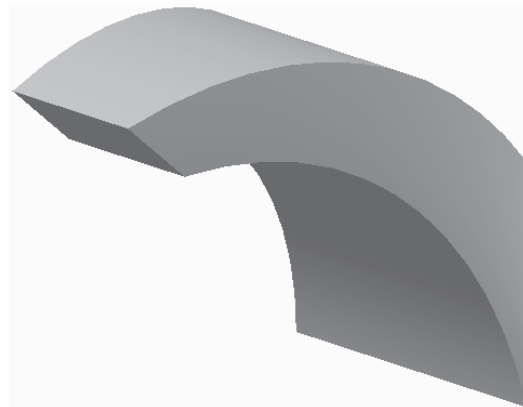


Figure 8-28 Shaded model of rotational open blend feature

1. Invoke the **Blend** option and from the **BLEND OPTS** menu, choose **Rotational > Regular Sec > Sketch Sec > Done**. The **ATTRIBUTES** menu is displayed.
2. Choose **Smooth > Open > Done** from the **ATTRIBUTES** menu. You are prompted to select a plane to sketch.
3. Select the **RIGHT** datum plane as the sketch plane and orient the **FRONT** datum plane to be on top.
4. After selecting the sketch plane and its orientation, draw the first section that is a square of side 50. The start point on this square should be at the lower right corner.
5. Place the coordinate system at the lower left corner of the square.
6. Choose the **Done** button to proceed for drawing the next section. Accept the default Y-axis rotation angle of 45-degree for section 2.
7. Draw the second section that is a square of size 25.

8. Place the coordinate system for this section at a distance of 100 toward the left of the lower left corner of the square. Remember that the start point of this section should also lie at the lower right corner of the rectangle.
9. Exit the sketcher environment. The **Message Input Window** appears and you are prompted that do you want to continue to draw the next section.
10. Choose the **No** button. Now, you can see the preview of the feature that you have created and then exit the feature creation tool.

General Blend

Using the **General** blend, sections are translated and rotated about the x, y, and z axes. The sections are aligned using the user-defined coordinate system. The coordinate system has to be manually placed in every section sketch that constitutes the blend feature.

USING BLEND VERTEX

As mentioned earlier, each section of the blend feature must have an equal number of entities. However, you can use the **Blend Vertex** option if the number of entities in all sections are not equal. For example, to create a blending between a rectangle and a triangle, add blend vertex on a point other than the start point of the triangle. The two vertices of the triangle will be blended with the two vertices of the rectangle and the blend vertex in the triangle will be blended with the remaining two vertices in the rectangle, as shown in Figure 8-29.

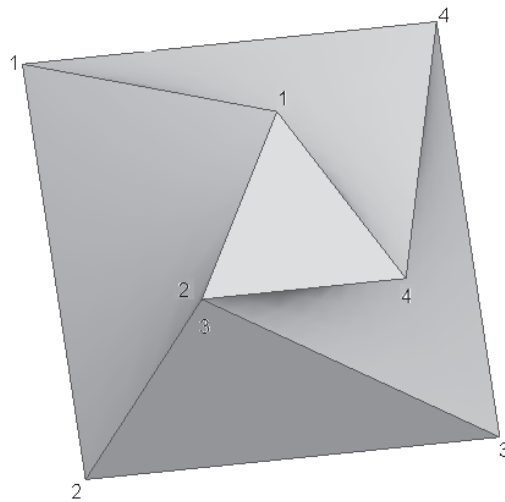


Figure 8-29 Blending a square with a triangle using the blend vertex

To add a blend vertex in a sketch, select the point where you want to place the blend vertex. The selected point is highlighted in red. Choose **Sketch > Feature Tools > Blend Vertex**. The blend vertex is placed at the selected point.



Note

The **Blend Vertex** option can be used only either in the first or last section of a blend feature.

SHELL OPTION



The **Shell** option scoops out the material from the model and at the same time removes the selected faces, leaving behind a thin model with some specified wall thickness. The **Shell** option can be invoked by choosing the **Shell** button from the **Engineering Features** toolbar. You can also choose this option from the menu bar by choosing **Insert > Shell**. The **Shell** dashboard is displayed, as shown in Figure 8-30.



Figure 8-30 The *Shell* dashboard

Options Tab

When you choose the **Options** tab from the **Shell** dashboard, a slide-down panel will be displayed, as shown in Figure 8-31.

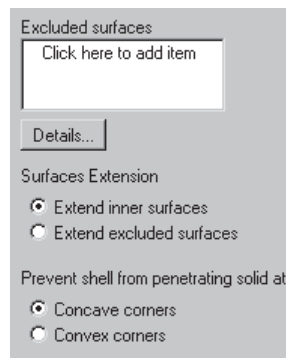


Figure 8-31 The *Options* tab slide-down panel

The **Excluded surfaces** collector lists the surfaces excluded from being removed. If you do not select any surface from being excluded, the entire part will be shelled.

The following steps explain briefly the procedure of creating a Shell feature by using the **Excluded surfaces** collector.

1. Invoke the **Extrude** dashboard and then create the extrude feature, as shown in Figure 8-32.
2. Once the **Extrude** feature is created, invoke the **Shell** dashboard by choosing the **Shell** button from the **Engineering Features** toolbar.
3. Select the top face of the model to remove it.
4. Choose the **Options** tab; the slide-down panel is displayed. Activate the **Excluded surfaces** collector by clicking on it.
5. Select the surfaces to be excluded by pressing the CTRL key, as shown in Figure 8-32.
6. Choose the **Build feature** button from the **Shell** dashboard to exit it.

The shell is created by excluding the surfaces that you have specified, as shown in Figure 8-33.

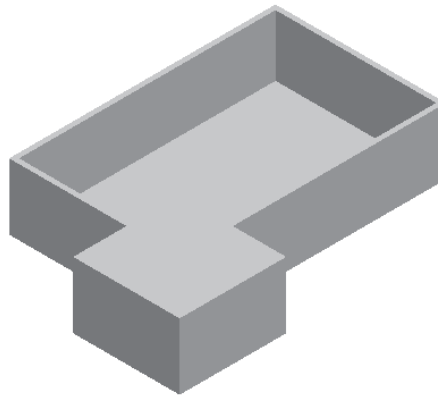
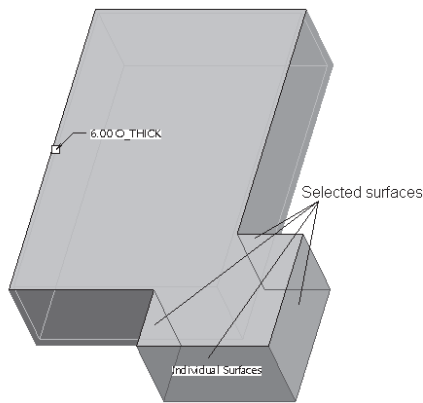


Figure 8-32 Creating shell by excluding surfaces **Figure 8-33** Shell created after excluding surfaces



Note

The **Shell** option is used on existing models and hence, this option is available only when a model exists in the drawing area.

Using the **Shell** dashboard, you can create two types of shell:

1. Constant thickness shell
2. Variable thickness shell

Creating Constant Thickness Shell

The constant thickness shell is a shell that has a uniform thickness on all four faces of the model. The following steps explain the procedure to create a constant thickness shell.

1. Invoke the **Shell** dashboard.
2. Select the top face of the model to remove it, as shown in Figure 8-34. The selected face will be removed from the model, leaving the specified thickness from the boundary of the selected face.
3. Enter the thickness value of the shell in the dimension edit box present on the dashboard.
4. Choose the **Feature Preview** button from the dashboard to preview the shell feature.
5. Choose the **Build feature** button from the **Shell** dashboard to exit it.

The shell is created on the selected face, as shown in Figure 8-35. The thickness of the shell is uniform on all faces of the model.



Note

When you invoke the **Shell** dashboard, the **Surface** filter is selected by default in the **Status Bar**.

Creating Variable Thickness Shell

The variable thickness shell is a shell that has different thickness values assigned to adjacent faces. The following steps explain the procedure to create a variable thickness shell.

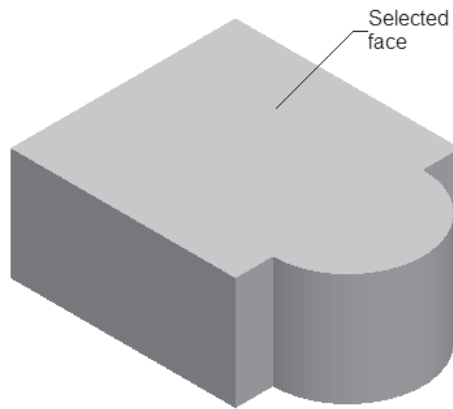


Figure 8-34 Face selected to shell

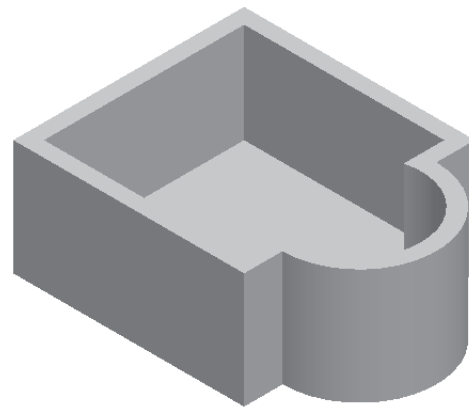


Figure 8-35 Shell created on the selected face

1. Invoke the **Shell** dashboard.
2. Select the top face of the model to remove it, as shown in Figure 8-36.
3. Choose the **References** tab to invoke the slide-down panel. In the slide-down panel, there are two collectors, **Removed surfaces** collector and **Non default thickness** collector. The **Removed surfaces** collector shows the surface id of the face that you have selected to remove. The **Non default thickness** collector shows the surfaces that you will select for creating the variable thickness shell.
4. Click in the **Non default thickness** collector and now select the adjacent faces that are shown in Figure 8-36. To select the second and successive faces, you need to use CTRL+left mouse button.
Now you need to specify the different thickness values with reference to the selected face.
5. Enter the thickness values in the dimension edit boxes that are present on the right side of the surfaces in the **Non default thickness** collector.

The shell is created on the selected faces, with the thickness values assigned to them, see Figure 8-37.

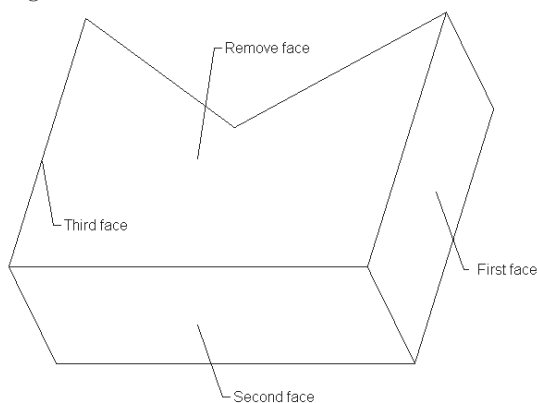


Figure 8-36 Faces of the model selected for variable thickness and the top face to remove

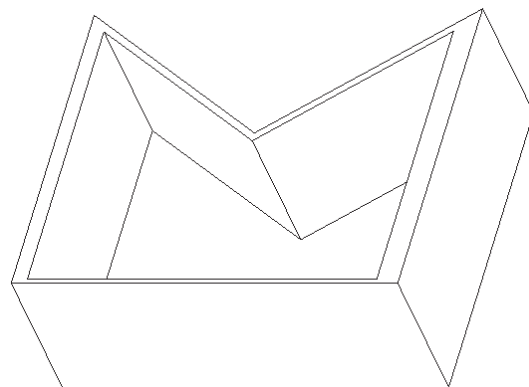


Figure 8-37 Variable thickness shell

**Note**

The thickness value entered can be positive or negative. If the value entered is positive, the material is removed, leaving the shell thickness inside the boundary of the selected face. But when the value entered is negative, the shell thickness is added outside the boundary of the selected face.

Figure 8-38 shows the model whose top surface is selected to be removed in order to create a shell. Figure 8-39 shows the model after shelling. Notice that the shell thickness is left on the selected face and the remaining material is removed.

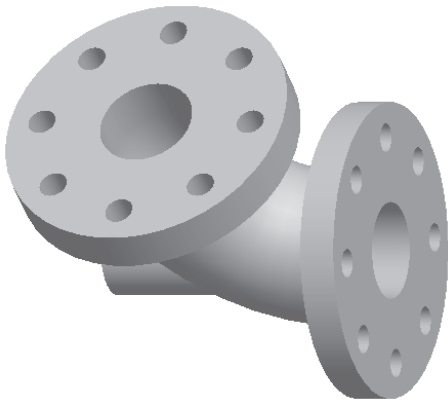


Figure 8-38 Solid model

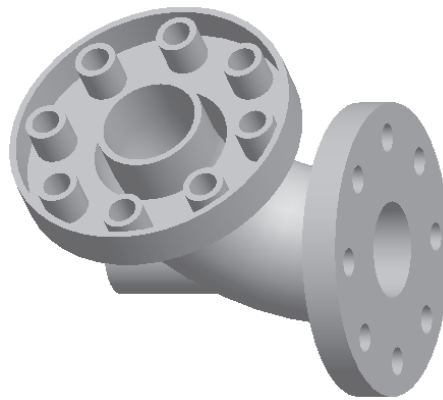


Figure 8-39 Model after shelling the top face

DATUM CURVES

Datum curves are useful in creation of advanced solid and surface features such as the sweep trajectories to create a sweep feature. A datum curve is considered as a feature and is displayed in the **Model Tree**. There are various tools and options to create a datum curve. You can choose the **Curve** button from the **Datum** toolbar to create a datum curve. Alternatively, you can choose the **Intersect**, **Project**, and **Wrap** options from the **Edit** menu to create a datum curve.

Creating Datum Curve Using the Curve Button



The **Curve** button is used to create datum curves. When you choose this button from the **Right Toolchest**, the **CRV OPTIONS** menu is displayed, as shown in Figure 8-40. The options in this menu are discussed next.

Thru Points

The **Thru Points** option creates a datum curve by selecting the existing datum points or vertices. The resulting datum curve may be a spline curve or can have a user-defined radii.

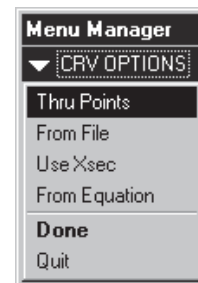


Figure 8-40 The **CRV OPTIONS** menu

When you choose **Done** from this menu, the **CONNECT TYPE** menu is displayed, as shown in Figure 8-41. The options in this menu are discussed next.

Spline

The **Spline** option creates a datum curve in the form of a spline and passing through the selected datum points or vertices.

Single Rad

The **Single Rad** option creates a datum curve that has a constant radius at the bends. This option is called single radius because the system prompts you to specify a radius value at the bend. The bend is the location on the datum curve that lies between two datum points or vertices.

Multiple Rad

As the name suggests, the **Multiple Rad** option allows you to specify radius at every bend that occurs on a datum curve. Using this option you can either specify the same radius that was specified at the previous bend or a new radius value.

Single Point

The **Single Point** option is used to select the datum points individually. If the datum points are created such that they are a single feature, then, all of them may be selected using this option.

Whole Array

The **Whole Array** option is used to select all datum points that act as a single feature.

From File

The **From File** option is used to import a datum curve from IGES, VDA, *.ibl file formats. You can save a model that is in any of the above-mentioned formats and export the geometry in the form of a curve. When you open the exported file to create a datum curve, the geometry of the model is converted to a datum curve.

Use Xsec

The **Use Xsec** option is used to create a datum curve that has the geometry of an existing cross section. The creation of sections is discussed in Chapter 7.

From Equation

The **From Equation** option is used to create datum curves by defining equations using the coordinate systems. When you choose this option, you are prompted to select a coordinate system. Select the default coordinate system from the drawing area. The **SET CSYS TYP** menu is displayed, as shown in Figure 8-42.

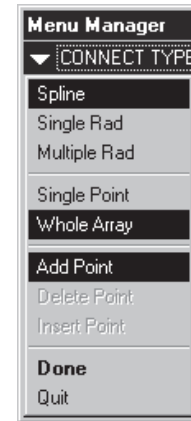


Figure 8-41 The **CONNECT TYPE** menu

You can choose the type of coordinate system from this menu. Choose the coordinate system depending on the equation you want to use to create a datum curve. If you choose the **Cartesian** option from the menu, the **rel.ptd - Notepad** window is displayed, as shown in Figure 8-43.



Figure 8-42 The **SET CSYS TYP** menu

Using this notepad, you can define the equations. The notepad shows the instructions that should be followed while writing an equation in the notepad. These instructions vary and depend on the type of coordinate system you have selected. After you have entered the equations in the notepad, save the file and then exit the notepad.

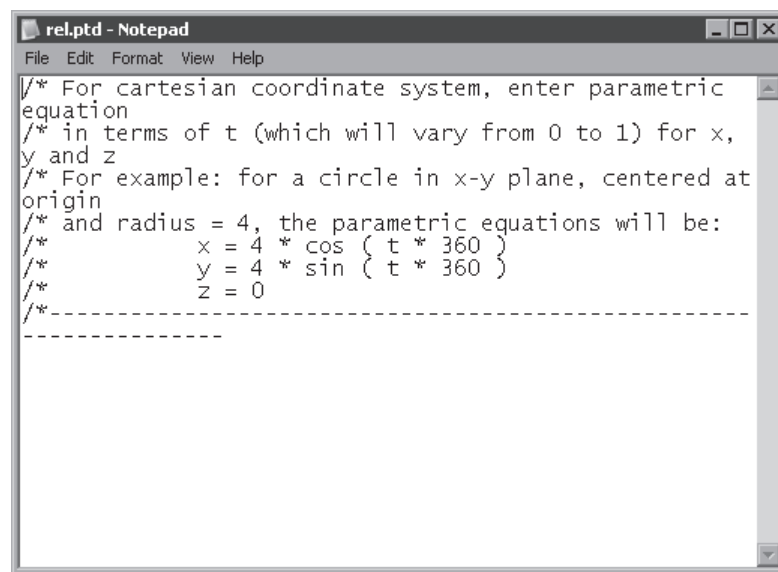


Figure 8-43 *rel.ptd-Notepad* window

Creating a Datum Curve in Spiral Shape

The following steps explain the procedure to create a spiral-shaped datum curve.

1. Choose the **Cylindrical** option from the **SET CSYS TYP** menu. The notepad is displayed. In the notepad the parametric equations to create a circle are given as follows:

$$r = 4$$

$$\text{theta} = t * 360$$

$$z = 0$$

In the above equations, the variable t varies from 0 to 1, r is the radius of the circle, and z is the third equation that is set equal to 0. Now, to understand the given equation of the circle, notice that the radius of the circle is given. The only value that is unknown is theta. The value of theta depends on the variable t . Therefore,

when $t = 0$, $\text{theta} = 0$

when $t = 1/2$, $\text{theta} = 180$ (semicircle)

and when $t = 1$, $\text{theta} = 360$ (circle)

2. Enter the following parametric equation of the spiral curve below the dashed line in the notepad:
 $IR = 8$
 $OR = 80$
 $URNS = 10$
 $r = IR + t * (OR - IR)$
 $theta = t * 360 * URNS$
 $z = 0$
 In the above equations, the value of r is selected to vary because in the spiral curve, the value of r always increases from center (at center of spiral, $r = 0$). The internal radius of the spiral $IR = 8$, outer radius $OR = 80$, and number of turns $URNS = 10$ are given.
3. Choose **File > Save** from the notepad and then exit the notepad.
4. Choose the **Preview** button from the **CURVE: From Equation** dialog box to preview the datum curve.
5. Choose **OK** from the dialog box to exit. The spiral-shaped datum curve appears in the drawing area, as shown in Figure 8-44.

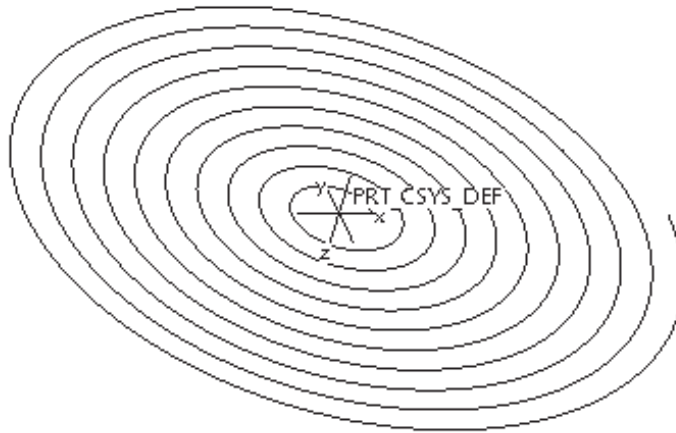


Figure 8-44 Spiral-shaped datum curve

Creating Datum Curve by Sketching

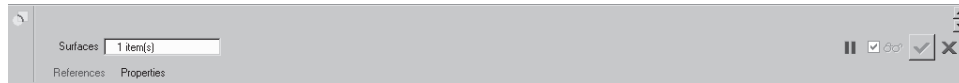


The **Sketch** button is used to sketch a datum curve using the sketcher tools. This is the most commonly used option to create a datum curve. The sketch can be open or closed and is drawn in the sketcher environment.

Intersect Option

The **Intersect** option can be invoked by choosing **Edit > Intersect** from the menu bar. This option is object action tool; this means that you need to select a datum plane and then invoke this option. The **Intersect** option creates a datum curve at the intersection of a face of the model and a datum plane, intersection of a face of the model and a quilt surface, intersection of a quilt surface and a datum plane. Note that you cannot create a datum curve at the intersection of two datum planes, two quilts, or two model faces using this option.

When you choose this option, the **Intersect** dashboard appears, as shown in Figure 8-45. You may need to choose the **Intersect** option twice to invoke the **Intersect** dashboard.



*Figure 8-45 The **Intersect** dashboard*

To create the ring on the circumference of the cylindrical feature, as shown in Figure 8-46, you need to create a datum curve that is on the circumferential surface of the cylindrical feature. To create the datum curve that lies on the outer surface, follow these steps:



Figure 8-46 Ring on the circumference of the cylindrical feature

1. Select the datum plane that is intersecting with the circumferential surface of the cylinder.
2. Choose **Edit > Intersect** from the menu bar. The **Intersect** dashboard is displayed.
3. Choose the **References** tab to open the slide-down panel. In the **Surfaces** collector, the selected datum plane is displayed.
4. Use CTRL+left mouse button and select the top and the bottom circumferential surface of the cylindrical feature, as shown in Figure 8-47.
5. Choose the **Build feature** button from the dashboard to exit the feature creation tool. The datum curve is created on the circumferential surface of the cylinder, as shown in Figure 8-48.

Project Option

The **Project** option projects a selected or sketched entity on one or more planar or non-planar surfaces or datum planes. The projected datum curve forms a true projection of the selected or sketched entity on the specified surfaces. The dimensions of the original entity may distort while projecting.

Choose **Edit > Project** from the menu bar; the **Project** dashboard is displayed, as shown in Figure 8-49. The tools and options available in this dashboard are discussed next.

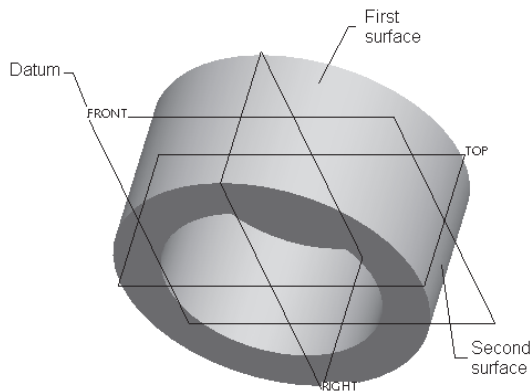


Figure 8-47 Selections made on the model

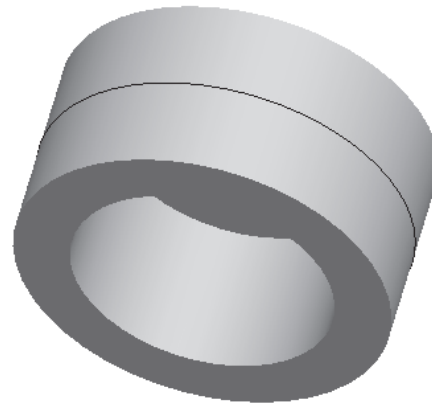


Figure 8-48 Datum curve created on the circumferential surface of the cylindrical feature

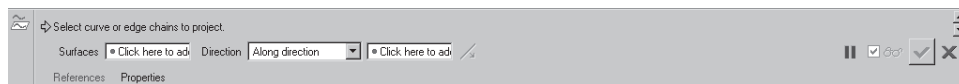


Figure 8-49 The **Project** dashboard

Surfaces Collector

The **Surfaces** collector is used to select the surface on which you need to project the sketched or selected datum curve.

Direction Drop-down List

The **Direction** drop-down list is used to specify the method of projection of the datum curve on the receiving surface or plane. The two options available in this drop-down list are:

1. Along direction
2. Normal to surface

Along direction

This option projects the datum curve in a direction shown by the yellow arrow. To specify the direction of projection you can select the default coordinate system, datum plane, edge, or surface. Figure 8-50 shows the datum curves that are overlapping. The datum curve that is selected to project on the receiving surface and the datum curve after projection are of same geometry. This is because, using the **Along direction** option, the true geometry is obtained after projection. Figure 8-51 shows the sketched datum curve after projecting on the receiving surface. The curve is sketched on a datum plane that is parallel to the bottom face of the model.

Normal to surface

This option of projecting a datum curve projects the datum curve normal to the receiving surface. Figure 8-52 shows the datum curve that is selected to project on the receiving

surface and the datum curve after projection. Figure 8-53 shows the sketched datum curve after projecting on the receiving surface.

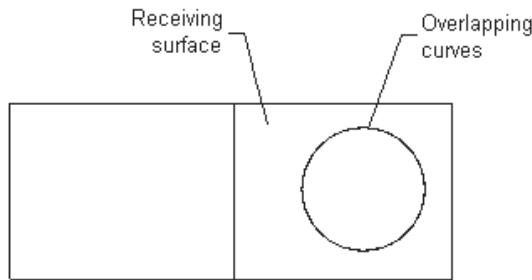


Figure 8-50 Top view of the projected datum curve

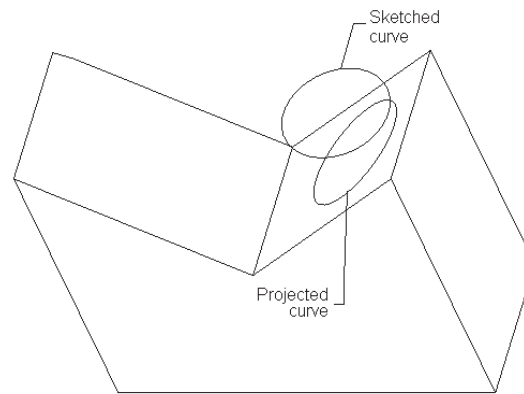


Figure 8-51 Projecting a datum curve

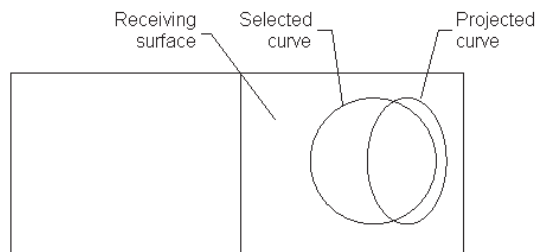


Figure 8-52 Top view of the projected datum curve

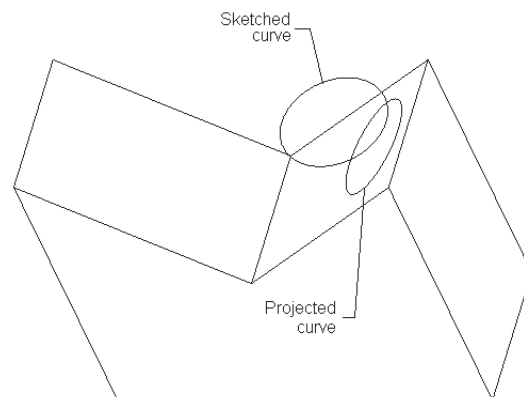


Figure 8-53 Projecting a datum curve

References Tab

When you choose the **References** tab, the slide-down panel is displayed, as shown in Figure 8-54. This slide-down panel is used to select an existing datum curve to project or sketch a datum curve to project. The drop-down list in the slide-down panel lists two options, **Project chains** and **Project a sketch**. These two options are discussed next.

Project chains

The **Project chains** option is used when the datum curve to project exists. Click in the **Chains** collector to make it yellow in color and then select the datum curve that you need to project. After selecting the datum curve, click in the **Surfaces** collector and select the receiving surface (plane or surface to project on to). Then you need to specify the direction of projection. Remember that if you are using the **Normal to surface** option, then you do

not need to specify the direction of projection. The **Direction Reference** collector is displayed while selecting the **Along direction** option, from the dashboard, whereas it does not appear while selecting the **Normal to surface** option.

Project a sketch

When you select the **Project a sketch** option from the drop-down list in the **References** slide-down panel, the slide-down panel appears, as shown in Figure 8-55.

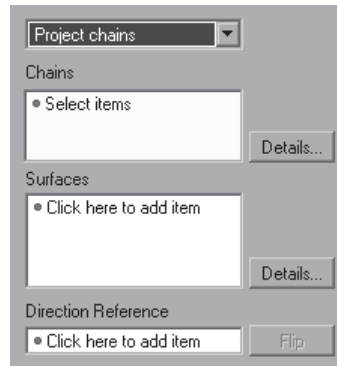


Figure 8-54 The **References** tab slide-down panel with the **Project chains** option selected

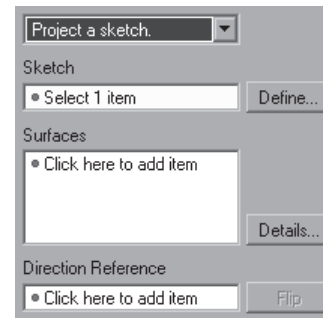


Figure 8-55 The **References** tab slide-down panel with the **Project a sketch** option selected

Choose the **Define** button to invoke the **Sketch** dialog box and to select the sketch plane for the curve to be projected. After sketching, exit the sketcher environment and select the surface or plane on which the curve will be projected. If you are using the **Along direction** option to project the curve then you need to click in the **Direction Reference** collector to specify the direction of projection.

Wrap Option

This option is used to create a datum curve by wrapping a sketched entity around a solid or a quilt. Choose **Edit > Wrap** from the menu bar, the **Wrap** dashboard is displayed, as shown in Figure 8-56. The tools and options in this dashboard are discussed next.

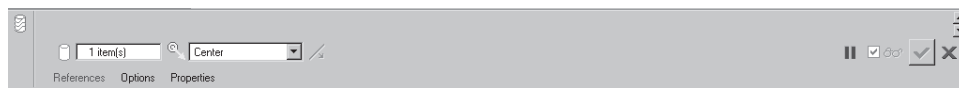


Figure 8-56 The **Wrap** dashboard

References Tab

When you choose the **References** tab, the slide-down panel is displayed, as shown in Figure 8-57. The **Define** button is used to invoke the **Sketch** dialog box that you can use to select the sketch plane and enter the sketcher environment to draw the curve.

The **Destination** collector in the slide-down panel is used to select the object on which you need to wrap the curve. Generally, if there is a single feature in the drawing area, then you do not need to select the object to wrap on. Pro/ENGINEER automatically

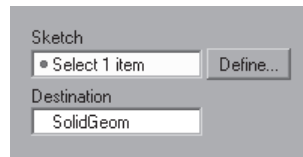


Figure 8-57 The **References** tab slide-down panel

wraps the selected curve or the sketched curve on the object. However, if you want to select a different object to wrap on then you can click in this collector to make it yellow in color and then select the object.

Options Tab

When you choose the **Options** tab, the slide-down panel is displayed, as shown in Figure 8-58.

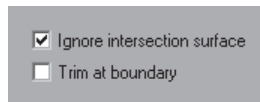


Figure 8-58 The **Options** tab slide-down panel

The **Ignore intersection surface** check box when selected ignores any intersection surface and wraps the selected curve on the destination object.

The **Trim at boundary** check box when selected trims the extra portion of the curve that is beyond the boundary of the destination object.

Follow the steps given below to sketch and wrap a curve on the rectangular block, as shown in Figure 8-59. It is assumed that the rectangular block of dimension 20x20x50 exists.

1. Choose **Edit > Wrap** from the menu bar to invoke the **Wrap** dashboard.
2. Choose the **References** tab to invoke the slide-down panel.
3. Choose the **Define** button to invoke the **Sketch** dialog box.
4. Choose the datum plane that is passing through the center of the rectangular block as the sketch plane and then choose a reference for orienting the sketch plane.
5. After entering the sketcher environment, draw a line that starts from the bottom of the rectangular block. Its start point is aligned with the center of the rectangular block and with the bottom edge. The end point of the line is at a distance of 1000 and at a height of 50 from the bottom of the rectangular block, see Figure 8-60.
6. Place a user-defined coordinate system at the start point of the line.
7. Exit the sketcher environment. The sketched curve is automatically wrapped on the circumference of the cylinder.
8. Choose the **Build feature** button to exit the **Wrap** dashboard.

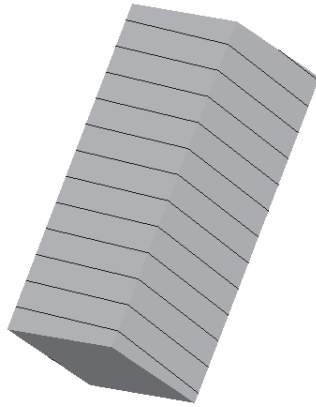


Figure 8-59 Curve wrapped on the rectangular block



Figure 8-60 Sketch of the curve to wrap

CREATING DRAFT FEATURES



In Pro/ENGINEER, drafts are created on existing surfaces. They are created by rotating the selected surface by a certain angle. One of the applications of draft features is in moulds and castings where a taper is required to separate the casting from the mould or vice versa. To create a draft, choose the **Draft** button from the **Engineering Features** toolbar. The **Draft** dashboard is displayed, as shown in Figure 8-61. The options in this dashboard are discussed next.

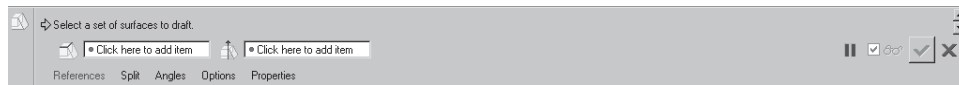


Figure 8-61 The **Draft** dashboard

References Tab

When you choose the **References** tab, the slide-down panel is displayed, as shown in Figure 8-62. The options in this slide-down panel are discussed next.

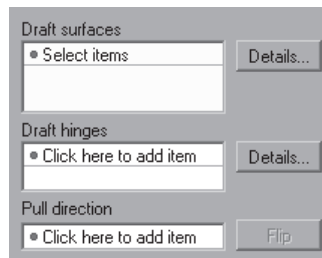


Figure 8-62 The **References** tab slide-down panel

Draft surfaces Collector

The **Draft surfaces** collector is selected by default. If this collector is clear by default, click in

this collector and then select the surface to which you need to add a draft angle. The maximum draft angle that can be added to a surface is 30-degree.

Draft hinges Collector

The **Draft hinges** collector is used to select the hinge of the draft surface. The hinge that you select can be an edge, a surface, an axis, or a datum plane. The draft surface is pivoted on the hinge that you select. In other words, the draft surface is rotated about the hinge. The hinge that you select need not intersect the draft surface. Figure 8-63 shows the draft surface and the hinge selected to create the draft and the resultant draft surface is shown in Figure 8-64.

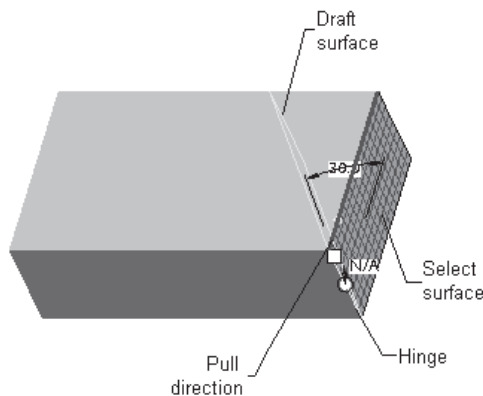


Figure 8-63 Parameters needed to create a basic draft surface

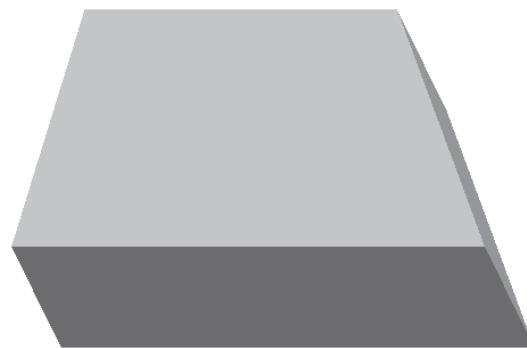


Figure 8-64 Resultant draft surface

Pull direction Collector

The **Pull direction** collector is used to specify the direction of rotation of the draft surface. The direction of pull is shown by the direction of the yellow arrow. Generally, when you select the hinge, the pull direction is selected by default. If the pull direction shown by the yellow arrow is not that is desired, click in the **Pull direction** collector and then select the pull direction. You can change the direction of the yellow arrow by choosing the **Flip** button.

Split Tab

When you choose the **Split** tab, the slide-down panel is displayed, as shown in Figure 8-65. The options in this slide-down panel are discussed next.

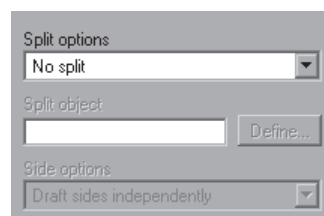


Figure 8-65 The **Split** tab slide-down panel

Split options Drop-down List

The options in this drop-down list are used to split the surface selected to draft. Using these options the selected surface gets split into two surfaces and different draft angles can be applied to both the surfaces. The options available in the **Split options** drop-down list are discussed next.

No Split

This option is used when you do not want to split the surface selected to give the draft angle.

Split by draft hinge

This option is available only when you have selected the hinge for the draft surface. When you select this option, the selected surface is divided into two surfaces at the location on the surface where the hinge intersects it. Figure 8-66 shows the surface that is split into two surfaces at the hinge.

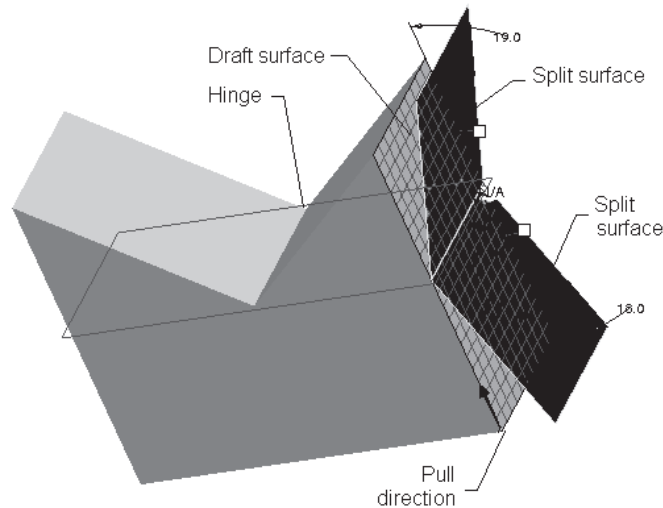


Figure 8-66 Split at draft hinge

Split by split object

This option when chosen activates the **Split object** collector and the **Define** button. This option is used to split the surface selected to draft by drawing a sketch. The surface gets split into two surfaces and the sketch defines the profile of the split. Figure 8-67 shows the parameters that you need to define to create a draft using split by split object. Figure 8-68 shows the draft created on the cylindrical surface.

Side options Drop-down List

The options in this drop-down list are used to specify how to apply the draft once the surface has been split. The options in this drop-down list are discussed next.

Draft sides independently

The two sides of the draft surface that are formed when the surface is split can be given

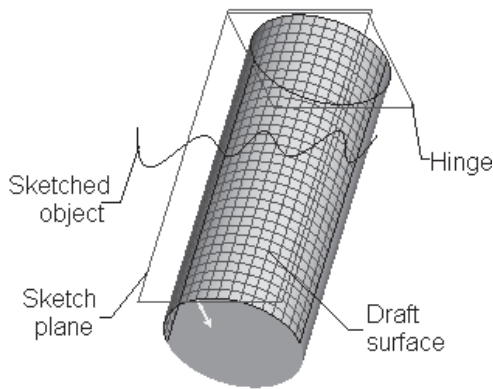


Figure 8-67 Split by split object



Figure 8-68 Resultant model

different draft angles. When you choose this option, the edit boxes appear on the **Draft** dashboard. You can enter the values of the draft angle in these edit boxes.

Draft sides dependently

The two sides of the split surface are given the same draft angle.

Draft first side only

Using this option you can apply the draft angle only to the first side of the split surface.

Draft second side only

Using this option you can apply the draft angle only to the second side of the split surface.

Angles Tab

Choose the **Angles** tab from the **Draft** dashboard after you have specified some angle for draft surface. When you choose this option, the slide-down panel is displayed, as shown in Figure 8-69.

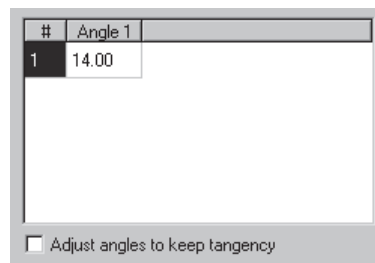


Figure 8-69 The Angles tab slide-down panel

In Pro/ENGINEER, you can create constant as well as variable angle drafts.

Constant Angle Drafts

By default, Pro/ENGINEER creates a constant angle draft. This means, the same draft angle is applied to the surface that you have selected.

Variable Angle Drafts

When you apply more than one value of the draft angle to the selected surface, it is called variable angle draft. Apply the variable angle draft after you have selected the hinge, the pull direction, and the draft surface.

To create the draft surface shown in Figure 8-70, follow the steps given next. It is assumed that you have the base feature of the model, as shown in Figure 8-71.



Figure 8-70 Draft feature on the model

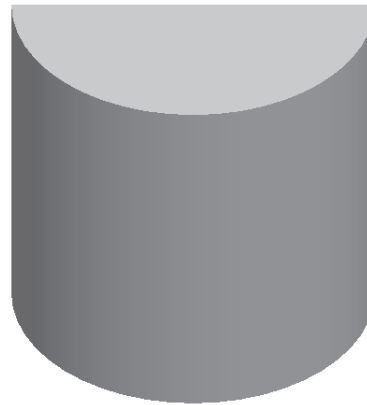


Figure 8-71 Base feature

1. Invoke the **Draft** dashboard and click in the first collector from left.
2. Select the top face of the base feature. Notice that in the Pull direction collector the direction of pull is selected automatically. Reverse the pull direction by choosing the Reverse pull direction button available on the right of the second collector.
3. Hold down the right mouse button to invoke the shortcut menu shown in Figure 8-72.

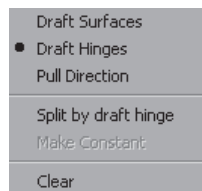


Figure 8-72 Shortcut menu

4. Choose the **Draft Surfaces** option from the shortcut menu and select the cylindrical surface of the base feature to add the draft angle. The drag handle appears on the base feature; you can use it to vary the angle of the draft. The draft angle value of **1** also appears in the drawing area. Also notice the white ball that appears on the edge of the top face (face selected as hinge).

- Bring the cursor on the white ball and hold the right mouse button to invoke the shortcut menu shown in Figure 8-73.



Figure 8-73 Shortcut menu

- Choose the **Add Angle** option from the shortcut menu. Notice that now there are two white balls on the edge of the top face. On the first ball the value of **0.5** appears. This value varies from one end of the face to the other end. This value represents the location of the point that you need to use for reference in order to apply the draft angles.
- Again bring the cursor to any one of the two white balls and invoke the shortcut menu.
- Choose the **Add Angle** option from the shortcut menu. Another point is added with a location value on the edge.
- Add eleven such points for locations varying from **0** to **1**.
- After adding eleven points, double-click on any of the location value that is present in the drawing area. The edit box appears, enter the value of **0** in this edit box. Similarly, locate all remaining ten points and increment them by 0.1. Figure 8-74 shows the model after locating all eleven points on the edge.
- Double-click on the value of the angle that corresponds to the location **0**. The edit box appears, enter a value of **15**.
- Double-click on the value of the angle that corresponds to the location **0.1** (these values appear above the model and have a default value of 1.0). The edit box appears; enter a value of **5**. Similarly, vary the angle at other locations. Every alternate location should have an angle of **15**. Figure 8-75 shows the preview of the model after modifying the values of the angle at all eleven locations.
- After modifying the values of the angle at all locations, choose the **Feature Preview** button from the **Draft** dashboard. The model appears, as shown in Figure 8-76.
- The model after mirroring the geometry and then shelling the top and bottom faces is shown in Figure 8-77.

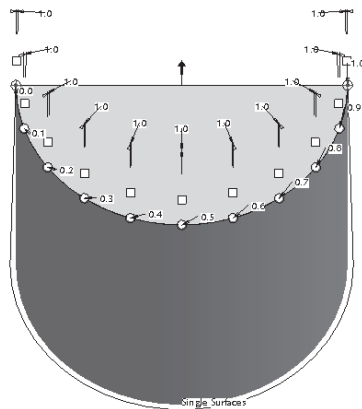


Figure 8-74 Location points on the edge

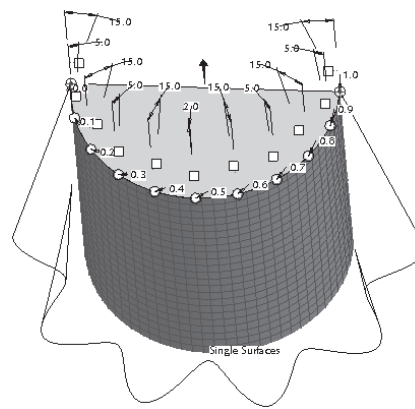


Figure 8-75 Location points with modified angle values



Figure 8-76 Model with the draft feature



Figure 8-77 Model of the lamp shade

Options Tab

When you choose this tab, the slide-down panel is displayed, as shown in Figure 8-78. The options in this slide-down panel are discussed next.

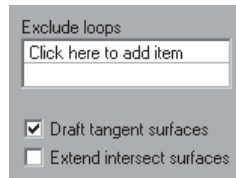


Figure 8-78 The Options tab slide-down panel

Exclude loops Collector

This collector is used to select a surface to which you do not want to add a draft angle. When you select a surface to add a draft angle, all loops that are on the surface are applied the same draft angle. However, using this collector you can select the loop to which you do not want to add a draft angle.

Figure 8-79 shows surface, when selected to add a draft angle, also selects the surface of the cylindrical feature. If you continue with the draft feature creation and exit the **Draft** dashboard, the draft is created, as shown in Figure 8-80.

To exclude the face of the cylindrical feature from the loop, click in the **Exclude loops** collector to turn it yellow in color. Now, bring the cursor close to the face of the cylindrical feature. The boundary of the face is highlighted in cyan. Select the face to exclude it from the loop. Figure 8-81 shows the draft surface after excluding the face of the cylindrical feature from the loop.

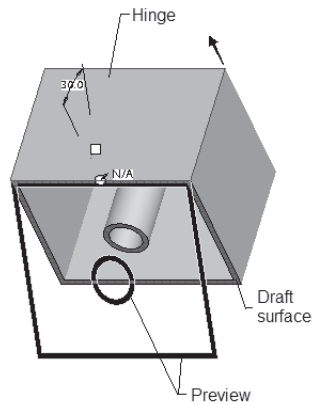


Figure 8-79 Surface for draft

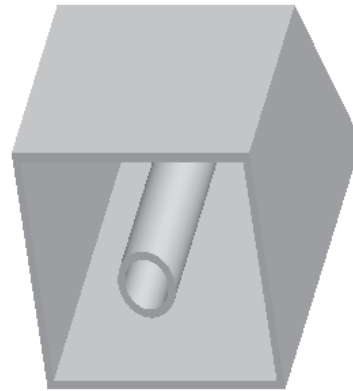


Figure 8-80 The draft created

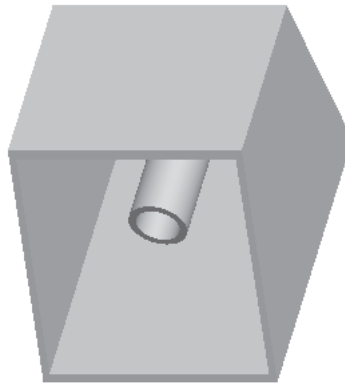


Figure 8-81 Draft after excluding the face of cylindrical feature

Draft tangent surfaces Check Box

When the **Draft tangent surfaces** check box is selected, it applies the draft to the surfaces tangent to the selected surface. In Figure 8-82, the surface shown is selected to add draft angle. Because the **Draft tangent surfaces** check box is selected, all surfaces that are tangent to the selected surface and other surfaces are automatically selected. Figure 8-83 shows the resultant model with the draft feature.

Extend intersect surfaces Check Box

The **Extend** check box is available only when the **Intersect** radio button is selected. When this check box is selected, the draft surface extends in the direction of the feature it intersects. Figure 8-84 shows the example of the draft that intersects with the adjacent feature and projects outside the edge of the feature at the bottom. Figure 8-85 shows the feature when the draft surface gets extended.

Figure 8-86 shows another example of extended draft feature. The following steps explain in brief the procedure to create the pencil-shaped model. It is assumed that you have created the model shown in Figure 8-87. The initial model shown is an octagon on the top of which a cylinder is created.

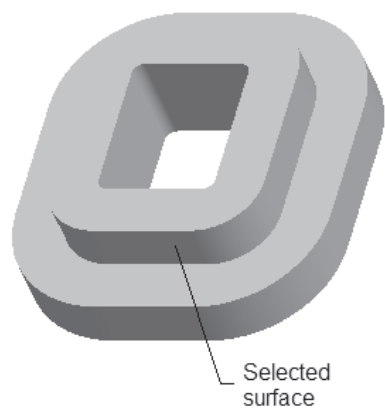


Figure 8-82 Single surface selected to draft



Figure 8-83 Draft applied to all tangent surfaces

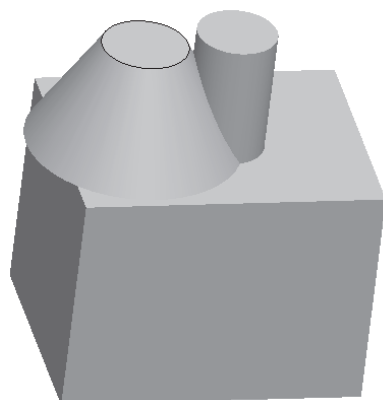


Figure 8-84 Draft created without selecting the *Extend intersect surface* radio button

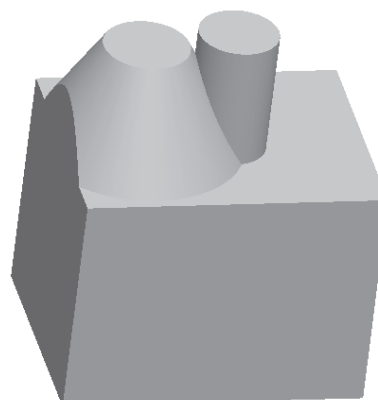


Figure 8-85 Draft created by selecting the *Extend intersect surface* radio button

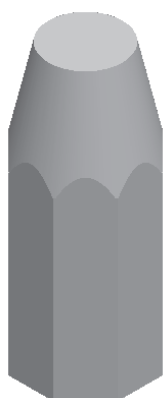


Figure 8-86 Example of extend intersect in a draft surface

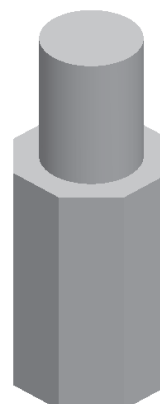


Figure 8-87 Model initially created

1. Invoke the **Draft** dashboard and select the cylindrical surface to draft.
2. Select the top face of the cylindrical feature as the hinge. The yellow arrow points in the upward direction.
3. Choose the **Options** tab and from the slide-down panel, select the **Extend intersect surfaces** check box.
4. Use the drag handle and increase the draft angle up to the vertex of the octagon. This can be easily done by viewing the preview of the draft surface as you increase the angle. You may even have to use the edit box present on the dashboard to enter the small increase in angle value. Remember that you will get the desired shape of the intersect only when the draft surface intersects the vertices of the octagonal feature.

Properties Tab

When you choose the **Properties** tab, the slide-down panel displays the feature id of the feature you have creating.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 8-88. This figure also shows the sectioned front view, top view, and the right-side view of the solid model with dimensions. The hidden lines are suppressed for clarity. **(Expected time: 45 min)**

The following steps are required to complete this tutorial:

- a. Examine the model and determine the number of features in it. The model is composed of six features, refer to Figure 8-88.
- b. Create the base feature, refer to Figures 8-89 through 8-91.
- c. Create the shell feature of given thickness, refer to Figure 8-92.
- d. Create the third and fourth extrude features on the two ends of the swept feature respectively, refer to Figures 8-93 and 8-94.
- e. Create the counterbore hole on the third and forth feature, refer to Figures 8-95 and 8-96.
- f. Pattern the counterbore holes, refer to Figure 8-97.

When the Pro/ENGINEER session is started, the first task is to set the working directory. Since this is the first tutorial of this chapter, you need to create a folder named *c08* if it does not exist and set it as the Working Directory.

Starting a New Object File

1. Start a new part file and name it as *c08tut1*.

The three default datum planes are displayed in the drawing area. The **Model Tree** is also displayed in the drawing area. Close the **Model Tree** by clicking on the sash present on the right edge of the **Model Tree**.

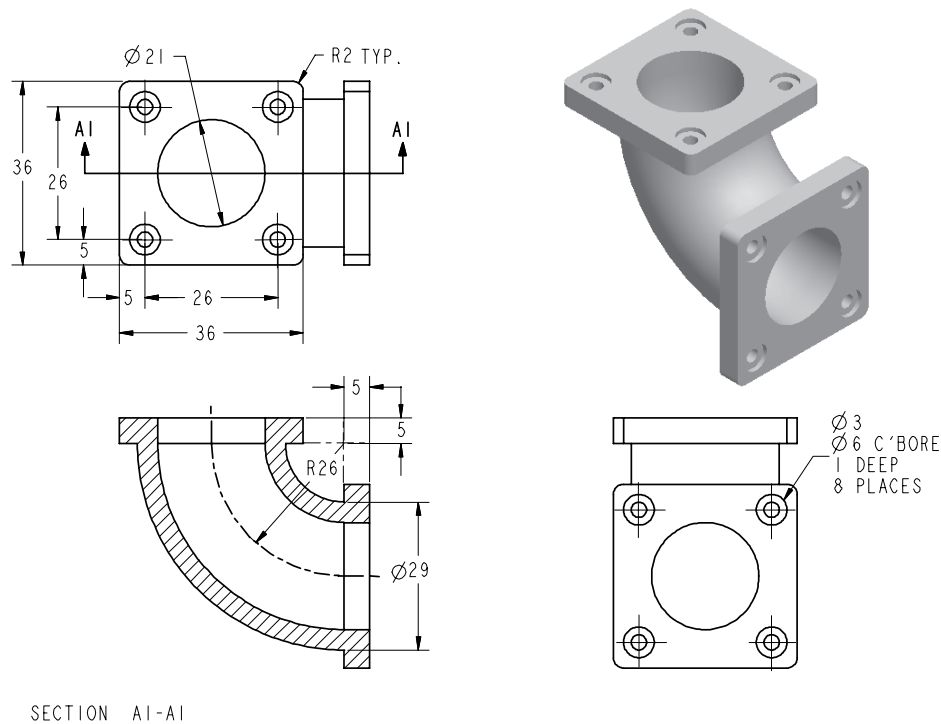


Figure 8-88 Top, front, right-side, and isometric views of the model.

Invoking the Sweep Option

You will use the menu bar present on the top of the screen to invoke the **Sweep** option.

1. Choose **Insert > Sweep > Protrusion** from the menu bar. The **SWEEP TRAJ** menu and the **PROTRUSION: Sweep** dialog box are displayed.
2. Choose the **Sketch Traj** option from the **SWEEP TRAJ** menu. You are prompted to select or create a sketching plane.

Selecting the Sketching Plane

The trajectory of the sweep feature will be sketched on the **FRONT** datum plane.

1. Select the **FRONT** datum plane as the sketching plane. A red arrow points in the direction of viewing the sketch plane.
2. Choose **Okay** from the **DIRECTION** submenu; the **SKET VIEW** submenu is displayed.
3. Choose **Top** from this submenu and select the **TOP** datum plane.

After you select the planes for orientation, the system takes you to the sketcher environment.

Drawing the Trajectory

From the model it is evident that the trajectory for the sweep feature is a quarter circle.

1. Choose the **Center and Ends** button from the **Sketcher Tools** toolbar. This button is available on the flyout that is displayed when you choose the black arrow on the right of the **3-Point / Tangent End** button.
2. Draw an arc such that the center of the arc lies at the intersection of the **TOP** and **RIGHT** datum planes, as shown in Figure 8-89. As you specify the center of the arc, the cursor snaps to the point of intersection of the two planes. Now, draw the arc and exit this tool. The endpoints of the arc are automatically aligned to the **TOP** and **RIGHT** datum planes. You will notice that an arrow is attached at the start point of the trajectory. This arrow points in the direction of sweep.



Tip: To change the start point on the trajectory, select the point where you want the start point. When the point is highlighted in red color, hold down the right mouse button to invoke a shortcut menu. Choose the **Start Point** option.

Modifying the Dimensions of the Trajectory

When you were drawing the arc, the arc was dimensioned automatically and a weak radial dimension was assigned to it. You need to modify the dimension as per your requirement.

1. Double-click on the dimension and modify the radial dimension to 26, as shown in Figure 8-90. You will notice the sketch refits on the screen.
2. Choose the **Done** button.

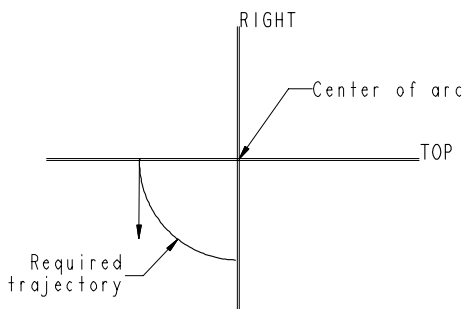


Figure 8-89 The sketch of required trajectory

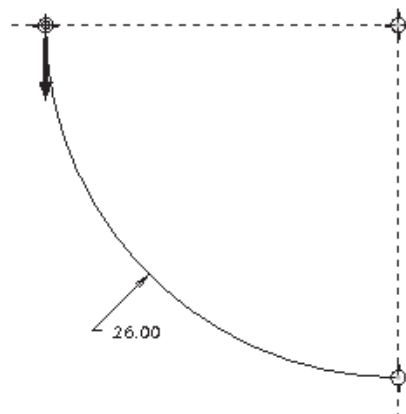


Figure 8-90 Dimension for the arc

Drawing the Section for Sweep

After choosing the **Done** button, the direction of viewing is modified such that the start point of the trajectory becomes normal to the screen. A yellow cross of infinite length appears on the screen. This cross consists of two perpendicular lines of infinite length.

The intersection point of these lines is the start point of the trajectory. The center of the circle should lie at the intersection of these lines. You are also prompted to draw the cross-section for the sweep.

1. Choose the **Center and Point** button and create a circle such that the center of the circle lies at the intersection of the two infinite perpendicular lines.

When you draw the circle, the cursor snaps to the intersection point of the cross.



Tip: When you enter the sketcher environment to define a section for sweep trajectory, it is often difficult to understand the orientation of the view. For this purpose, a yellow-colored cross that determines the orientation of the section with respect to the trajectory is available.

Modifying the Dimensions of the Section

1. Choose the **One-by-One** button and using the left mouse button, double-click on the dimension and modify the diameter dimension to 29.

The sketch accepts the value and refits on the screen.

2. Choose the **Done** button.

Previewing the Swept Feature

The sweep feature is completed and now it can be previewed.

1. Choose the **Preview** button from the **PROTRUSION: Sweep** dialog box that is present on the top right corner of the window.
2. Choose the **Named View List** button from the **View** toolbar; the flyout is displayed. Next, choose the **Default Orientation** option from the flyout; the model orients in the drawing area, as shown in Figure 8-91.

You can use the middle mouse button to change the orientation of the model.

3. Now, choose the **OK** button from the **PROTRUSION: Sweep** dialog box to exit it.


Creating the Shell Feature

The sweep feature is completed and now you can create the next feature. The next feature is the shell feature.



Note

Instead of using the **Shell** option, two concentric circles can be drawn while drawing the section for the sweep feature in order to obtain the desired hollow feature. Also, the **Sweep > Thin Protrusion** option can be used to obtain the same hollow feature. However, in this tutorial you will use the **Shell** option.

1. Choose the **Shell** button from the **Engineering Features** toolbar. The **Shell** dashboard is displayed and you are prompted to select the surfaces to be removed. 
2. Select one end surface of the swept feature and then using CTRL+left mouse button, select the other end surface. The two surfaces selected are highlighted in red.
3. Enter the thickness value of the shell as **4** in the dimension edit box present on the **Shell** dashboard and press ENTER.
4. Choose the **Build feature** button to exit the **Shell** dashboard.

The default trimetric view of the shell feature is shown in Figure 8-92. You can use the middle mouse button to view the model from various directions.

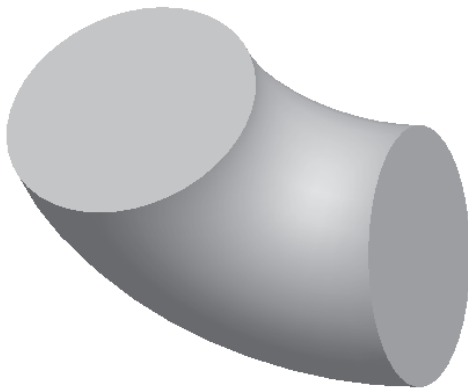


Figure 8-91 Trimetric view of the model

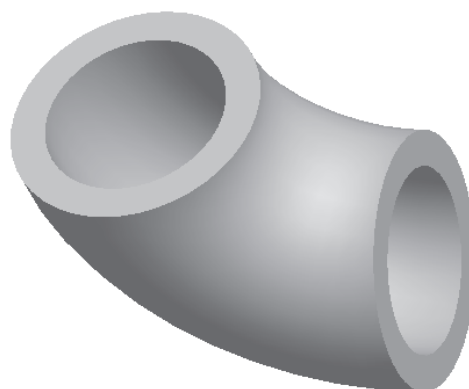


Figure 8-92 Shell feature without datum

Creating the Extrude Features

The next feature is a protrusion feature with a depth of extrusion of 5 and is created at both the ends of the swept feature. While drawing the circle of the sketch for the extrude feature, remember to use the edge of the shell in order to create a hole in the extruded feature also.

1. Choose the **Extrude** button from the **Base Features** toolbar. The **Extrude** dashboard is displayed.
2. Choose the **Placement** tab to display the slide-down panel and invoke the **Sketch** dialog box. Select the top face of the swept feature as the sketching plane.
3. Select the **RIGHT** datum plane and then choose the **Right** option from the **Orientation** drop-down list and then enter the sketcher environment.
4. The **References** dialog box is displayed with the status **Partially Placed**. Select the **FRONT** datum plane. Now note that the status displayed in the **Reference Status** area of the

References dialog box is **Unsolved sketch**. Choose the **Solve** button in the dialog box; now the status changes to **Fully Placed**. Next, choose the **Close** button to exit it.

5. Draw the sketch of the extrude feature and apply constraints and dimensions, as shown in Figure 8-93.
6. Create the extruded feature having a depth extrusion of 5. Similarly, create the next extruded feature at the other end of the swept feature. Select the sketching plane, draw the sketch similar to the first extruded section, apply the same dimensions and constraints, and extrude the sketch to the given distance.

The protrusion features created at both the ends of the sweep feature are shown in Figure 8-94.

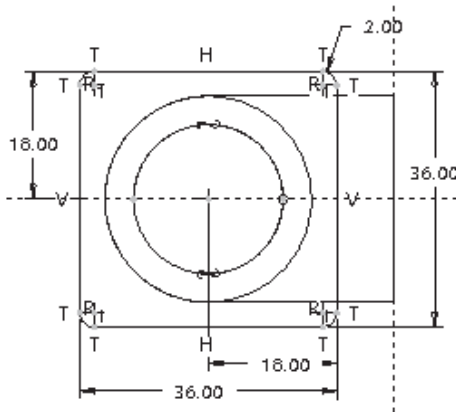


Figure 8-93 Sketch with dimensions and constraints

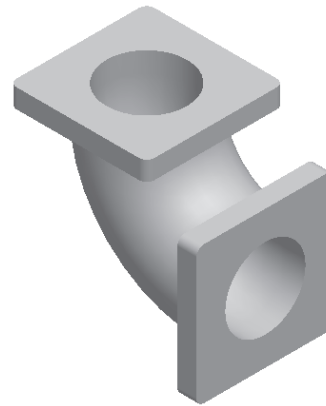


Figure 8-94 Two extruded features created on the ends of the sweep feature

Creating the Hole Feature

After creating extruded features at both the ends of the swept feature, counterbore holes will be created. One hole is to be created on each extruded surface and then they are to be patterned on individual planes separately to create the remaining three instances.

1. Choose the **Hole** button from the **Engineering Features** toolbar. The **Hole** dashboard is displayed.
2. Choose the **Use sketch to define drill hole profile** button from the **Hole** dashboard; the **Open an existing sketched profile** and **Activates Sketcher to create section** buttons are displayed.
3. Choose the **Activates Sketcher to create section** button; the system takes you to the sketcher environment. Sketch the section for the counterbore hole, as shown in Figure 8-95. Sketch a center line, apply constraints, and diametrically dimension the entities.
4. After completing the sketch, choose the **Done** button to exit the sketcher environment.

The system exits the sketcher environment and the **Hole** dashboard is redisplayed. Now, you are prompted to select a surface, axis, or point to place the hole.

5. Select the top face of the third extruded feature for the placement of hole. The preview of the hole appears in the drawing area.

Now, you need to specify the placement parameters for the hole. For this purpose refer to Figure 8-88 and look for the dimensions that can help in placing the hole. The two edges are used to dimension the hole.

6. Choose the **Placement** tab and click in the **Offset References** collector to turn it yellow.
7. Select the front edge and then use CTRL+left mouse button to select the left edge of the third feature for specifying the linear references. The hole is at a distance of 5 from both the edges. Default dimensions appear on the hole.

You can also drag the handle and place them at the two edges.

8. Double click on the dimensions and modify the linear distances to 5.
9. Choose the **Build feature** button from the **Hole** dashboard. The hole is created on the selected face.
10. Create another hole on the fourth feature using the same procedure as discussed above. The default trimetric view of the model that is completed until now is shown in Figure 8-96.

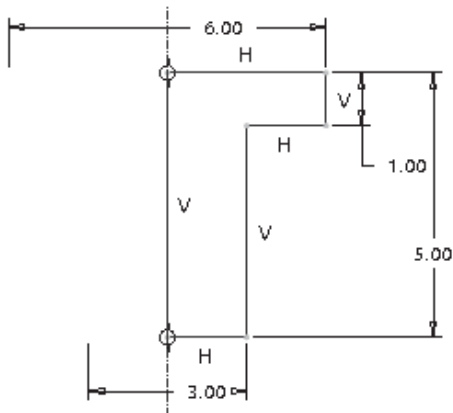


Figure 8-95 Sketch with dimensions and constraints for the counterbore hole

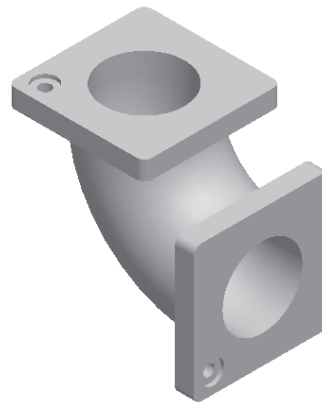


Figure 8-96 One hole each on the two extruded features

Creating the Pattern of the Holes

After one hole is placed on each of the two faces, they are patterned.



Tip: To draw the sketch of the second hole when you enter the sketcher environment do not again draw the same sketch that was drawn for the first hole. Instead, import the sketch for the hole from the **In Session** folder. The **In Session** folder contains all files that you have created in the current session.

To import the sketch of the hole in the sketcher environment, choose **Sketch > Data from File**. The **Open** dialog box appears. Choose the **In Session** folder to display the files created in the current session. Select the third .sec file and open it. The sketch file for the hole will be the third file because you have created three sketches since this tutorial was started.

1. Select the hole feature from the **Model Tree** or from the drawing area and hold down the right mouse button to invoke the shortcut menu. Choose the **Pattern** option from the shortcut menu. The **Pattern** dashboard is displayed.
2. Create the pattern of the hole.

Similarly, create pattern of the hole on the other extruded feature also. The default trimetric view of the complete model is shown in Figure 8-97.

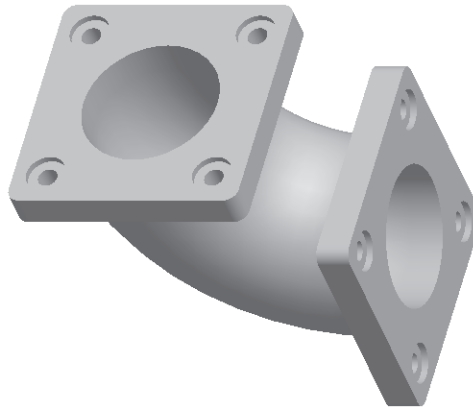


Figure 8-97 The default trimetric view of the complete model

Saving the Model

1. Choose the **Save** button from the **File** toolbar and save the model.



Note

As discussed in earlier chapters, the model tree is used to get an idea of the order of feature creation. In the **Model Tree**, the id numbers displayed in front of the features may be different when you create the features.

Tutorial 2

In this tutorial, you will create the model of the pencil shown in Figure 8-98 and then change the color of the pencil. Figure 8-99 shows the front view, the section view, and the detail view of the model. **(Expected time: 30 min)**



Figure 8-98 Solid model of the pencil

The following steps are required to complete this tutorial:

- Examine the model and determine the number of features in it. The model is composed of six features, refer to Figure 8-98.
- Create the base feature, refer to Figures 8-100 and 8-101.
- The second feature is a cylinder on the top face of the base feature, refer to Figure 8-102.
- Create the draft feature, refer to Figure 8-103.
- Create the revolve feature that will create the tip of the pencil, refer to Figures 8-104 and 8-105.
- Create the revolve feature that will create the tail-end of the pencil, refer to Figure 8-106 and 8-107.
- Write the text on the pencil by using the sketched datum curves, refer to Figure 8-108 and 8-109.

Make sure that *c08* is the current Working Directory.

Starting a New Object File

- Start a new part file and name it as *c08tut2*.

The three default datum planes are displayed in the drawing area.

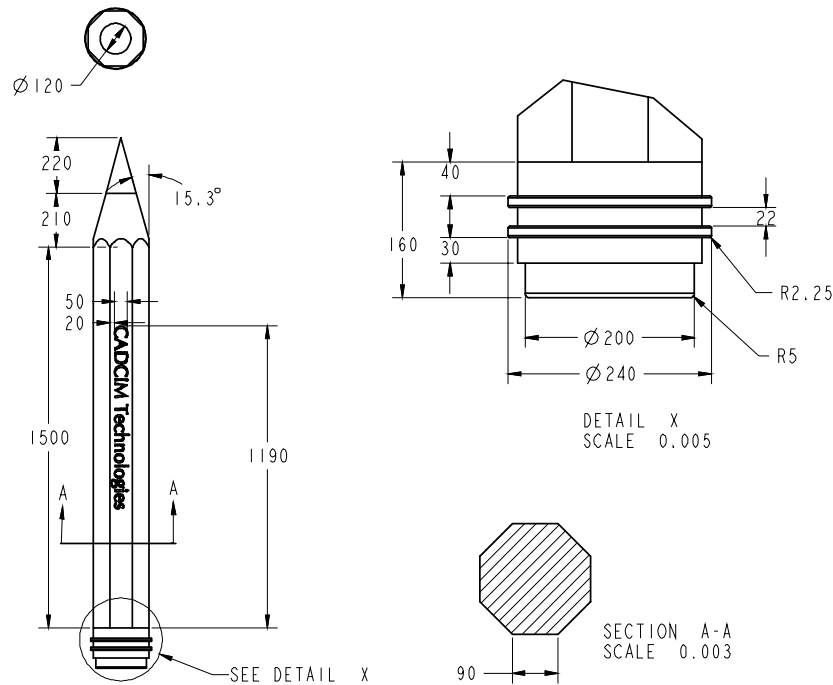


Figure 8-99 Top, front, sectioned, and detail views of the model

Creating the Base Feature

The base feature of the pencil is the protrusion feature in which the octagon is extruded to a depth of 1500.

1. Choose the **Extrude** button to display the **Extrude** dashboard and invoke the **Sketch** dialog box.
2. Select the **TOP** datum plane as the sketching plane.
3. Select the **RIGHT** datum plane from the drawing area and then select the **Right** option from the **Orientation** drop-down list, only if these are not selected by default.
4. Draw the sketch of the octagon and apply the constraints and dimensions, as shown in Figure 8-100. The construction circle is used to draw the octagon because it becomes easy and less time-consuming.
5. Exit the sketcher environment and extrude the sketch to a depth of 1500. The base feature is, as shown in Figure 8-101.

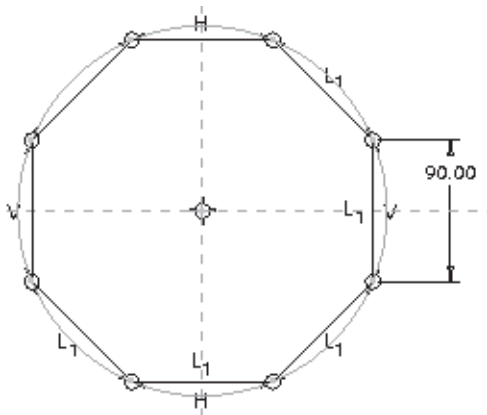


Figure 8-100 Sketch of the octagonal feature

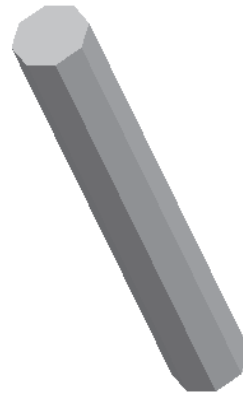


Figure 8-101 Octagonal sketch after extruding


Creating the Second Feature

The second feature is a cylindrical feature that is drawn on the top face of the base feature.

1. Choose the **Extrude** button from the **Base Features** toolbar; the **Extrude** dashboard is displayed. Invoke the **Sketch** dialog box and select the top face of the base feature as the sketching plane.
2. Enter the sketcher environment and draw the sketch for the cylinder. The diameter of the circle is 120.
3. After exiting the sketcher environment, extrude the circle to a depth of 210. The model after creating the cylindrical feature is shown in Figure 8-102.

Creating the Draft Feature

The draft feature on the cylindrical surface will be created and the draft surface will be allowed to intersect the base feature. This is because when the draft surface intersects the base feature and is extended, the required shape is obtained at the intersecting edge.

1. Choose the **Draft** button from the **Engineering Features** toolbar. The **Draft** dashboard is displayed. 
2. Select the cylindrical surface of the cylinder to draft.
3. Choose the **References** tab to invoke the slide-down panel. Click the **Draft hinges** collector and then select the top face of the cylindrical feature as the hinge. The yellow arrow points in the upward direction.
4. Choose the **Reverse pull direction** button present on the right of the **Pull direction** collector to change the direction of the arrow so that it points downward.

5. Choose the **Options** tab to invoke the slide-down panel and then select the **Extend intersect surfaces** check box.
6. Enter the angle value **15.3** in the dimension edit box present on the **Draft** dashboard.
7. Choose the **Build feature** button from the **Draft** dashboard to complete the draft feature, as shown in Figure 8-103.

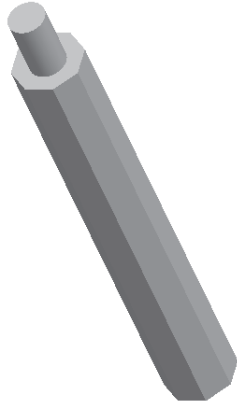


Figure 8-102 Model after creating the cylindrical feature

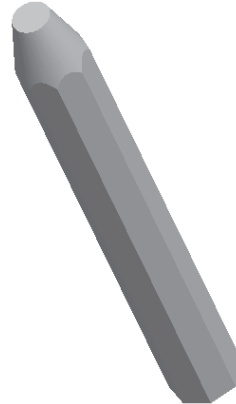


Figure 8-103 Model after creating the draft feature

Creating the Fourth Feature

The fourth feature is a revolve feature that will be used to create the tip of the pencil.

1. Choose the **Revolve** button from the **Base Features** toolbar; the **Revolve** dashboard is displayed.
2. Invoke the **Sketch** dialog box and select the **FRONT** datum plane as the sketching plane.
3. After entering the sketcher environment, draw the sketch, apply constraints, and dimension it, as shown in Figure 8-104.
4. After exiting the sketcher environment, the angle of revolution of 360-degree is selected by default.
5. Choose the **Build feature** button to complete the feature. The default trimetric view of the model after creating the fourth feature is shown in Figure 8-105.

Creating the Fifth Feature

The fifth feature is a revolve feature that is used to create the top portion of the pencil.

1. Choose the **Revolve** button from the **Base Features** toolbar; the **Revolve** dashboard is displayed.

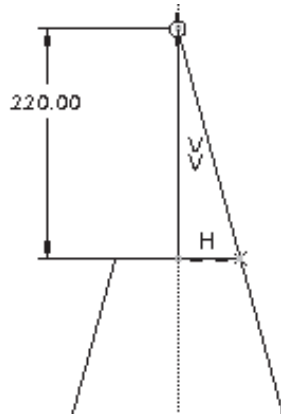


Figure 8-104 Sketch for the tip of the pencil

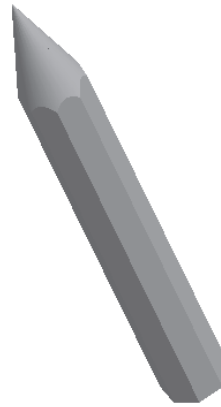


Figure 8-105 Model after creating the tip

2. Invoke the **Sketch** dialog box and select the **FRONT** datum plane as the sketching plane.
3. After entering the sketcher environment, draw the sketch, apply constraints, and dimension it, as shown in Figure 8-106.
4. After exiting the sketcher environment, the angle of revolution of 360-degree is selected by default.
5. Choose the **Build feature** button to complete the feature. The default trimetric view of the model after creating the fourth feature is shown in Figure 8-107.

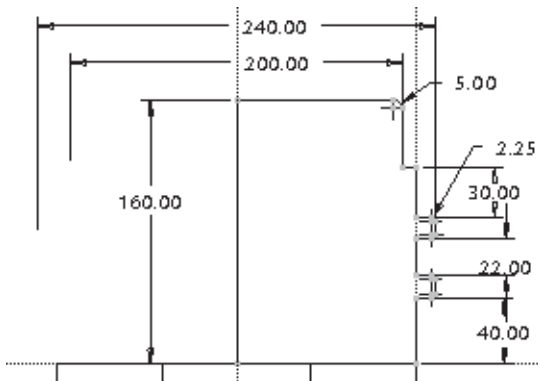


Figure 8-106 Sketch with dimensions and constraints



Figure 8-107 Model after creating the top of the pencil

Writing the Text

Generally, the text on a model should be written using the datum curves.

1. Choose the **Sketch** button from the **Right Toolchest**; the **Sketch** dialog box is displayed.

2. Select the front face of the base feature as the sketching plane.
3. After entering the sketcher environment, create the text using the **Text** button. Figure 8-108 shows the text written with dimensions.
4. Choose the **Done** button to exit the sketcher environment.

The model after creating the text is shown in Figure 8-109.

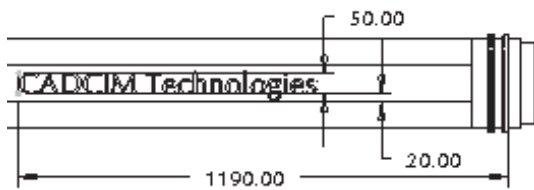


Figure 8-108 Text with dimensions



Figure 8-109 Model after creating the text

Changing the Colors

The colors in Pro/ENGINEER can be applied on selected surfaces. Remember that the colors you apply are saved with the model and remain on the model until you clear them.

1. Choose **View > Color and Appearance** from the menu bar; the **Appearance Editor** is displayed.
2. Select the +sign. The new color is added.
3. In the **Color** area, choose the **Color** button. The **Color Editor** is displayed. Set the RGB values to 0. The color obtained is black. You need to apply this color to the tip of the pencil.
4. Choose the **Close** button from the **Color Editor**.
5. Select the **Surfaces** option from the drop-down list present in the **Assignment** rollout. The **Select** message box is displayed and you are prompted to select the surfaces to alter colors.
6. Select the tip of the pencil and press the middle mouse button. Then choose the **Apply** button to apply the color to the selected surface. The tip of the pencil is changed to black.
7. To add another color to any of the surfaces of the pencil, select the +sign and modify the

color to the required color. Follow steps 3 to 6 to add colors to other surfaces of the pencil.

8. After modifying the colors, choose the **Properties** rollout and then choose the **Close** button.

Saving the Model

1. Choose the **Save** button from the **File** toolbar and save the model.



Note

Colors are not a feature of the model, and therefore, they will not appear in the **Model Tree**.

Tutorial 3

In this tutorial, you will create a blend feature shown in Figure 8-110. The two views of the blend feature are shown in Figure 8-111 with dimensions. After creating the model, you will redefine it such that the straight blending is changed into a smooth blending.

(Expected time: 45 min)

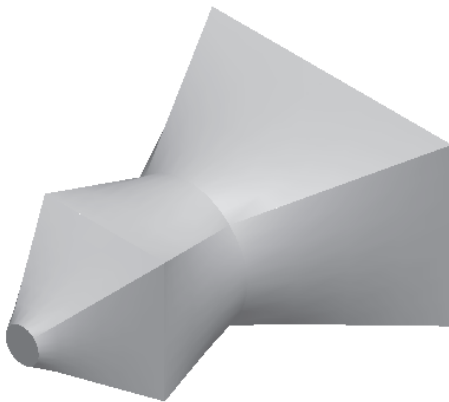


Figure 8-110 Isometric view of the model

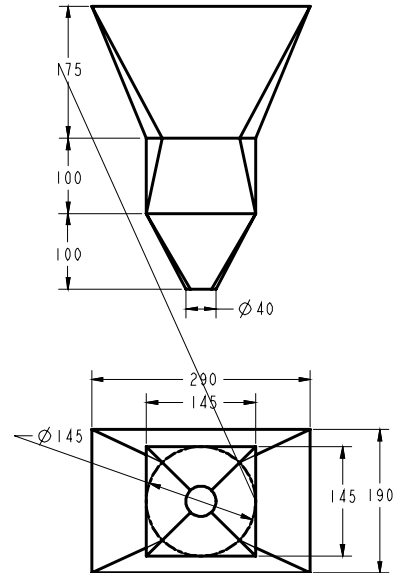


Figure 8-111 Top and front views of the model

The following steps are required to complete this tutorial:

- a. Examine the blend feature and determine the number of sections in this feature. The blend consists of four sections, refer to Figure 8-111.
- b. Create the sketch for the first section of the blend feature, refer to Figures 8-112.
- c. Create the sketch for the second section of the blend feature, refer to Figures 8-113.
- d. Create the sketch for the third section of the blend feature.
- e. Create the sketch for the fourth section, refer to Figures 8-114. Give the depth between section numbers 1 and 2, 2 and 3, and 3 and 4.
- f. Redefine the model to change the straight blending into a smooth blending, refer to Figure 8-116.

Make sure that *c08* is the current Working Directory.

Starting a New Object File

1. Start a new part file and name it as *c08tut3*.

The three default datum planes are displayed in the drawing area.

Invoking the Blend Option

1. Choose **Insert > Blend > Protrusion** from the menu bar. The **BLEND OPTS** menu is displayed.
2. Choose **Parallel > Regular Sec > Sketch Sec > Done** options from the **BLEND OPTS** submenu. The **ATTRIBUTES** menu is displayed and you will be prompted to choose the **Straight** or **Smooth** option from this menu.
3. Choose **Straight > Done** option from the **ATTRIBUTES** menu.

A **Smooth** blend will be created during the redefining of the model using the same sections that will be used to create the given model .

Selecting the Sketch Plane

1. Select the **FRONT** datum plane as the sketching plane. A red arrow points in the direction of feature creation and you are prompted to specify the direction of feature creation.
2. Choose **Okay** from the **DIRECTION** submenu; the **SKET VIEW** submenu is displayed.
3. Choose **Top** from this menu and using the left mouse button select the **TOP** datum plane present in the drawing area.

Drawing the First Section

The first section is a rectangle of 290x190 units.

1. Draw the sketch of the rectangular section and then add constraints and dimensions to it, as shown in Figure 8-112.

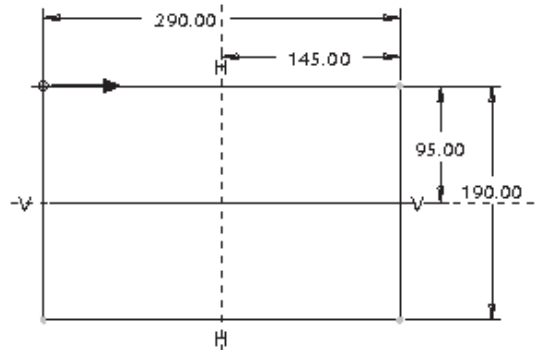


Figure 8-112 First rectangular section with dimensions

After drawing the rectangular section, you need to toggle the section and draw the next section.



Note

While drawing the sections for the blend feature, the start point is very important. The start point should be similar to those shown in the figures. If the start point is not at the desired point then select the point where you need the start point. Hold down the right mouse button to invoke a shortcut menu and choose the **Start Point** option.

Toggling the Section

Toggling of a section is required in order to sketch the next section. Since in this tutorial four sections are used to create the required model, therefore, whenever you finish drawing one section you need to toggle to the next section.

1. Choose **Sketch > Feature Tools > Toggle Section** from the menu bar. You can also hold down the right mouse button to display the shortcut menu and choose the **Toggle Section** option from the shortcut menu.

When you choose **Toggle Section**, the previous section becomes inactive and appears gray in color.



Tip: To toggle the section, simply right-click when you are in the selection mode. The selection mode in the sketcher environment can be invoked by choosing the **One-by-One** button.

Drawing the Next Section

The next section is a circle.

1. Draw the sketch of the circular section, refer to Figure 8-113. Modify the diameter of the circle to 145.

As discussed earlier, the number of entities per section must be equal in a blend feature. Since, a circle is a single entity, it should be divided at four points.

Dividing the Circular Section

The circular section should be divided at four points because the rectangle and square have four entities. When you divide a circle at four points, the number of entities becomes four.

1. Choose the black arrow on the right of the **Delete Segment** button; a flyout is displayed. Choose the **Divide** button from the flyout.
2. Select the circle at four points, as shown in Figure 8-113.

As you select points on the circle to divide it, some weak dimensions appear on the circle. Next, you need to apply constraints on the four points that were selected to divide the circle.

Applying Constraints on the Four Points

1. Choose the **Constrain** button; the **Constraints** dialog box is displayed.
2. Choose the **Make line or two vertices vertical** constraint button from the **Constraints** dialog box and select the two division points on the left side of the circle to lie in a vertical line. Similarly, select the two points on the right to apply the constraint.
3. Now, choose the **Make line or two vertices horizontal** constraint button and select the two division points on the upper half and the lower half to lie in a horizontal line.
4. Modify the vertical dimension of the upper left division point, as shown in Figure 8-113. After the circular section is completed, the two sections with dimensions will look similar to the sections shown in Figure 8-113.

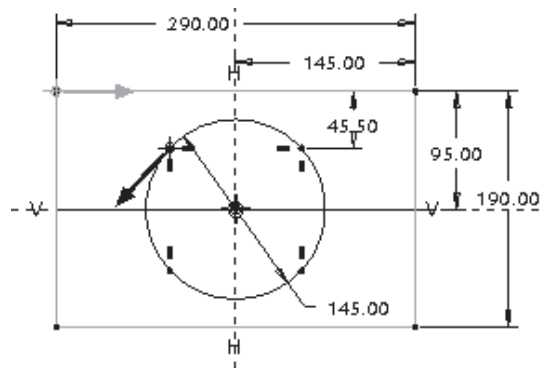


Figure 8-113 Two completed sections

5. Now, toggle the section and create the next section. The next section to be drawn is a

square. After drawing the square section, draw the circular section. Divide the circular section into four entities similar to section 2 and then constrain and dimension it.

Figure 8-114 shows all sections completed with dimensions.

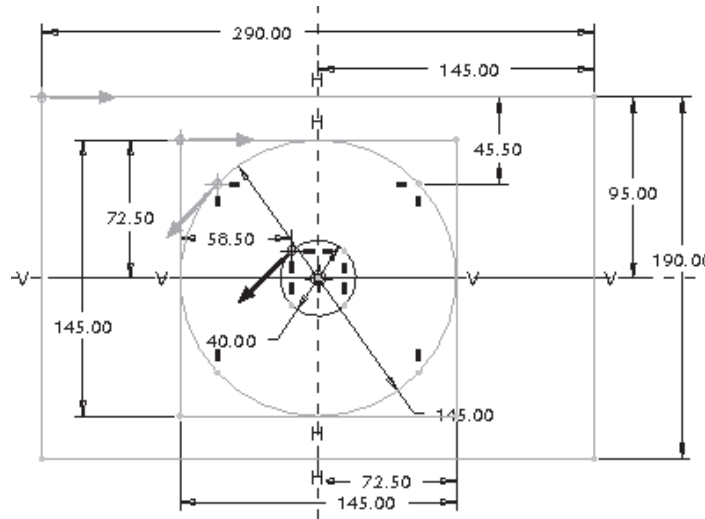


Figure 8-114 All four completed sections before giving depth



Note

In Figure 8-114, note the direction of the start points indicated by arrows. These arrows are important to avoid a twisted feature.

Applying Depth to the Sections

After the sketches of all sections are completed, you need to specify the depth between each section. The dimensions for depth between each section can be referred to from Figure 8-111.

1. Choose the **Done** button.

The **Message Input Window** appears and you are prompted to enter the depth for section 2.

2. Enter the value **175** in the **Message Input Window** and press ENTER.

Similarly, the **Message Input Window** appears again and you are prompted to enter the depth for section 3. Enter 100 for section 3 and 100 for section 4.

Previewing the Blend Feature

The blend feature is completed and it can now be previewed.

1. Choose the **Preview** button from the **PROTRUSION: Blend, Parallel, Regular Sections** dialog box that is displayed at the top right corner of the window.

2. Choose the **Default Orientation** option from the **Named View List** flyout; the model orients in the drawing area, as shown in Figure 8-115.
3. Now, choose the **OK** button that is present on the **PROTRUSION: Blend, Parallel, Regular Sections** dialog box; the trimetric view of the model is displayed in the drawing area.

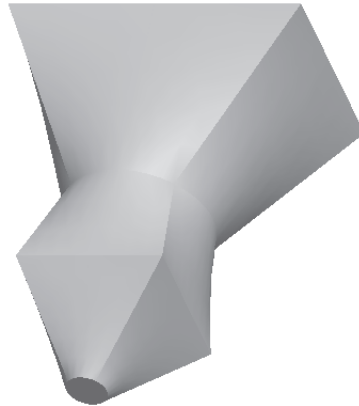


Figure 8-115 Trimetric view of the model

Saving the Model

1. Choose the **Save** button from the **File** toolbar and save the model.

Redefining the Blend Feature

After saving the straight blend feature, you will redefine this feature so that it is converted into a smooth blend.

1. Select the model in the drawing area. The edges of the model turn red in color.
2. Press and hold down the right mouse button in the drawing area until a shortcut menu is displayed.
3. Choose the **Edit Definition** option from the shortcut menu; the **PROTRUSION: Blend, Parallel, Regular Sections** dialog box is displayed.
4. Select the **Attributes** option under the **Element** tab; the **Attributes** option is highlighted.
5. Choose the **Define** button from this dialog box; the **ATTRIBUTES** menu is displayed.
6. Choose **Smooth > Done** from the **ATTRIBUTES** menu.
7. Now, choose the **OK** button; the smooth blend is created, as shown in Figure 8-116.

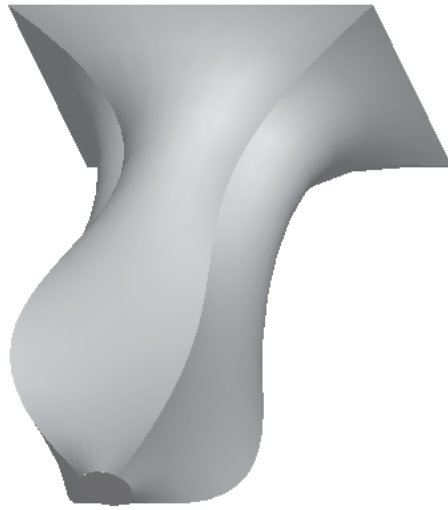


Figure 8-116 Smooth blend feature

Tutorial 4

In this tutorial, you will create the model of a tap shown in Figure 8-117. All dimensions for the three sections are shown in the figure. **(Expected time: 30 min)**

This model is created using the general blend. In general blend, each section should have a coordinate system. The coordinate system helps in the alignment of the sections. Each section will be dimensioned with its coordinate system.

The following steps are required to complete this tutorial:

- Examine the model and determine the number of sections in the blend feature, refer to Figure 8-117.
- Create the first section for the blend feature, refer to Figure 8-118.
- Create the second section for the blend feature, refer to Figure 8-119.
- Create the third section for the blend feature, refer to Figure 8-120.

Starting a New Object File

- Start a new part file and name it as *c08tut4*. The three default datum planes appear in the drawing area if they were not turned off in the previous tutorial.

Invoking the Blend Option

The given model is created using the general type of blend.

- Choose **Insert > Blend > Protrusion** from the menu bar; the **BLEND OPTS** submenu is displayed.

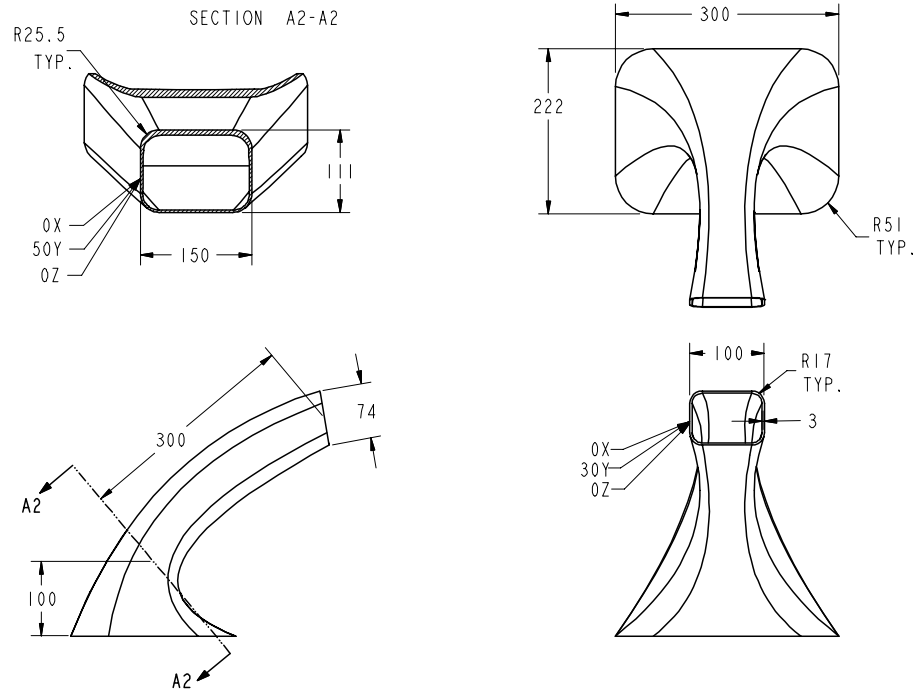


Figure 8-117 Sectioned, left-side, front, and top views of the model

2. Choose **General > Regular Sec > Sketch Sec > Done** from this submenu; the **ATTRIBUTES** menu is displayed.
3. Choose **Smooth > Done**.

You are prompted to select a sketching plane.

4. Select the **TOP** datum plane and choose **Okay** from the **DIRECTION** submenu. From the **SKET VIEW** submenu, choose the **Right** option and select the **RIGHT** datum plane. The system takes you to the sketcher environment.

Drawing the Sketch for the First Section of the Blend

The blend consists of three sections. The dimensions for the second section are half of the dimensions of the first section. Similarly, the dimensions for the third section are one-third of the dimensions of the first section.

1. Draw the sketch for the first section.

2. Choose **Sketch > Coordinate System** from the menu bar. A reference coordinate system is attached to the cursor. Place the coordinate system at the intersection of the two planes using the left mouse button. It will be automatically aligned with the two datum planes.
3. Apply constraints and modify the dimensions of the sketch, as shown in Figure 8-118.
4. Choose the **Done** button; the **Message Input Window** is displayed and you are prompted to enter the x-axis rotation angle for section 2.
5. Enter **0** as the value in this window and press ENTER. You are prompted to enter the y-axis rotation angle for section 2. Enter **50** as the value and press ENTER. Now, you are prompted to enter the z-axis rotation angle for section 2. Enter **0** as the value and press ENTER.

The system takes you to the sketcher environment and allows you to draw the sketch for the second section. Here, you will notice that the datum planes are not displayed even if they are turned on. The reason for this is that the datum planes are not required now. The use of datum planes is to reference the sketch you draw. But, the second section that you will draw is already referenced to the first section. Hence, the datum planes are not displayed and you can draw the next section anywhere in the drawing area.

Drawing the Sketch for the Second Section

1. Draw the sketch for the second section, insert a reference coordinate system, apply constraints and dimensions to the sketch, and modify the dimensions, as shown in Figure 8-119.

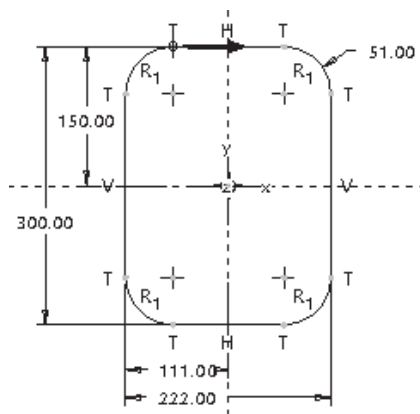


Figure 8-118 Sketch with dimensions and constraints for the first section

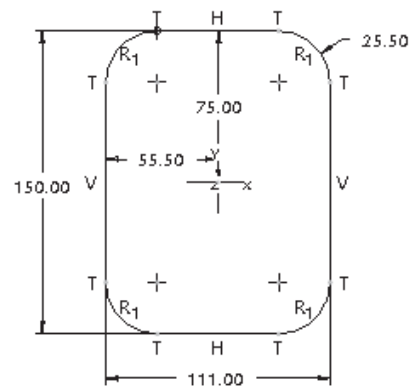


Figure 8-119 Sketch with dimensions and constraints for the second section

- After completing the sketch, choose the **Done** button.

The **Message Input Window** is displayed and you are prompted to specify if you want to continue to the next section.

3. Enter **Y** in this window and press ENTER. You have entered **Yes** in this window because you have to draw the third section of the blend feature.

The **Message Input Window** is displayed and you are prompted to enter the x-axis rotation angle for section 3.

4. Enter **0** as the value in this window and press ENTER. You are prompted to enter the y-axis rotation angle for section 3. Enter **30** as the value and press ENTER. Now, you are prompted to enter the z-axis rotation angle for section 3. Enter **0** as the value and press ENTER.

The system takes you to the sketcher environment and allows you to draw the sketch for the third section.

Drawing the Sketch for the Third Section

1. Draw the sketch for the third section, insert a reference coordinate system, apply constraints and dimension to the sketch, and modify the dimensions, as shown in Figure 8-120.
2. After completing the sketch, choose the **Done** button.

The **Message Input Window** is displayed and you are prompted if you want to continue to the next section.

3. Enter **N** in this window and press ENTER. You have entered **No** in this window because all sections that are needed to create the blend feature are completed.

The **Message Input Window** is displayed and you are prompted to enter the depth for section 2.


Specifying the Depths between Sections

1. Enter a value of **100** in the **Message Input Window** and press ENTER. This depth is the distance between the first section and the second section. Now, you are prompted to enter the depth for **section 3**.
2. Enter a value of **300** in this window and press ENTER. This depth is the distance between the second section and the third section.

The blend feature is complete and you can now preview it.

3. Choose the **Preview** button from the **PROTRUSION: Blend, Parallel, Regular Sections** dialog box; the preview of the model is displayed. Now, choose **OK**; the default trimetric view of the blend feature is created, as shown in Figure 8-121.

Creating Shell

1. Choose the **Shell** button from the **Engineering Features** toolbar; the **Shell** dashboard is displayed and you are prompted to select the surfaces to be removed from the part. 
2. Select the one end surface of the swept feature using the left mouse button and then

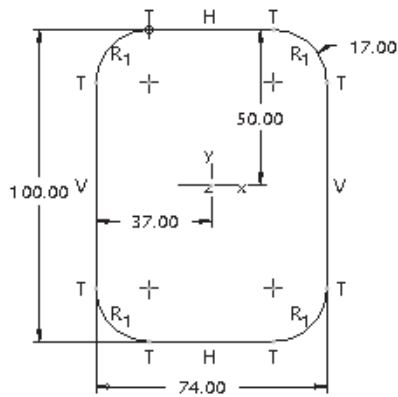


Figure 8-120 Sketch with dimensions and constraints for the third section

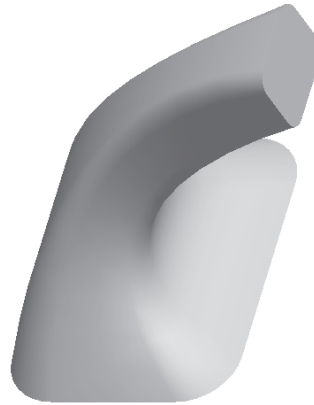


Figure 8-121 Default trimetric view of the blend feature

using CTRL+left mouse button, select the other end surface. The two surfaces selected are highlighted red in color.

3. Enter the thickness value of the shell as **3** in the dimension edit box present on the **Shell** dashboard and press ENTER.
4. Choose the **Build feature** button to exit the **Shell** dashboard.

The default trimetric view of the shell feature is shown in Figure 8-122. You can use the middle mouse button to change the orientation of the model.

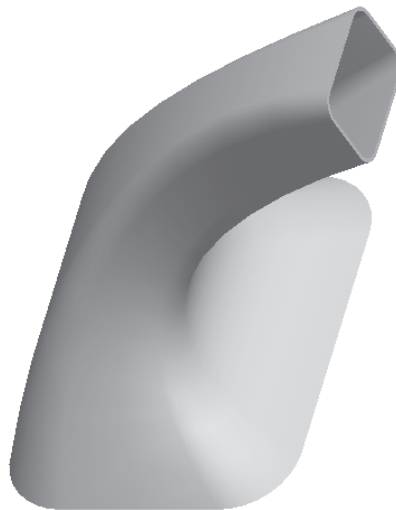


Figure 8-122 Default trimetric view of the model for Tutorial 4

Saving the Model

1. Choose the **Save** button from the **File** toolbar and save the model.

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. The types of Protrusion available in Pro/ENGINEER are different from those that are available for Cut. (T/F)
2. The **Cut** option is available only when at least a base feature exists. (T/F)
3. If the shell thickness value is negative then the shell thickness is added outside the boundary of the face selected for shelling. (T/F)
4. When a trajectory is closed, then there are two options available: **Add Inn Fcs** and **No Inn Fcs**. (T/F)
5. To procedure to create a Sweep Cut feature is the same as of Sweep Protrusion. (T/F)
6. The **Sweep** option extrudes a section along a _____.
7. The cross-section of the swept feature remains _____ throughout the sweep.
8. The sketching plane that you select will be _____ to the screen when you draw the trajectory.
9. An _____ section and open trajectory are not possible.
10. A quilt is a _____ feature.

Review Questions

Answer the following questions:

1. What is the maximum permissible angle for the rotation of sections in a **Rotational** blend?
(a) 120 (b) 90
(c) 180 (d) 45
2. What is the maximum possible draft angle that can be applied in Pro/Engineer?
(a) 10-degree (b) 30-degree
(c) 60-degree (d) 90-degree

3. What is the minimum number of sections required for a blend feature?
 - (a) one
 - (b) two
 - (c) three
 - (d) None of the above
4. Can a trajectory of a sweep feature be modified independent of the geometry of the section?
 - (a) No
 - (b) Yes
 - (c) In some cases
 - (d) None of the above
5. In which one of the following types of blend, sections are translated and rotated about the x, y, and z-axes?
 - (a) **Parallel**
 - (b) **Rotational**
 - (c) **General**
 - (d) None of the above
6. You can create a **Cut** feature using the **Sweep** option. (T/F)
7. While creating a **Rotational** or a **General** blend, you need to create a coordinate system. (T/F)
8. In **General** blend, the section is rotated about the y-axis of the coordinate system. (T/F)
9. The **Rotational** blend option is same as the **Parallel** blend option, if the rotational blend angle entered between the two sections equals to 0-degree. (T/F)
10. There must be an equal number of vertices in each section for blending. (T/F)

Exercises

Exercise 1

Create the foundation bolt shown in Figure 8-123. Its shaded model is shown in Figure 8-124.
(Expected time: 30 min)

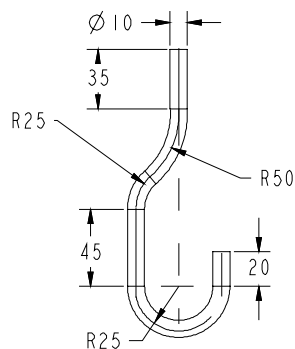


Figure 8-123 Figure for Exercise 1



Figure 8-124 Model for Exercise 1

Exercise 2

In this exercise, you will create the model shown in Figure 8-125. The figure 8-126 shows the sectioned front view, top view, and the right-side view of the solid model with dimensions.

(Expected time: 45 min)

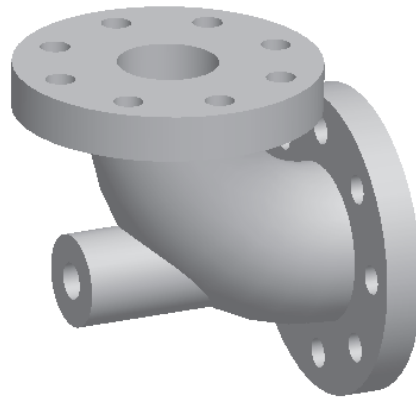


Figure 8-125 Model for Exercise 2

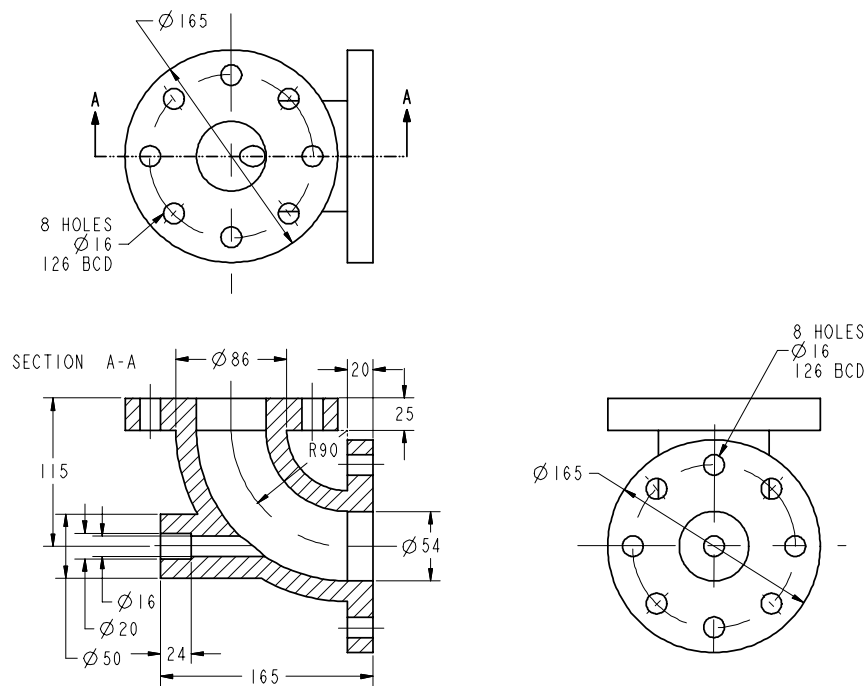


Figure 8-126 Top, front, and right-side views of the model

Exercise 3

In this exercise, you will create the model of a soap case shown in Figure 8-127. Figure 8-128 shows the sectioned front view, top view, right-side view, and the detail view with dimensions.

(Expected time: 45 min)

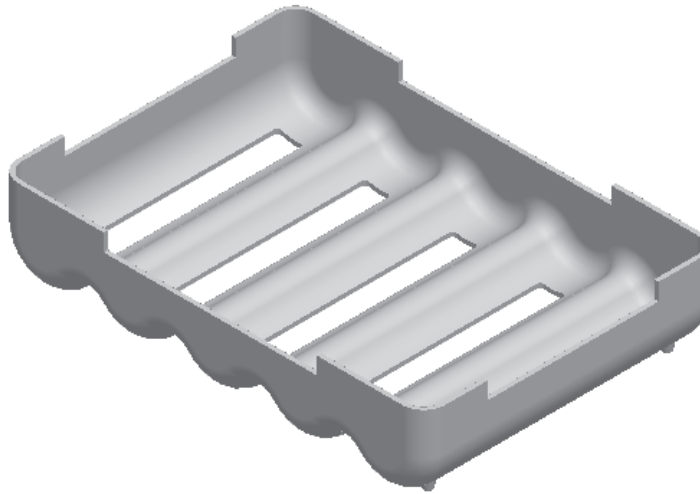


Figure 8-127 Isometric view of the soap case

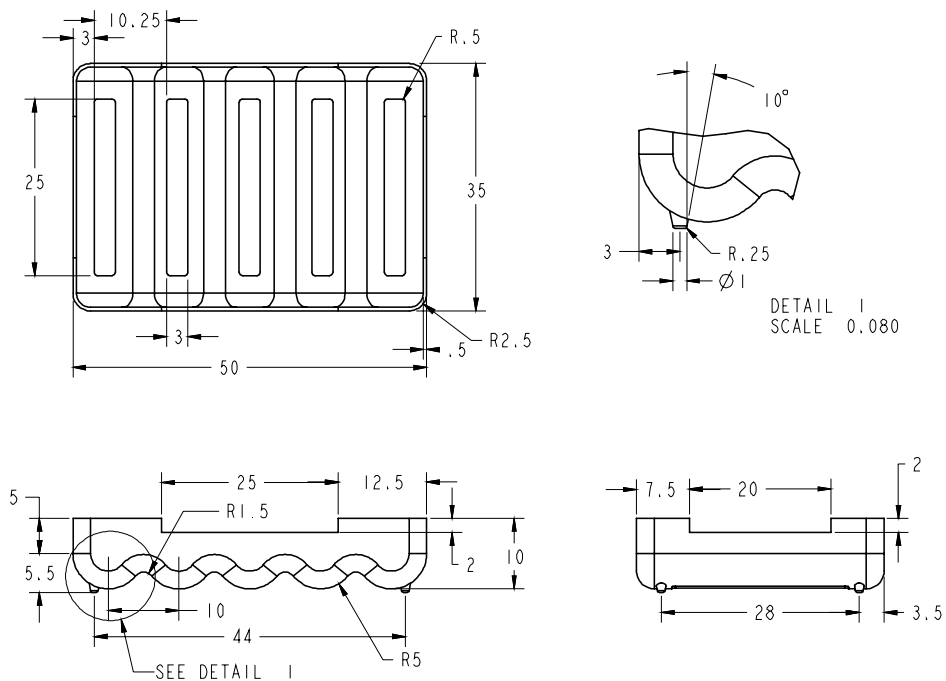


Figure 8-128 Top, sectioned front, right-side, and detail view of the soap case

Exercise 4

In this exercise, you will create the model of a carburetor cover shown in Figure 8-129. Figure 8-130 shows the sectioned top view, sectioned front view, sectioned right-side view, and the sectioned bottom view with dimensions. **(Expected time: 45 min)**

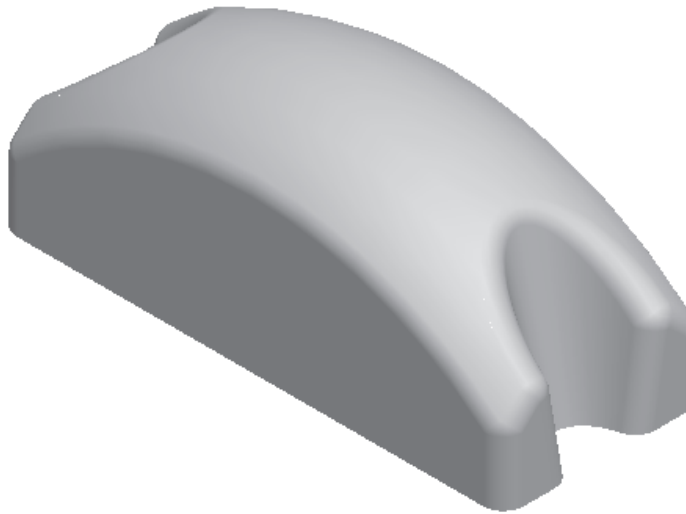


Figure 8-129 Isometric view of the carburetor cover

Hint

1. Create the sketch of the base feature that includes a rectangle of 125x50 and then extrude it.
2. Choose **Insert > Sweep > Cut** from the menu bar.
3. Select the **FRONT** datum plane as the sketching plane for sketching a trajectory.
4. Create a trajectory using the **3-Point / Tangent End** button. The start point and the endpoint of the arc should be at a distance of 28 from the bottom of the base feature and the radius of the arc should be 100.
5. Exit the sketcher environment using the **Done** button.
6. Choose the **Merge Ends** option from the **ATTRIBUTES** menu.
7. You will again enter the sketching environment to create the section for the sweep feature. Choose the **3-Point / Tangent End** button from the **Sketcher Tools** toolbar. The arc created should be tangent to the reference lines, and the endpoints of the arc should be aligned with the edges of the base feature and should have a radius of 35.
8. Choose **OK** from the **CUT: Sweep** dialog box.
9. Choose **Insert > Blend > Cut** from the menu bar.
10. Choose **Done** from the **BLEND OPTS** submenu.
11. Choose **Done** from the **ATTRIBUTES** menu.
12. Select the bottom face of the base feature as the sketching plane.
13. Choose **Okay** from the **DIRECTION** submenu.
14. Choose the **TOP** option from the **SKET VIEW** submenu and select the **RIGHT** datum plane.
15. Create an ellipse of Rx12 and Ry8 using the **Ellipse** button.

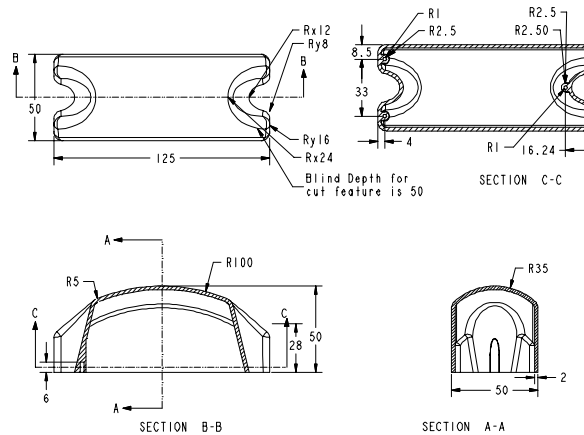


Figure 8-130 Top view, sectioned front view, sectioned right-side view, and sectioned bottom view of the carburetor cover

16. Choose **Sketch > Feature Tools > Toggle Section**.
17. Create another ellipse of Rx24 and Ry16.
18. Exit the sketcher environment using the **Done** button.
19. Choose **Okay** from the **DIRECTION** menu.
20. Choose **Done** from the **DEPTH** menu and enter the depth of cut in the **Message Input Window** and press ENTER.
21. Mirror the cut feature about the **RIGHT** datum plane.
22. Create a round feature on all edges except the edges enclosing the bottom planar surface of the base feature.
23. Invoke the **Shell** option from the menu bar. Remove the bottom face of the base feature.
24. Using the bottom face of the base feature as the sketching plane create the three protrusion feature that are the supporting structures for the screws. Extrude these features using the **Extrude up to next surface** button.
25. Using the **Hole** dashboard, create the hole in the protrusion feature create earlier.

Answers to Self-Evaluation Test

1. F, 2. T, 3. T, 4. T, 5. T, 6. trajectory, 7. constant, 8. parallel, 9. open, 10. surface.