Chapter 7

Advanced Modeling Tools-I

Learning Objectives

After completing this chapter you will be able to:

- *Create sweep feature using the Sweep > Solid and Sweep > Thin options.*
- *Create features using sweep cut.*
- *Create Parallel, Rotational, and General blend.*
- *Use Blend Vertex in blend features.*
- *Create Shell feature.*

OTHER PROTRUSION OPTIONS

The **Extrude** and **Revolve** options available in the **SOLID OPTS** menu were discussed in Chapter 3, 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 the 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 on the graphics screen.

In the next section you will learn about both protrusion and cut sweeps.

SWEEP

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 the 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.

Sweep > Solid

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. In the **Sweep > Solid** option that is discussed here, material defined by the section is added in the specified path. Figure 7-1 shows the **SOLID OPTS** submenu that is displayed when you choose **FEAT > Solid > Protrusion**. You can also choose the **Sweep** option from the menu bar. Choose **Insert** menu from the menu bar; various options are displayed in this menu. Choose **Protrusion > Sweep**. The **SWEEP TRAJ** menu appears. Similarly, when you choose the **Done** option in the **SOLID OPTS** menu, the **SWEEP TRAJ** menu is displayed as shown in Figure 7-2. The options in this menu are discussed next.

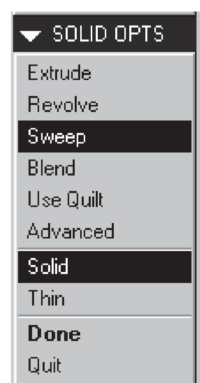


***Figure 7-1 SOLID OPTS** menu*

***Figure 7-2 SWEEP TRAJ** menu*

Sketch Traj

The **Sketch Traj** 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. 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 7-3 shows how the section is swept along the sketched trajectory, and Figure 7-4 shows the shaded image of the swept feature.

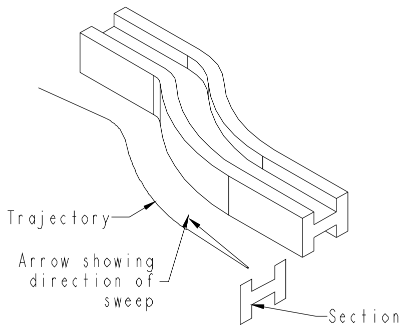


Figure 7-3 *Sweep along the sketched trajectory*

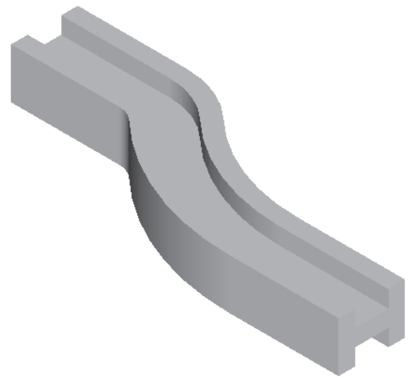


Figure 7-4 *Shaded image*

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 the **ATTRIBUTES** menu is displayed as shown in Figure 7-5. There are two options that are available: **Add Inn Fcs** (Add inner faces) and **No Inn Fcs** (No inner faces).

Figure 7-5 ATTRIBUTES *menu*

For **Add Inn Fcs**, only open sections are possible, as shown in Figure 7-6. The shaded image of the corresponding swept feature is shown in Figure 7-7. The two figures below explain the **Add Inn Fcs** option.

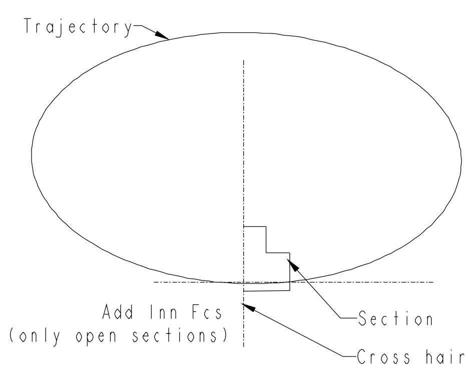


Figure 7-6 Add Inn Fcs *option*

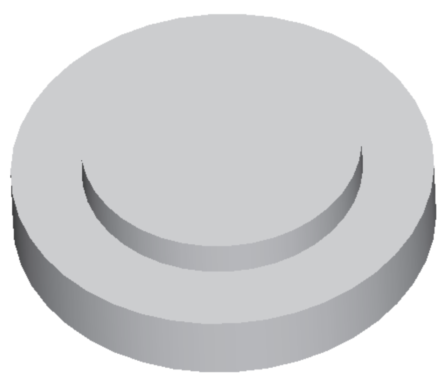


Figure 7-7 *Shaded image*

For **No Inn Fcs**, only closed sections are possible, as shown in Figure 7-8. The shaded image of the corresponding swept feature is shown in Figure 7-9.

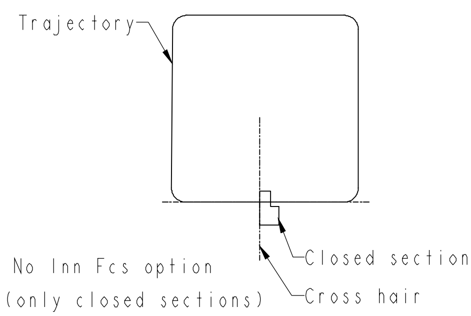


Figure 7-8 ***No Inn Fcs** option*

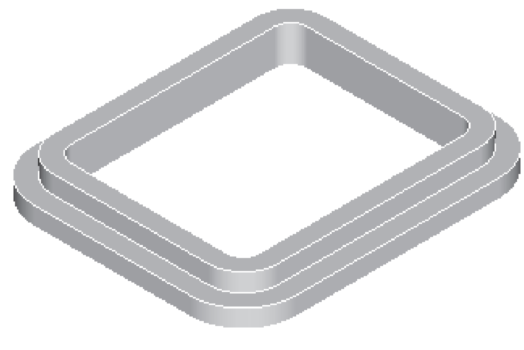


Figure 7-9 *Swept feature*

Select Traj

The **Select Traj** option allows you to select a trajectory on the graphics screen. 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.

Figure 7-10 and Figure 7-11 show two examples of selecting the edges of the base feature and then using these as a trajectory to sweep and their corresponding swept features.

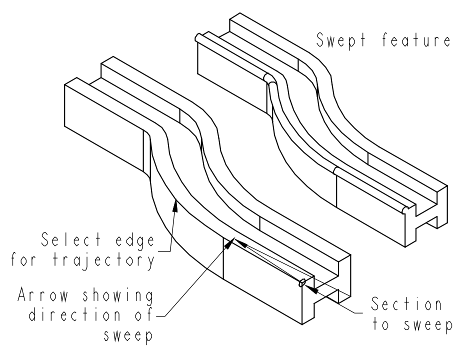


Figure 7-10 *Sweep along the selected trajectory*

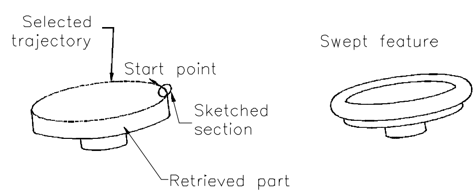


Figure 7-11 *Sweep along the selected trajectory*

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 trajectory with the adjacent geometry and the second option is to leave the ends of the trajectory free. These options are available in the **ATTRIBUTES** menu that is displayed after you complete the sketch of the trajectory and choose the **Continue with the current section** button. The **ATTRIBUTES** menu is shown in Figure 7-12.

***Figure 7-12 ATTRIBUTES** menu*

The **ATTRIBUTES** menu is displayed only when the trajectory is aligned to an edge or surface of the feature that already exists on the graphics screen. 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.

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 7-13 show the **Merge Ends** and **Free Ends** options. In the figure shown below, the trajectory is aligned with the adjacent geometry in both the cases. Figure 7-14 shows the corresponding shaded image.

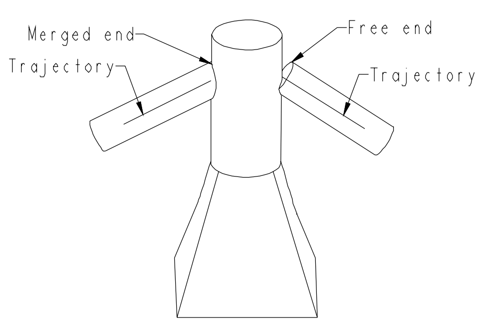


***Figure 7-13 Merge Ends** and **Free Ends** options*

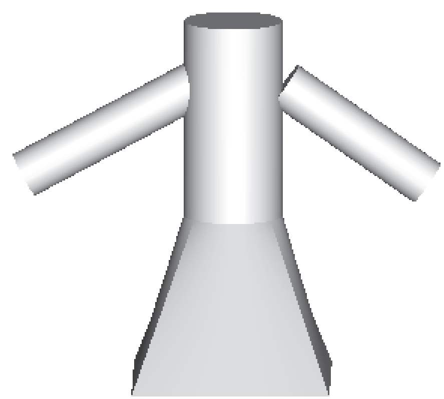
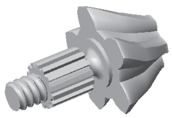


***Figure 7-14** Shaded image*

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 7-15. You can use the **CHAIN** menu only when 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. When you select an edge using the left mouse button, the color of the selected edge changes to blue. Before selecting the edge, make sure that the **Select** option in the **CHAIN** menu is highlighted. The edge once selected and confirmed by choosing **Done Sel** from the **GET SELECT** menu or by using the middle mouse button can also be unselected by selecting the **Unselect** option from the **CHAIN** menu.

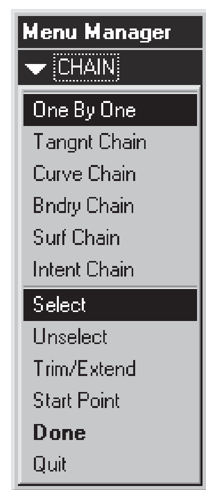


***Figure 7-15 CHAIN** menu*

Tangent Chain

Using this option, you can select an edge or edges tangent to the selected edge. When you select an edge, all the 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 **Tangent Chain** option.

Curve Chain

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

Bndry Chain

The **Bndry 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 7-16.

***Figure 7-16 CHOOSE** menu*

Surf Chain

Figure 7-17 CHAIN OPT menu

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 7-17. Choose either **Select All** or **From-To** from the **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.

Sweep > Thin

The **Sweep > Thin** option creates a thin sweep feature with a specified thickness. This option is similar to the **Extrude > Thin** option that was discussed in Chapter 3. In the case of thin features, a certain thickness has to be specified. The thickness is specified on one sides 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 7-18 shows the sections that can be used to create the model shown in Figure 7-19.

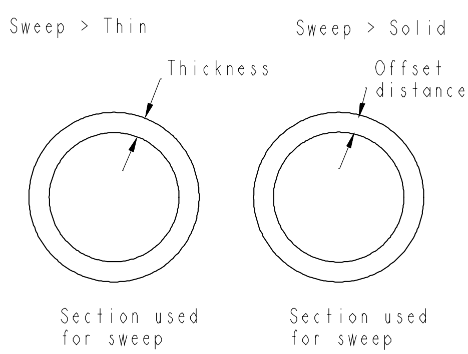


Figure 7-18 Two possible options to create the same model

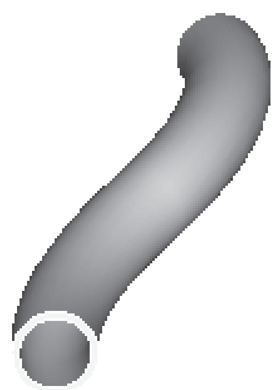


Figure 7-19 Model created using the same sections

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 existing feature. The **Cut** option can be invoked from **PART > Feature > Create > Solid > Cut > Sweep**. Cut can be a solid swept cut or thin swept cut. Figure 7-20 shows trajectories for the **Sweep Cut** feature. Figure 7-21 shows the shaded model of an open trajectory sweep cut and a closed trajectory sweep cut.

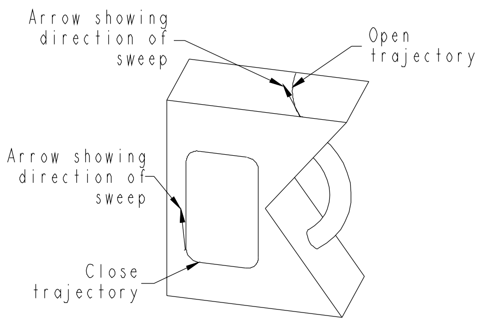


Figure 7-20 *Trajectories for* ***Sweep Cut***

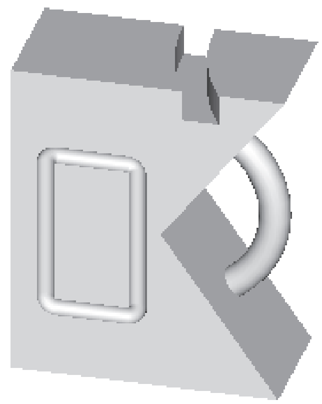


Figure 7-21 *Shaded model with sweep cuts*

BLEND

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 **Blend** is one of the types of **Protrusion** options and **Cut** options that are available in Pro/ENGINEER. The **Blend** option is used where the feature to be created has varying cross sections. It can be invoked from **PART** > **Feature** > **Create** > **Solid** > **Protrusion** or **Cut** > **Blend** > **Solid** or **Thin**. You can also invoke the **Blend** option from the menu bar. Choose **Insert** > **Protrusion** or **Cut** > **Blend**.

Figure 7-22 BLEND OPTS *submenu*

When you choose the **Blend** option, the **BLEND OPTS** submenu is displayed as shown in Figure 7-22. The options in this menu are discussed next.

Parallel

Parallel blends have sections that are drawn parallel to each other and a distance is defined between the parallel sections.

After choosing **Parallel** > **Regular Sec** > **Done** from the **BLEND OPTS** submenu, the **ATTRIBUTES** menu is displayed as shown in Figure 7-23. The options in this menu are discussed next.

Figure 7-23 ATTRIBUTES *menu*

Straight

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

Smooth

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

Figure 7-24 shows three sections that are used to create the blend feature. Figure 7-25 and Figure 7-26 show the parallel blend features with straight edges and smooth edges respectively created using the sections shown in Figure 7-24.

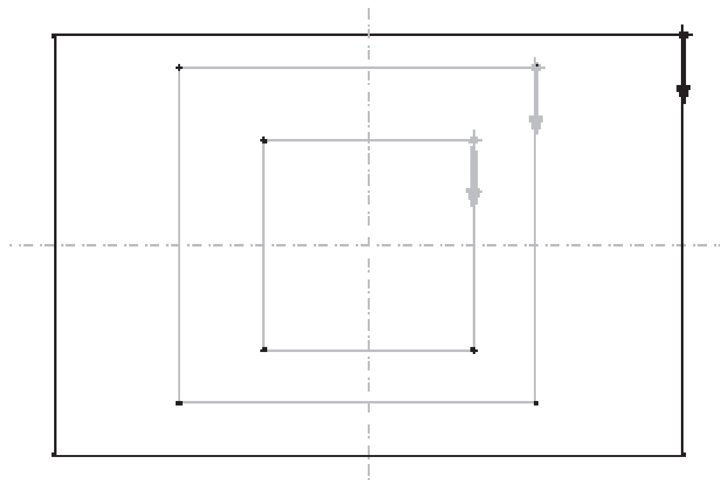


Figure 7-24 *Parallel sections*

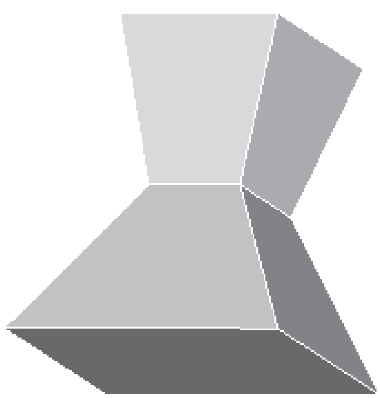


Figure 7-25 *Parallel blend with straight edges*

Figure 7-26 *Parallel blend with smooth edges*

Note

While drawing a section, the start point of the section should be similar in order to avoid twisted blend features.

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

1. *After completing first section, use the* ***Toggle Section*** *option to proceed to 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* ***Continue with the current section*** *button before drawing the second section, the system prompts to use the* ***Toggle Section*** *option to continue with the second section.*

2. *Active section appears in cyan color and the other sections appears in gray.*

3. *All the sections in a blend feature must always 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 the 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* ***Continue the current section*** *button, the sections can be modified by using the* ***Toggle Section*** *option.*

Rotational

Rotational blends have sections that are rotated about the y-axis up to a maximum of 120-degree and the distance between each section 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 **Rotational** blend option functions in the same way as the **Parallel** blend option.

Figure 7-27 *ATTRIBUTES* ***menu***

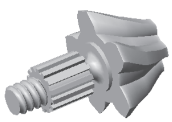
Note that all the nonparallel blends can be open or closed. Therefore, after choosing **Rotational** > **Regular Sec** > **Done** from the **BLEND OPTS** submenu, the **ATTRIBUTES** menu is displayed as shown in Figure 7-27. The options in this menu are discussed next.

Open

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

Closed

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.



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

Figure 7-28 shows the sections used to create a rotational smooth blend feature. The three default datum planes can also be seen. From Figure 7-28, 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 7-29 shows the shaded model of the same blend feature.

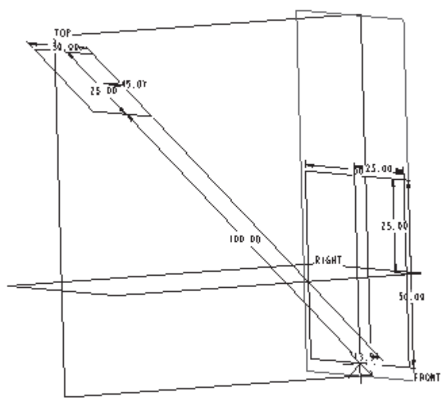


Figure 7-28 The two sections with dimensions and the default datum planes used to create the blend feature shown in the adjacent figure

Figure 7-29 Shaded model of rotational open blend feature

General

Using the **General** option, 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 is if the number of entities in all the sections are not the same. 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 rectangle and the blend vertex in the triangle will be blended with the remain two vertices in the rectangle.

Note
*The **Blend Vertex** can be used only in either 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 from the menu bar by choosing **Insert** > **Shell**. The **Shell** option can also be invoked from the **Menu Manager** by choosing **FEAT** > **Create** > **Solid** > **Shell**. The **Shell** option is used on existing models and hence this option is available only when a model exists on the graphics screen.

***Figure 7-30 FEATURE REFS** menu*

When you choose the **Shell** option, the **FEATURE REFS** menu shown in Figure 7-30 is displayed. The options in this menu are discussed next.

Add

The **Add** option is chosen to select the faces to be removed from the model. The selected face will be removed from the model, leaving the specified thickness from the boundary of the selected surface as shown in Figure 7-31 and Figure 7-32.

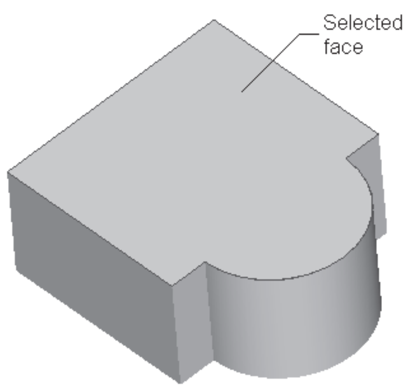


***Figure 7-31** Highlighted surface to shell*

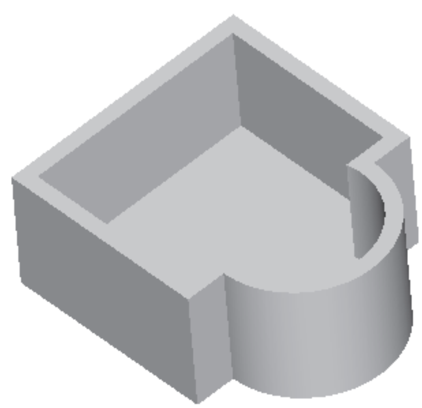


***Figure 7-32** Shell created on the selected surface*

Remove

The **Remove** option is available only when you have selected the faces using the **Add** option and chosen **Done Sel** from the **GET SELECT** submenu. This option is chosen to restore the faces that were selected to be excluded from the model with the **Add** option.

Remove All

The **Remove All** option is used to remove all the faces that were selected using the **Add** option. The **Remove** and **Remove All** options are generally used when the selection of faces has to be changed and these options are available only when the faces are already selected using the **Add** option.

Done Refs

The **Done Refs** option is used to confirm the selection after the selection of all the faces is completed.

Quit Refs

The **Quit Refs** option is used to abort the selection procedure.

Note

*When the system prompts you to enter the shell thickness value, the **Message Input Window** appears. 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 face selected. But when the value entered is negative, the shell thickness is added outside the boundary of the face selected.*

Tip: *You can also create a variable thickness shell. To create a shell with variable thickness, follow the step given below:*

1. *Choose the **Shell** option from the **Insert** menu in the menu bar. The **FEATURE REFS** menu is displayed and you are prompted to select a face to remove.*

2. *Using the left mouse button select the face to remove, the selected face will turn red. Choose **Done Sel** from the **GET SELECT** submenu and choose **Done Refs** from the **FEATURE REFS** menu. The **Message Input Window** is displayed and you are prompted to enter the thickness, enter a value that will define the wall thickness and press ENTER.*

3. *Now, choose the **Spec Thick** option from the **SHELL** dialog box and choose the **Define** button. The **SPEC THICK** menu is displayed. In this menu the **Set Thickness** option is selected by default and you are prompted to select a surface to specify different thickness.*

4. *Using the left mouse button select the surface to specify different thickness and enter the value of the different thickness in the **Message Input Window** and press ENTER.*

5. *Repeat the step 4 to specify the different thickness value to the other surfaces. After specifying the different thickness to all the surfaces choose **Done** from the **SPEC THICK** menu and choose **OK** from the **SHELL** dialog box.*

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**. The option to create a datum curve can be invoked from the menu bar by choosing **Insert > Datum > Curve**. You can also use the **Insert a datum curve.** button from the **Right Toolchest** to invoke the options for creating a datum curve.

When you choose the **Insert a datum curve.** button from the **Right Toolchest**, the **CRV OPTIONS** submenu is displayed as shown in Figure 7-33. Some of the options available in this submenu are discussed next.

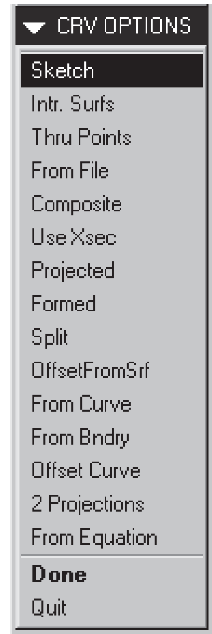


***Figure 7-33 CRV OPTIONS** submenu*

Sketch

The **Sketch** option is used to sketch a datum curve using the sketcher tools. The sketch can be open or closed.

Intr. Surfs

The **Intr. Surfs** (Intersection of Surfaces) 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.

Thru Points

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

From File

The **From File** option is used to import a datum curve from IGES, VDA, *.ibl file formats.

Projected

The **Projected** 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 length of the original entity may distort while projecting.

Formed

This option is used to create a datum curve by wrapping a sketched entity around a solid or a quilt.

TUTORIALS

Tutorial 1

In this tutorial you will create the model shown in Figure 7-34. This figure also shows the sectioned front view, top view, and the right side view of the solid model with dimensions.

(Expected time: 40 min)

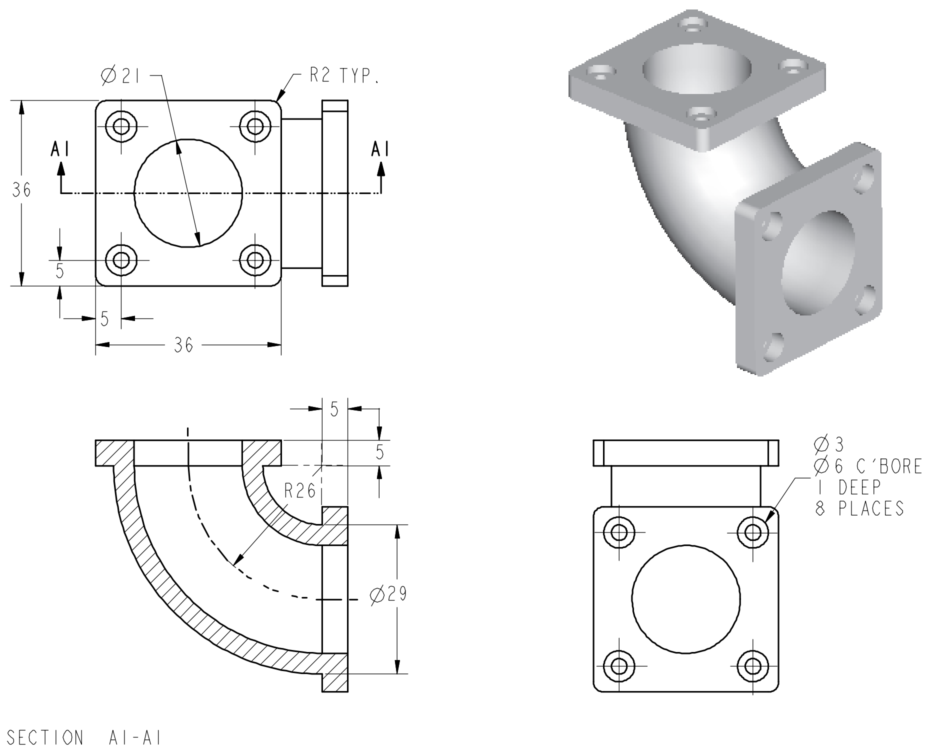


Figure 7-34 *Top, front, right-side, and isometric views of the model. The hidden lines are suppressed for clarity*

The following steps outline the procedure for creating this model:

a. First, examine the model and determine the number of features in it. The model is composed of six features, see Figure 7-34.

b. The base feature is a sweep feature, see Figure 7-37. Select the sketching plane for the base feature, draw the sketch using the sketcher tools, and apply constraints and dimensions.

c. The second feature is a shell of given thickness. Both the end surfaces of the sweep feature will be removed during shelling, see Figure 7-38.

d. The third and the fourth features are extruded features and will be created on the two ends of the swept feature respectively. These features have different sketching planes and hence will be created as separate features. Select the sketching plane, draw the sketch using the sketcher tools, apply constraints and dimensions, and then extrude the sketch to the given distance, see Figure 7-40.

e. The fifth feature is a counterbore hole that will be created on the third feature, see Figure 7-42. This hole will be created using the **HOLE** dialog box.

f. The sixth feature is a counterbore hole that will be created on the fourth feature, see Figure 7-42. This hole will be created using the **HOLE** dialog box. Then, the two counterbore holes will be patterned on the third and the fourth feature, see Figure 7-43.

After understanding the procedure for creating the model, you are now ready to create it. When 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 **c07** if it does not exist. Choose the **New Directory** button in the **Select Working Directory** dialog box and create a directory named **c07** at **C:\ProE**.

Creating New Object File

1. Open a new part file and name it as **c07tut1**.

   The three default datum planes are displayed on the graphics screen. The **Model Tree** also appears on the graphics screen. Exit the **Model Tree** by choosing the **Model Tree on/off** button from the **Model Display** toolbar.

Invoking the Sweep Option

There are two methods to invoke the **Sweep** option. The first method is to use the menu bar present on the top of the screen and the second method is to use the **Menu Manager** present on the right side of the graphics screen.

1. Choose **Insert > Protrusion > Sweep** from the menu bar or choose **PART > Feature > Create > Solid > Protrusion > Sweep > Solid > Done** from the **Menu Manager**. The **SWEEP TRAJ** menu is 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 **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 submenu and select the **TOP** datum plane.

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

Specifying References

The **References** dialog box is displayed on the top right corner of the screen and the status displayed in the **Reference status** area of the **References** dialog box is **Fully Placed**. This means that the references required are already defined and you can now start drawing the sketch. Exit the **References** dialog box by choosing the **Close** button from the dialog box.

Drawing the Trajectory

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

1. Choose the **Create an arc by picking its center and endpoints.** button from the **Sketcher Tools** toolbar. This button is available on the flyout that is displayed when you choose the black arrow that is on the right side of the **Create an arc by 3 points or tangent to an entity at its endpoint.** button.

2. Draw an arc such that the center of the arc lies at the intersection of the two perpendicular planes namely **TOP** and **RIGHT** as shown in Figure 7-35. 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.

Modifying the Dimensions of the Trajectory

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

1. Using the left mouse button double-click on the dimension and modify the radial dimension to 26 as shown in Figure 7-36. You will notice the sketch refits on the screen.

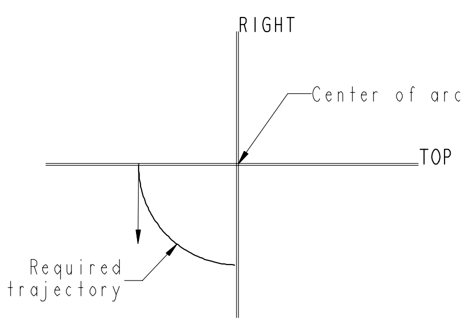


Figure 7-35 The sketch of required trajectory

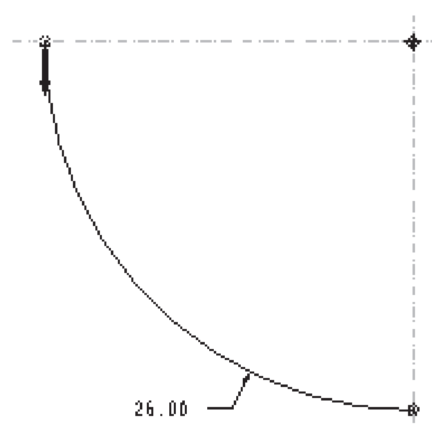


Figure 7-36 Dimension for the arc

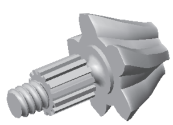
2. Choose the **Continue with the current section.** button.

Drawing the Section for Sweep

After choosing the **Continue with the current section** button, the direction of viewing is modified such that the start point of the trajectory becomes normal to the screen. A blue 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. The **References** dialog box is also displayed. The status displayed in the **Reference status** area of the **References** dialog box is **Fully Placed**. Close the **References** dialog box. You are also prompted to draw the cross-section for the sweep.

1. Choose the **Create circle by picking the center and a point on the circle.** button and create a circle such that the center of 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. The circle will be dimensioned automatically because the **Intent Manager** is on.



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 blue colored cross is available that determines the orientation of the section with respect to the trajectory.*

Modifying the Dimensions of the Section

1. Choose the **Select one item at a time - shift to gather more than one item.** button and using the left mouse button double-click on the dimension and modify the diameter dimension to 21.

The sketch accepts the value and refits on the screen.

2. Choose the **Continue with the current section.** button.

Previewing the Swept Feature

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

1. Choose the **Preview** button from the **PROTRUSION** dialog box that is present on the top right corner of the window.

2. Choose the **Saved view list** button from the **View** toolbar. From the drop-down list choose the **Default** option. The model orients on the graphics screen as shown in Figure 7-37.

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

3. Now, choose the **OK** button from the **PROTRUSION** 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 shell. The swept feature is shelled and the shell thickness lies outside the section. That is, the shell thickness is -4.

1. Choose **Insert > Shell** from the menu bar. The **FEATURE REFS** menu is displayed and you are prompted to select one or more surfaces to remove.

2. Select the two end surfaces of the swept feature using the left mouse button. The edges of the two surfaces selected are highlighted red in color.

3. Choose **Done Sel** from the **GET SELECT** submenu to accept the selection made on the feature. Now, choose **Done Refs** from the **FEATURE REFS** menu.

4. The **Message Input Window** appears and you are prompted to enter the thickness value. Enter **-4** in this window and press ENTER.

   The shell feature is created and you can now preview it.

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** option can be used to obtain the same hollow feature. However, in this tutorial you will use the **Shell** option.*

Previewing the Shell Feature

1. Choose the **Preview** button from the **SHELL** dialog box.

   The default trimetric view of the shell feature is shown in Figure 7-38. You can use the CTRL+middle mouse button to change the orientation of the model.

2. Now, choose the **OK** button from the **SHELL** dialog box to exit it.

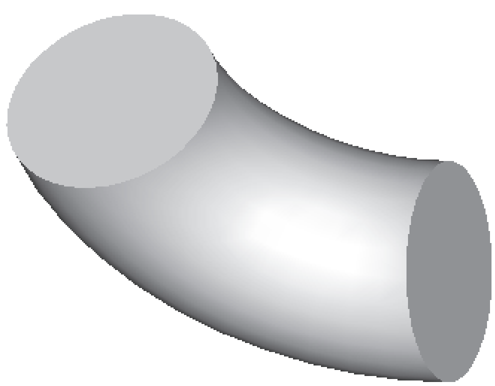


Figure 7-37 *Sweep feature without datum planes*

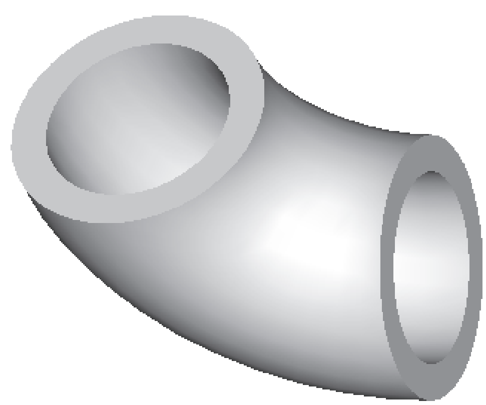


Figure 7-38 *Shell feature without datum planes*

Creating the Extrude Features

The next feature is an extrude feature having a depth of extrusion of 5 that 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 **Insert** > **Protrusion** > **Extrude**. The **ATTRIBUTES** menu is displayed.

2. Choose **One Side** > **Done**. Select the top surface of the swept feature as the sketching plane and choose **Okay**.

3. Choose the **Right** option from the **SKET VIEW** submenu and select the **RIGHT** datum plane.

4. The **References** dialog box is displayed with the status **Not Placed**. Select the **FRONT** and **RIGHT** datum planes. Now, note that the status displayed under the **Reference Status** area in the **References** dialog box is **Fully Placed**.

5. Draw the sketch of the extrude feature and apply constraints and dimensions as shown in Figure 7-39.

6. Create the extruded feature having a depth of 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 extruded features created at both the ends of the swept feature are shown in Figure 7-40.

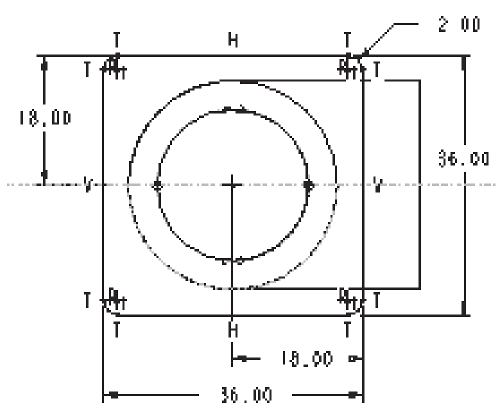


Figure 7-39 *Sketch with dimensions and constraints*

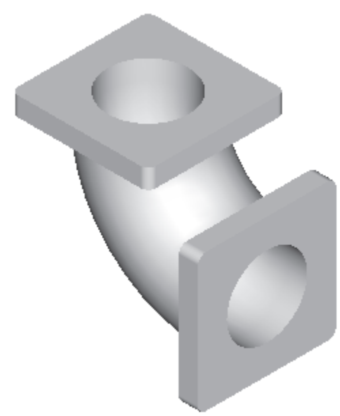


Figure 7-40 *The two extruded features on the ends of the sweep feature*

Creating the Hole Feature

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

1. Choose **Insert** > **Hole** from the menu bar. The **HOLE** dialog box is displayed.

2. Choose the **Sketched** radio button from the **Hole Type** area in the **HOLE** dialog box. The system takes you to the sketcher environment; sketch the section for the counterbore hole as shown in Figure 7-41. Sketch a center line, apply constraints, and diametrically dimension the entities.

3. After completing the sketch, choose the **Continue with the current section.** button. The system exits the sketcher environment and the **Hole** dialog box is redisplayed. Now, you are prompted to select the references for the hole placement.

4. Choose the top planar surface of the third extruded feature for the placement of hole. You are prompted to select the linear references.

5. Select the top edge and the side edge of the third feature for specifying the linear references. The hole is at a distance of 5 from both the edges.

6. Enter the value of **5** in both the **Distance** edit boxes.

7. Choose the **Build feature and repeat the same feature type creation.** button from the **HOLE** dialog box. The hole is created on the selected surface and the **HOLE** dialog box is displayed again.

8. 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 7-42.

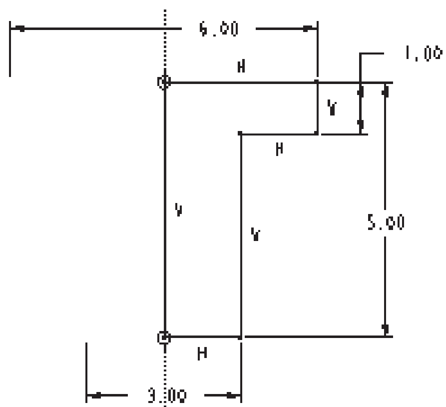


Figure 7-41 Sketch with dimensions and constraints for the counterbore hole

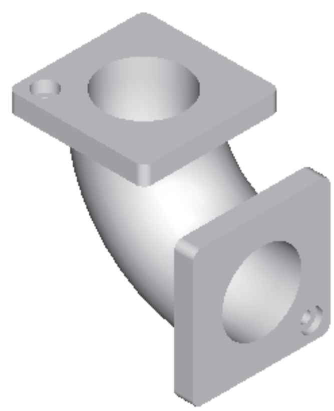


Figure 7-42 One hole each on the two extruded features

Patterning the Hole

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

1. Choose **PART** > **Feature** > **Pattern**. Create pattern of the sketched hole using **Identical** type of pattern.

Similarly, create pattern of the hole on the other extruded feature also. The default trimetric view of the complete model is shown in Figure 7-43.

Saving the Model

1. Choose the **Save the active object** button from the **File** toolbar and save the model. The order of feature creation can be seen from the **Model Tree** shown in Figure 7-44. 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.

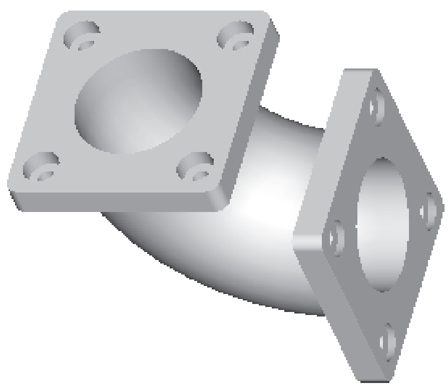


Figure 7-43 *The default trimetric view of the model*

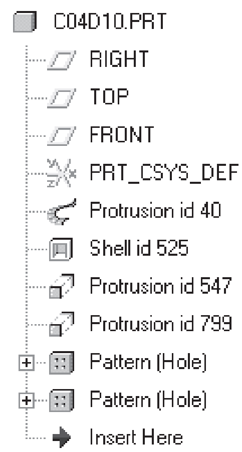


Figure 7-44 Model Tree *for Tutorial 1*

Tutorial 2

In this tutorial you will create the model shown in Figure 7-45. This figure also shows the front view, top view, and the right-side view of the solid model with dimensions.

(Expected time: 30 min)

The following steps outline the procedure for creating this model:

a. First, examine the model and determine the number of features in it. The model is composed of three features, see Figure 7-45.

b. The base feature is a sweep, see Figure 7-48. Select the sketching plane for the base feature, draw the trajectory, apply dimensions and constraints, draw the section of the sweep feature, and then apply dimensions and constraints to the section.

c. The second feature is an extrude feature. Create a datum plane that will be used as the sketching plane for this feature, see Figure 7-49. Draw the sketch for this feature, apply dimensions and constraints, and then extrude the sketch upto the outer curved surface of the base feature, see Figure 7-50.

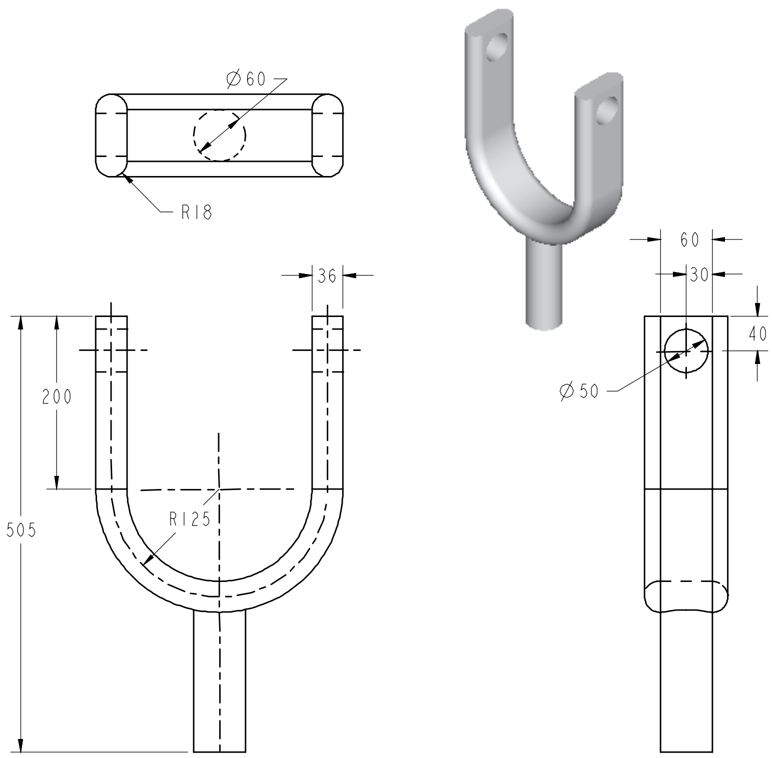


Figure 7-45 *Top, front, right side, and isometric views of the model*

d. The third feature is a hole, see Figure 7-51. The hole will be created on the base feature using the **HOLE** dialog box.

After understanding the procedure for creating the model, you are now ready to create it. The working directory is already selected in Tutorial 1 and therefore you do not need to select it again. However, if you want to change the working directory, choose **File** > **Set Working Directory** and then select c07 in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it as **c07tut2**.

The three default datum planes are displayed on the graphics screen.

Invoking the Sweep Option

As mentioned earlier, there are two ways to invoke the **Sweep** option. One way is to use the menu bar present on the top of the screen and the other way is to use the **Menu Manager** that is present on the right of the graphics screen.

1. Choose **Insert** > **Protrusion** > **Sweep** from the menu bar. The **SWEEP TRAJ** menu is 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

1. Select **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 select the **TOP** datum plane present on the graphics screen.

Specifying References

After you select the planes for orientation, the system takes you to the sketcher environment. The **References** dialog box is displayed at the top right corner of the screen and the status displayed in the **Reference status** area of the **References** dialog box is **Fully Placed**. Exit the **References** dialog box by choosing the **Close** button from the dialog box.

Drawing the Trajectory

The sketch for the trajectory is U-shaped.

1. Draw the sketch for the trajectory and then add constraints and dimensions shown in Figure 7-46.

   Note that an arrow is attached at the start point of the trajectory. This arrow points in the direction of sweep.

2. Choose the **Continue with the current section.** button.

Drawing the Section for Sweep

After choosing the **Continue with the current section** button, the viewing direction is modified such that the start point of the trajectory becomes normal to the screen. A blue color cross of infinite length appears on the screen. You are prompted to draw a section for the sweep. The section of the sweep feature is as shown in Figure 7-47. Draw the section for the trajectory such that the sketch is symmetrical about these lines.

1. Draw the section sketch and then add the constraints and dimensions as shown in Figure 7-47.

2. Choose the **Continue with the current section.** button.

Previewing the Sweep Feature

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

1. Choose the **Preview** button from the **PROTRUSION** dialog box that is displayed on the top right corner of the window.

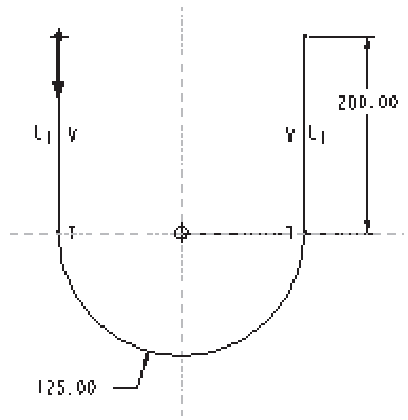


Figure 7-46 Sketch for trajectory of the sweep feature

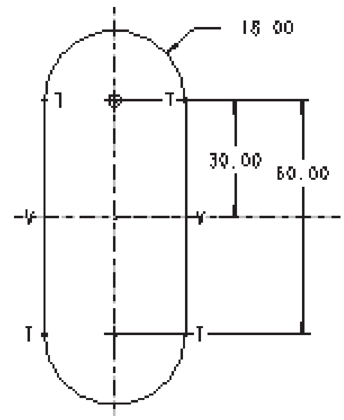


Figure 7-47 Sketch for the section of the sweep feature

2. Choose the **Saved view list** button from the **View** toolbar. From the drop-down list choose the **Default** option. The model orients on the graphics screen as shown in Figure 7-48. You can use the CTRL+middle mouse button to change the orientation of the model.

3. Now, choose the **OK** button from the **PROTRUSION** dialog box.

Creating the Second Feature

To create the extrude feature, you need to create a datum plane that is offset from the top surface of the sweep feature. The offset distance is -505 from the top face of the swept feature.

1. Choose the **Insert a datum plane.** button from the **Right Toolchest to create a datum plane**. The **DATUM PLANE** submenu is displayed. Choose the **Offset** option from this submenu. You are prompted to select a datum plane or a coordinate system .

2. Select the top planar face of the base feature using the left mouse button. The **OFFSET** submenu is displayed.

3. Choose the **Enter Value** option from this submenu. The **Message Input Window** is displayed and you are prompted to enter the offset value in the direction indicated by the arrow. Since the arrow points in the opposite direction, therefore, enter a value of -505 and press ENTER.

4. Choose **Done** from the **DATUM PLANE** submenu. Choose the **Refit the object to fully display it on the screen** button. The datum plane **DTM1** is created as shown in the Figure 7-49.

5. Now, choose **Insert** > **Protrusion** > **Extrude** from the menu bar to invoke the extrude option and select **DTM1** as the sketching plane.

6. Create a circle of 60 diameter as the sketch for the second feature.

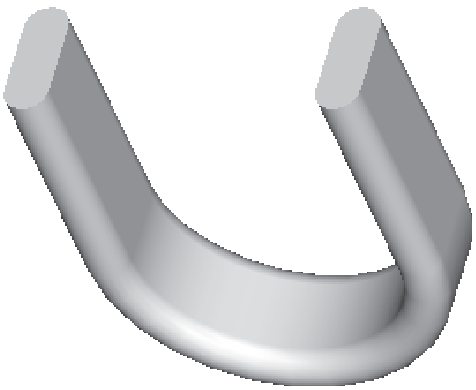


Figure 7-48 The sweep feature

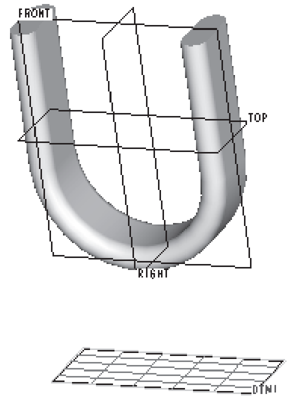


Figure 7-49 Datum plane at an offset distance

7. Choose the **Continue with the current section.** button. The **SPEC TO** menu is displayed.

8. Choose the **UpTo Surface** option from this menu and choose **Done**. You are prompted to select a surface or create a datum plane to extrude up to.

9. Select the outer curved surface of the sweep feature and choose **OK** from the **Feature Creation** dialog box.

   After creating the extrude feature, the model is as shown in Figure 7-50.

Creating the Hole feature

Using the **Hole** dialog box, create the hole. In this tutorial, a through all hole has to be created on the base feature as shown in Figure 7-51.

Figure 7-50 Model after creating the extrude feature

Figure 7-51 Model with the hole feature

Saving the Model

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

   The order of feature creation can be seen from the **Model Tree** shown in Figure 7-52. The feature id numbers displayed in the **Model Tree** may be different when you create the features.

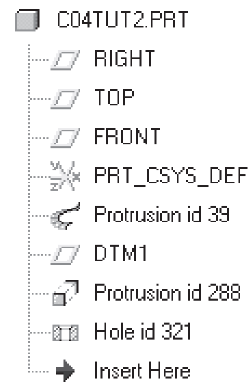


Figure 7-52 ***Model Tree*** *for Tutorial 2*

Tutorial 3

In this tutorial you will create a blend feature shown in Figure 7-54. The two views of the blend feature are shown in Figure 7-53 with dimensions. After creating the model you will redefine such that the straight blending is changed into smooth blending. **(Expected time: 45 min)**

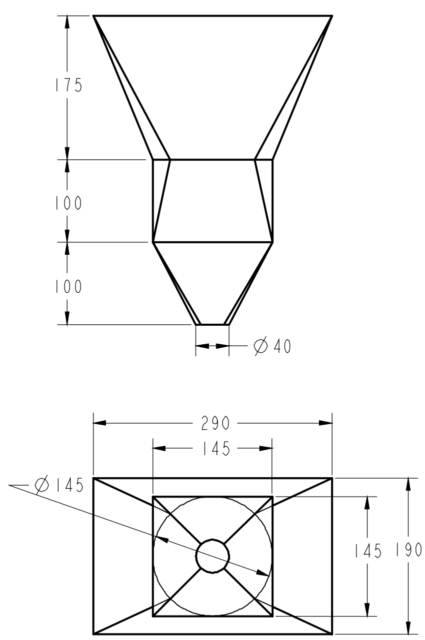


Figure 7-53 *Top and front views of the model*

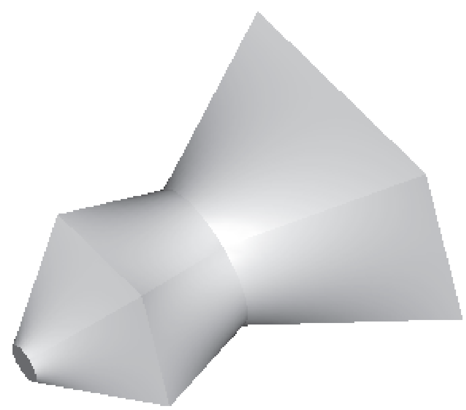


Figure 7-54 *Isometric view of the model*

The following steps outline the procedure for creating this model:

a. First, examine the blend feature and determine the number of sections in this feature. The blend consists of four sections, see Figure 7-53.

b. Select the sketching plane for the blend feature. Draw the first section, apply dimensions and constraints, see Figure 7-55, and then toggle the section to draw the sketch for the second section.

c. Draw the second section, apply the dimensions and constraints, see Figure 7-56 and then toggle the section to draw the third section.

d. Draw the third section, apply the dimensions and constraints, see Figure 7-57 and then toggle the section to draw the fourth section.

e. Draw the fourth section, apply the dimensions and constraints, see Figure 7-57 and then 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.

After understanding the procedure for creating the model, you are now ready to create it.

Creating New Object File

1. Open a new part file and name it as **c07tut3**.

   The three default datum planes are displayed on the graphics screen.

Invoking the Blend Option

There are two methods to invoke the **Blend** option: the first method is to use the menu bar present on the top of the screen and the second method is to use the **Menu Manager** on the right side of the graphics screen.

1. Choose **Insert** > **Protrusion** > **Blend** from the menu bar or choose **PART** > **Feature** > **Create** > **Solid** > **Protrusion** > **Blend** > **Solid** > **Done** from the **Menu Manager**. The **BLEND OPTS** submenu 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 **Straight** or **Smooth** option from this menu.

3. Choose **Straight** > **Done** option from the **ATTRIBUTES** menu.

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

Selecting the Sketching Plane

Select the **FRONT** datum plane as the sketching plane.

1. Select **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 on the graphics screen.

Specifying References

After you select the planes for orientation, the system takes you to the sketcher environment. The **References** dialog box is present at the top right corner of the screen and the status displayed in the **Reference Status** area of the **References** dialog box is **Fully Placed**. Close the **References** dialog box.

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 7-55.

   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.

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 appear gray in color.

Drawing the Next Section

The next section is a circle.

1. Draw the sketch of the circular section, refer to Figure 7-56. Modify the diameter of circle to 145.

   As discussed earlier, the number of entities per section must be equal in a blend feature. A circle is a single entity. Therefore, the circle 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 that is on the right of the **Dynamically trim section entities.** button and then choose the **Divide an entity at the point of selection.** button.

2. Select the circle at four points as shown in Figure 7-56.

   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 **Impose sketcher constraints on the section.** button. The **Constraints** dialog box is displayed.

2. Choose the **Make line or two vertices vertical** constrain 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** constrain 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 7-56. After the circular section is completed, the two sections with dimensions should look similar to the sections shown in Figure 7-56.

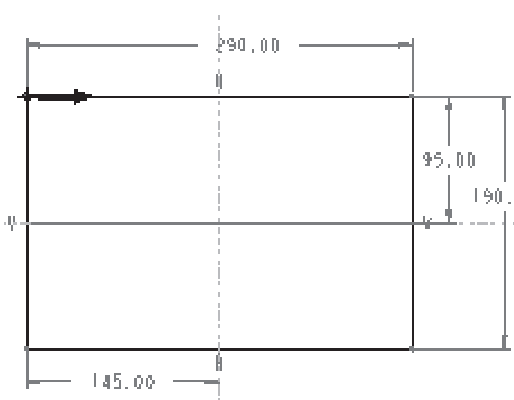


Figure 7-55 *First rectangular section with dimensions*

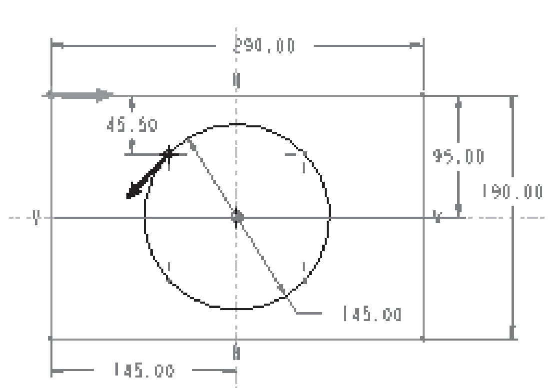


Figure 7-56 *The two completed sections*

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 the section 2 and then constrain and dimension it.

   Figure 7-57 shows all the sections completed with dimensions.

Note

In Figure 7-57, note the direction of the start points shown by arrows. These are important to avoid a twisted feature.

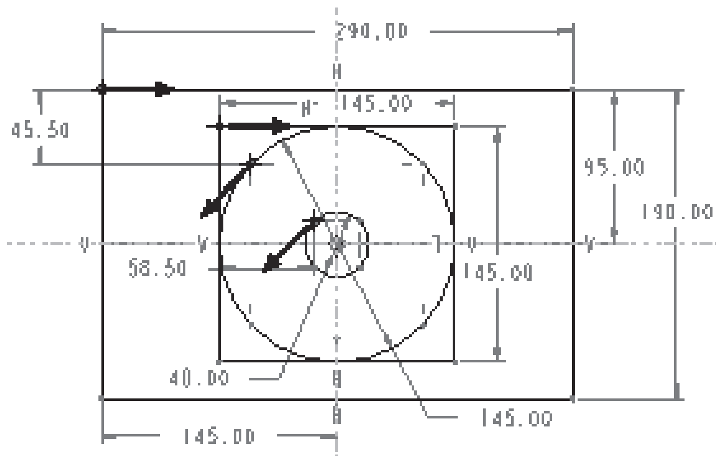


Figure 7-57 *All four completed sections before giving depth*

Giving Depth to the Sections

After the sketches of all the 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 7-53.

1. Choose the **Continue with the current section.** button.

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

2. Enter a value of **175** in the 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** dialog box that is displayed on the

top right corner of the window.

2. Choose the **Saved view list** button from the **View** toolbar. From the drop-down list choose the **Default** option. The model orients on the graphics screen as shown in Figure 7-58. You can use the CTRL+middle mouse button to change the orientation of the model.

3. Now, choose the **OK** button that is present on the **PROTRUSION** dialog box.

Saving the Model

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

   The order of feature creation is evident from the **Model Tree** shown in Figure 7-59. The feature id displayed in the **Model Tree** may be different when you create the blend feature.

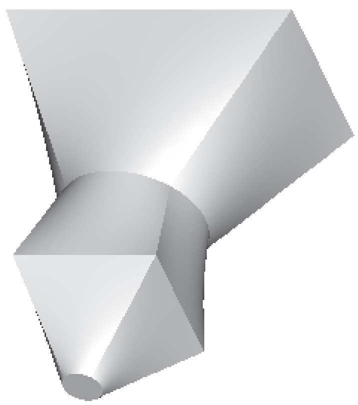


Figure 7-58 *Shaded model*

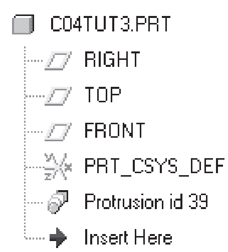


Figure 7-59 Model Tree *for Tutorial 3*

Redefining the Blend Feature

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

1. Select the model on the graphics screen. The edges of the model turn red in color.

2. Press and hold down the right mouse button on the graphics screen until a shortcut menu appears.

3. Choose the **Redefine** option from the shortcut menu. The **PROTRUSION** dialog box is displayed.

4. Select the **Attributes** option under the **Elements** tab. The **Attributes** option is highlighted.

5. Choose the **Define** button from the **PROTRUSION** dialog box. The **ATTRIBUTES** menu is displayed.

6. Choose **Smooth** > **Done** from the **ATTRIBUTES** menu.

7. Choose the **OK** button from the **PROTRUSION** dialog box. The smooth blend is created as shown in Figure 7-60.

Figure 7-60 *Smooth blend feature*

Tutorial 4

In this tutorial you will create the model of a tap shown in Figure 7-61. 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 outline the procedure for creating this model:

a. First examine the model and determine the number of sections in the blend feature. The model consists of three sections, see Figure 7-61. The first section is the bottom of tap, Figure 7-62. The second section, Figure 7-63, is tilted at 50-degree with respect to y-axis and the third section, Figure 7-64, is further located 30-degree with respect to y-axis.

b. Select the sketching plane for the blend feature, draw the first section, insert a coordinate system, apply dimensions and constraints, see Figure 7-62 and then continue to the second section.

c. Draw the second section, insert a coordinate system, apply dimensions and constraints, see Figure 7-63 and then continue to the third section.

d. Draw the third section, insert a coordinate system, apply dimensions and constraints, see Figure 7-64 and then specify the distance between the sections.

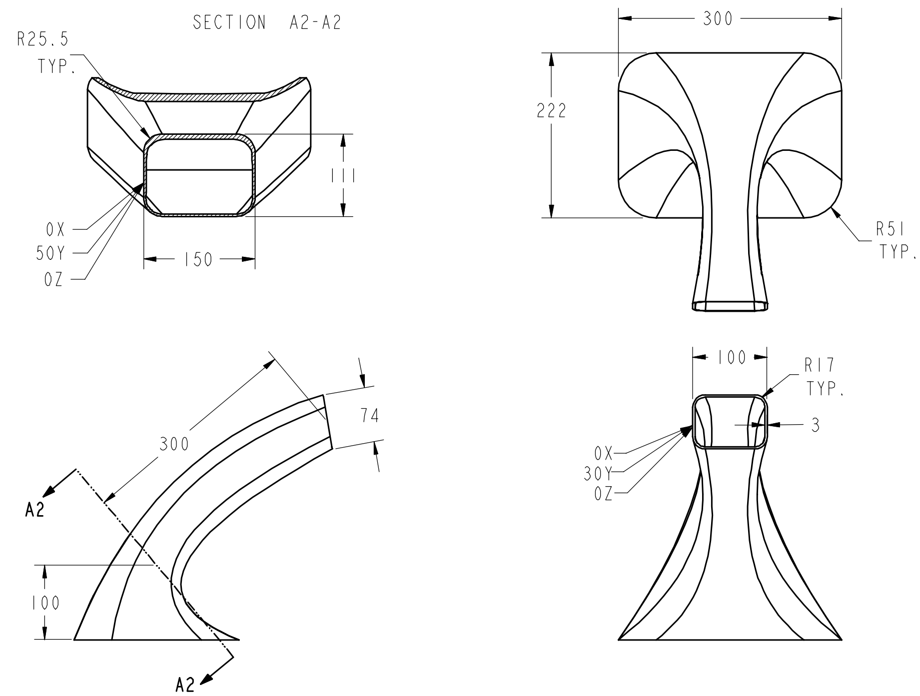


Figure 7-61 *Sectioned view, left-side, front, and top views of the model*

After understanding the procedure for creating the model, you are now ready to create it.

Creating New Object File

1. Open a new part file and name it as **c07tut4**. The three default datum planes appear on the graphics screen 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.

1. Choose **Insert** > **Protrusion** > **Blend**. The **BLEND OPTS** submenu is displayed.

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 7-62.

4. Choose the **Continue with the current section** 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 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 on the graphics screen.

Drawing the Sketch for the Second Section

1. Draw the sketch for the second section, insert a reference coordinate system, apply constraints, and dimension to the sketch and modify the dimensions as shown in Figure 7-63.

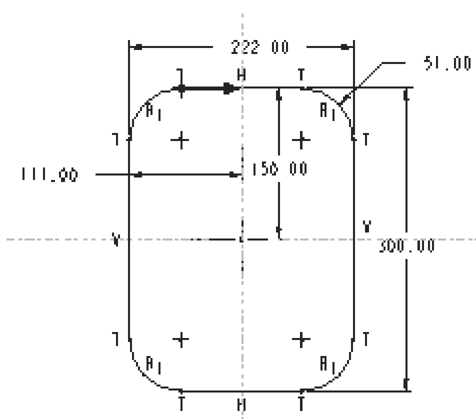


Figure 7-62 *Sketch with dimensions and constraints of section 1*

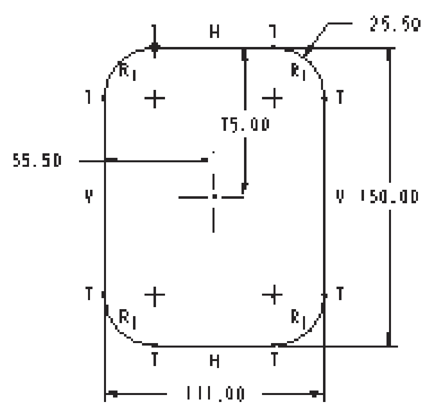


Figure 7-63 *Sketch with dimensions and constraints of section 2*

2. After completing the sketch, choose the **Continue with the current section.** 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 7-64.

2. After completing the sketch, choose the **Continue with the current section.** button.

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

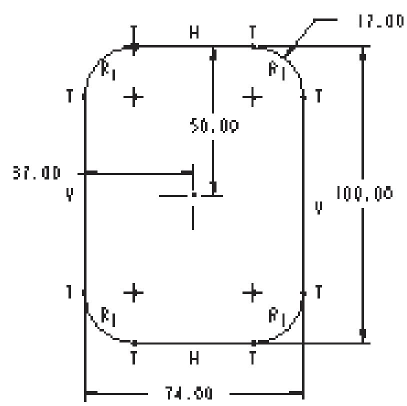


Figure 7-64 *Sketch with dimensions and constraints of section 3*

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

   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 **Preview** from the **PROTRUSION** dialog box and choose **OK**. The default trimetric view of the blend feature is shown in Figure 7-65.

Figure 7-65 *The default trimetric view of the blend feature*

Creating Shell

1. Choose **Insert** > **Shell** from the menu bar. The **FEATURE REFS** menu is displayed and you are prompted to select the surfaces to remove.

2. Select the two end planar surfaces of the blend feature. When you select the surfaces, the edges of the planar surfaces turn red in color.

3. Choose **Done Sel** from the **GET SELECT** submenu or use the middle mouse button to confirm the selection.

4. Choose **Done Refs** from the **FEATURE REFS** menu. The **Message Input Window** is displayed and you are prompted to enter the thickness value for shell.

5. Enter **3** in this window and press ENTER. Select the **Preview** button and then select **OK** from the **SHELL** dialog box. The default trimetric view of the model is shown in Figure 7-66.

Figure 7-66 *The default trimetric view of the model for Tutorial 4*

Saving the Model

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

Self-Evaluation Test

Answer the following questions and then compare your answers to the answers 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. **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 trajectory is closed then there are two options that are available: **Add Inn Fcs** and **No Inn Fcs**. (T/F)

5. To create a Sweep Cut feature the procedure to follow is the same as in the case 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 you select will be __________ to the screen when you draw the trajectory.

9. __________ 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. In which of the following numbered section of a blend feature, **Blend Vertex** option can be used?

   (a) second (b) fourth
   (c) last (d) None of the above

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. In 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 0-degree. (T/F)

10. There must be equal number of vertices in each section for blending. (T/F)

Exercises

Exercise 1

Create the foundation bolt shown in Figure 7-67. The shaded model is shown in Figure 7-68.
(Expected time: 30 min)

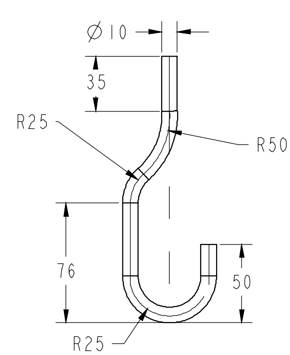


Figure 7-67 *Figure for Exercise 1*

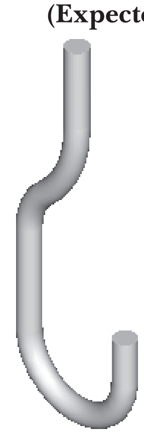


Figure 7-68 *Shaded model*

Exercise 2

In this exercise you will create the model shown in Figure 7-69. This figure also shows the sectioned front view, top view, and the right-side view of the solid model with dimensions.

(Expected time: 40 min)

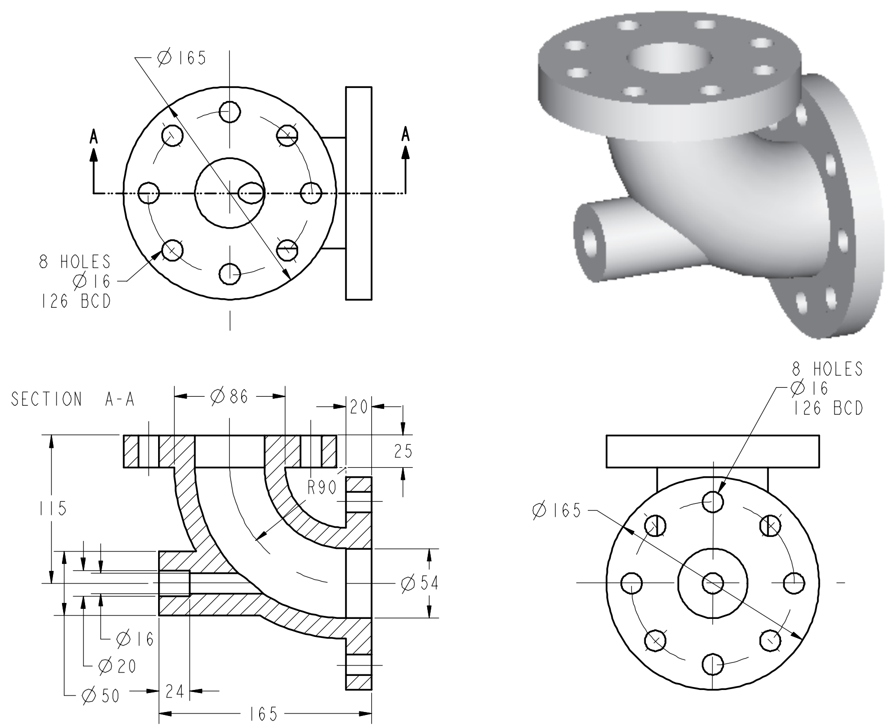


Figure 7-69 *Top, front, right-side, and isometric views of the model*

Exercise 3

In this exercise you will create the model of a soap case shown in Figure 7-70. Figure 7-71 shows the sectioned front view, top view, right-side view, and the detail view with dimensions.

(Expected time: 50 min)

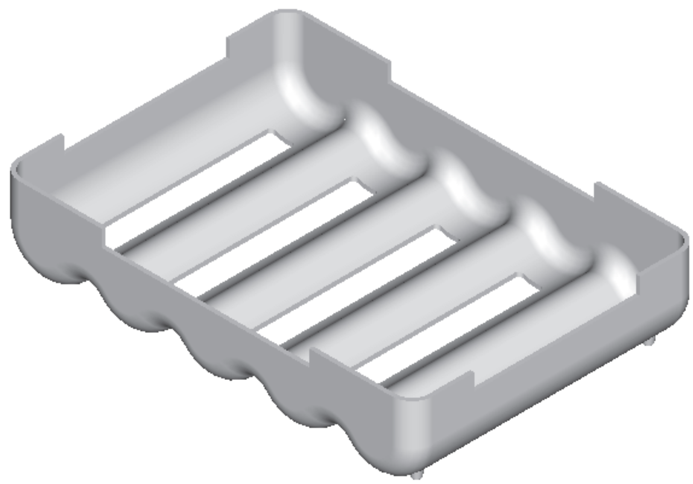


Figure 7-70 Isometric view of the soap case

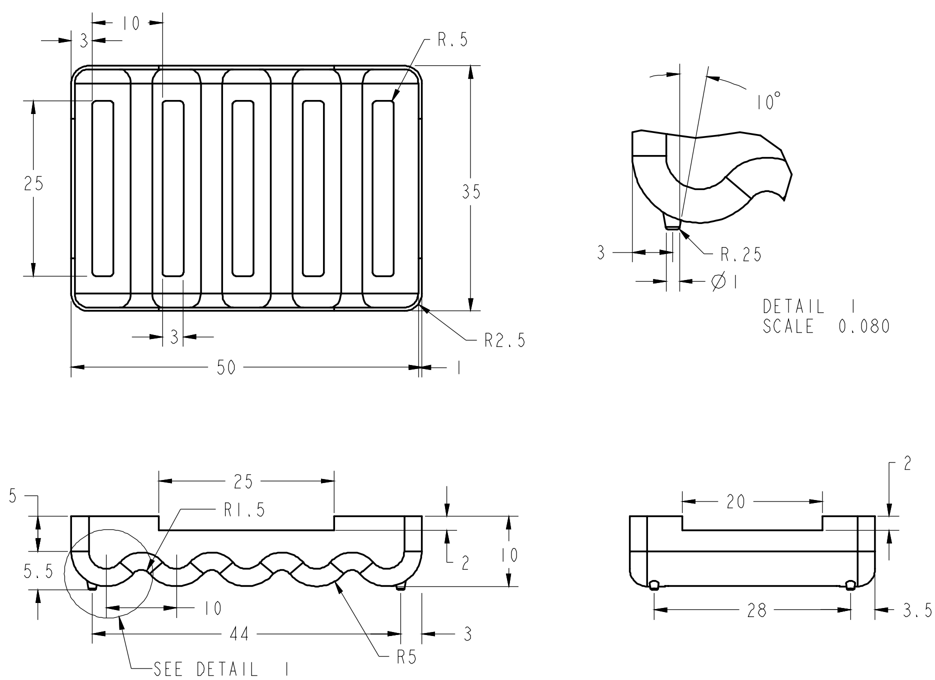


Figure 7-71 Top, sectioned front, right-side, and detail view of the soap case

Exercise 4

In this exercise you will create the model of a carburettor cover shown in Figure 7-72. Figure 7-73 shows the sectioned top view, sectioned front view, sectioned right-side view, and the sectioned bottom view with dimensions. **(Expected time: 50 min)**

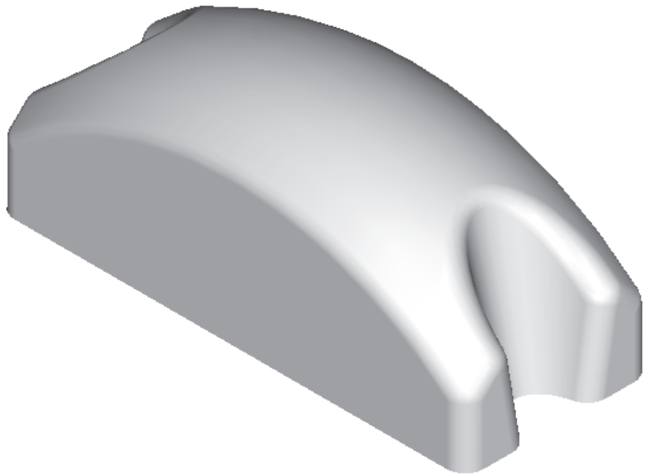


***Figure 7-72** Isometric view of the carburettor cover*

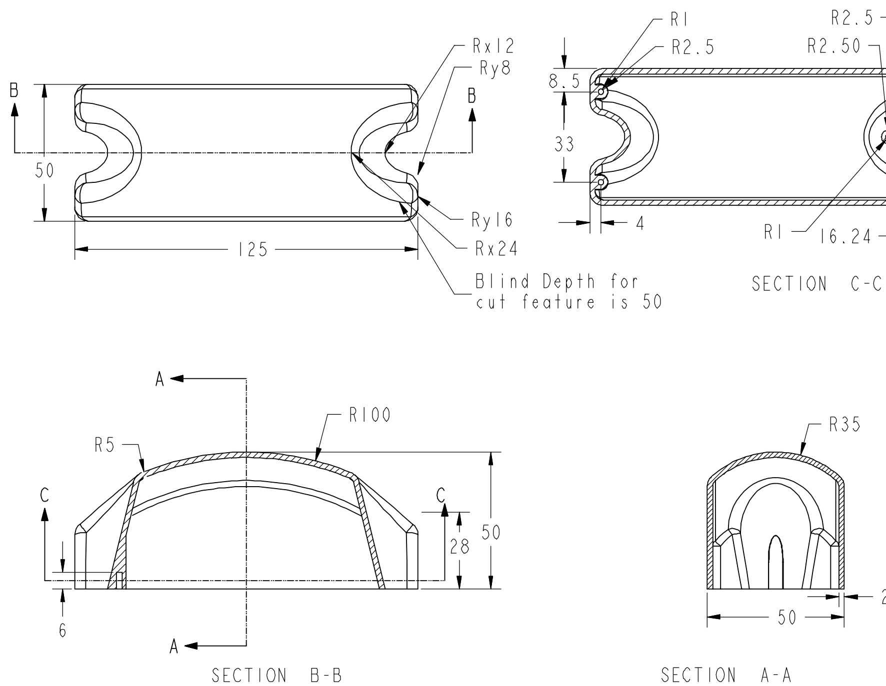


***Figure 7-73** Top view, sectioned front view, sectioned right-side view, and sectioned bottom view of the carburettor cover*

Answers to the Self-Evaluation Test

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