Chapter 8

Advanced Modeling Tools-II

Learning Objectives

After completing this chapter you will be able to:

- *Use different Advanced options for creating complex models.*
- *Create features using Var Sec Swp option.*
- *Create features using Swept Blend option.*
- *Create features using Helical Sweep option.*
- *Create features using Sect to Srfs option.*
- *Create features using Srfs to Srfs option.*
- *Create features using From File option.*
- *Use Relations.*

ADVANCED FEATURE OPTION

Pro/ENGINEER facilitates its users with some advanced options to create complex features with greater ease. All these options are available in the **ADV FEAT OPT** submenu. The **ADV FEAT OPT** submenu is displayed when you choose **PART** > **Feature** > **Create** > **Solid** > **Protrusion** > **Solid** > **Advanced** > **Solid** > **Done** from the **Menu Manager**. The options in the **ADV FEAT OPT** submenu are also displayed when you choose **Insert** > **Protrusion** from the menu bar.

Figure 8-1 shows you the different options available in this submenu.

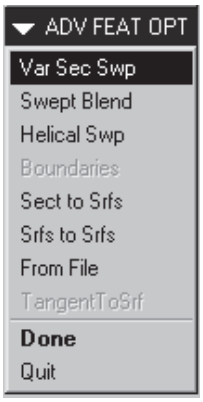


***Figure 8-1 ADV FEAT OPT** menu*

Var Sec Swp Option

The **Var Sec Swp** (Variable Section Sweep) option is used to create a sweep feature in which the section of the sweep feature varies according to the trajectories. This feature is created using more than one trajectory and the section that will be swept along the origin trajectory and the variation of the section is controlled by the X-trajectory, and other trajectories. You can sketch the sweep trajectory using different sketcher tools. You can also select a previously created longitudinal entity as the trajectory for sweep. The **Var Sec Swp** option is available for both protrusion and cut options. Choose the **Var Sec Swp** option from the **ADV FEAT OPT** submenu or choose **Insert** > **Protrusion** > **Variable Section Sweep** from the menu bar; the **SWEEP OPTS** submenu is displayed as shown in Figure 8-2.

***Figure 8-2 SWEEP OPTS** submenu*

NrmToOriginTraj option

The **NrmToOriginTraj** (Normal To Origin Trajectory) option is used to create a variational section sweep feature in which the swept section is normal to the origin trajectory and the variation in the section is controlled by the X-trajectory, and other trajectories. When you use this option, you need to select or sketch an origin trajectory, an X-trajectory, and other trajectories if required.

The following points should be remembered while creating a feature using the **NrmToOriginTraj** option.

1. You should define the origin trajectory that is normal to the section.

2. After creating the origin trajectory you have to create an X-trajectory that defines the horizontal vector of the section.

3. You can also define the additional trajectories to facilitate the creation of a complex profile.

4. When you draw the section for the variational section sweep feature, the section should be aligned to the endpoints of the X-trajectory and the other trajectories. If the section is not aligned then the feature will be created without following the trajectory path and will be similar to a feature created using the **Sweep** option.

When you choose the **Done** option from the **SWEEP OPTS** submenu the **VAR SEC SWP** menu is displayed as shown in Figure 8-3. Using this menu you can select a trajectory or you can sketch a trajectory. The selected or sketched trajectory other than the origin trajectory, and the X-trajectory can be deleted using the **Remove Traj** option from the **VAR SEC SWP** menu.

***Figure 8-3** VAR SEC SWP menu*

Figure 8-4 shows the origin trajectory and the X trajectory. The two trajectories are used to create the sweep feature shown in Figure 8-5. Now, in Figure 8-6, the X trajectory and the origin trajectory are interchanged and the sweep feature that is obtained is shown in Figure 8-7.

Note the difference in the swept features. In Figure 8-5, the section is normal to the selected origin trajectory and when the curve is selected as the origin trajectory, the section is normal to it as shown in Figure 8-7.

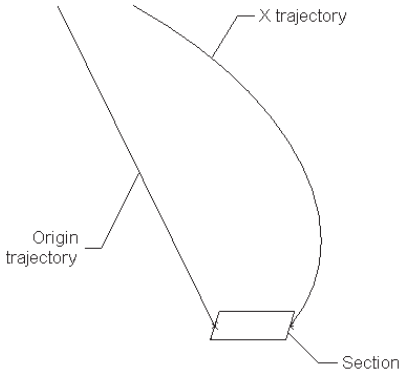


***Figure 8-4** The two trajectories*

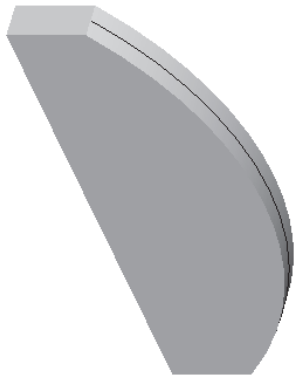


***Figure 8-5** Variable section sweep feature*

Pivot Dir option

The **Pivot Dir** (Pivot Direction) option is used to create a variable section sweep in which the section to be swept is normal to the selected plane, edge, curve, or axis. You are required to select or create a plane, an edge, a curve, or an axis to define the pivot direction, an origin trajectory, a section, and other trajectories if required. Figure 8-8 shows the origin trajectory and the pivot plane to which the section will be perpendicular. Figure 8-9 shows the corresponding swept feature.

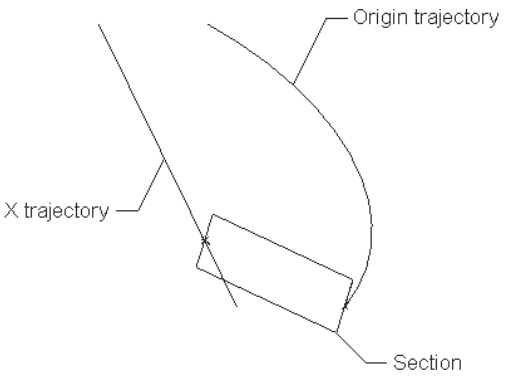


Figure 8-6 *The two trajectories and section*

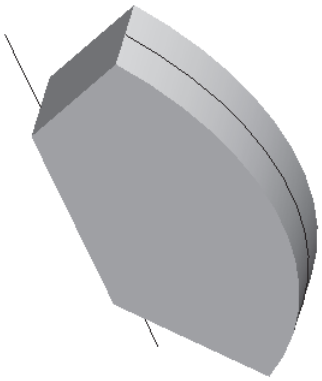


Figure 8-7 *Resultant variable section sweep feature*

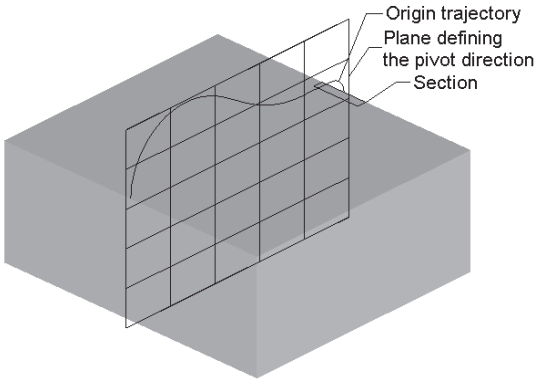


Figure 8-8 *Origin trajectory and the pivot plane*

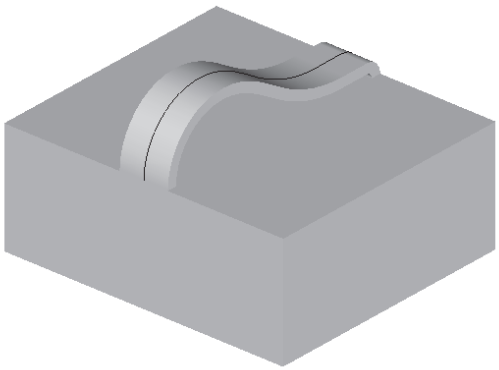


Figure 8-9 *Swept feature on the base feature*

Norm To Traj option

When you use the **Norm To Trj** (Normal To Trajectory) option to create the variable sweep, you have to define an origin trajectory, a normal trajectory, a section, and other trajectories if required. The section is perpendicular to the normal trajectory and is swept along the origin trajectory. Figure 8-10 shows the two trajectories that are used to create the swept feature shown in Figure 8-11.

Swept Blend

The **Swept Blend** feature is a combination of sweep and blend. To create a sweep feature you need a trajectory along which the section is swept. You have an option of sketching or selecting a previously created feature as the trajectory of the sweep. To create a parallel blend you need to have more than one section to be blended. The combination of both these options is the **Swept Blend** option.

In this option you can blend two or more than two sections having same number of entities and the path of the transition of blending is defined by selecting or sketching a trajectory. You

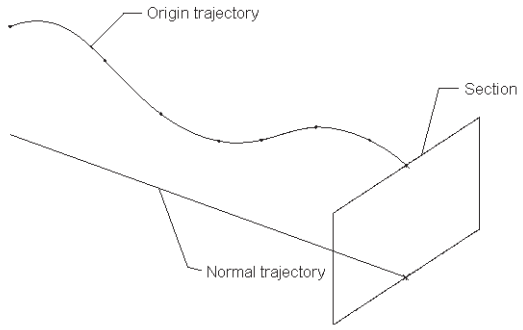


Figure 8-10 *Trajectories and section*

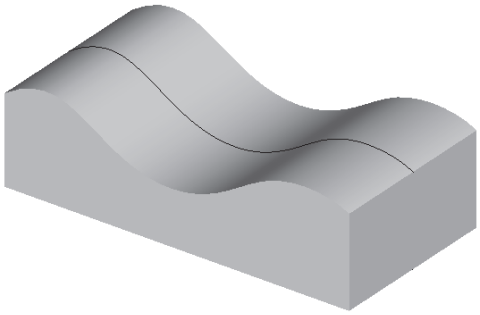


Figure 8-11 *Resultant swept feature*

can invoke the **SWEPT BLEND** option from the **Menu Manager** or from the menu bar.

Choose **Insert > Protrusion > Swept Blend** from the menu bar; the **BLEND OPTS** submenu is displayed. From the **Menu Manager**, choose **PART > Feature > Create > Solid > Protrusion > Advanced > Solid > Done**. The **ADV FEAT OPT** submenu is displayed. Choose the **Swept Blend** option and choose **Done**. The **BLEND OPTS** submenu is displayed as shown in Figure 8-12.

Figure 8-12 BLEND OPTS *submenu*

Figure 8-13 shows the three rectangular sections. The three sections are connected through a trajectory by connecting their vertices. The shaded view of the model is shown in Figure 8-14. The model is also shelled.

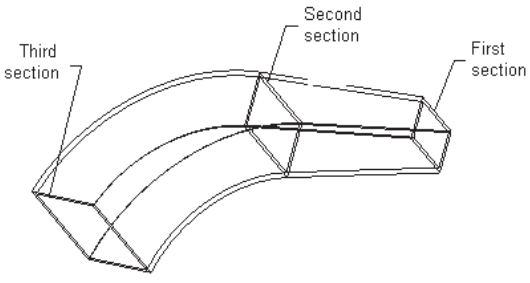


Figure 8-13 *Three sections in swept blend*

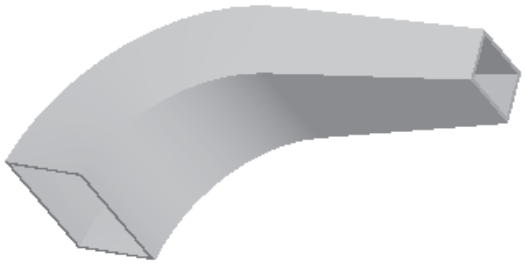


Figure 8-14 *Shaded view of the feature*

Helical Sweep

The **Helical Sweep** option is used to create helical swept features. You have to define a trajectory that will revolve along an axis, a pitch value, and a cross-section to create a helical feature using this option. The distance of the trajectory from the center line defines the radius of the helical path and the length of the trajectory defines the length of the swept feature. The main use of this option is to create the helical springs and threads.

***Figure 8-15 ATTRIBUTES** menu*

When you choose the **Helical Sweep** option from the **Insert** menu in the menu bar or when you choose the **Helical Swp** option from the **ADV FEAT OPT** submenu and choose **Done**, the **ATTRIBUTES** menu is displayed as shown in Figure 8-15. The options in this menu are discussed next. Figure 8-16 shows the parameters to be defined for the helical sweep and Figure 8-17 shows the resultant helical sweep feature.

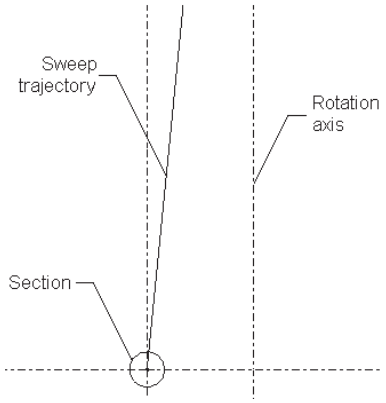


***Figure 8-16** Sweep trajectory, sweep section, and axis of rotation to create a helical feature*

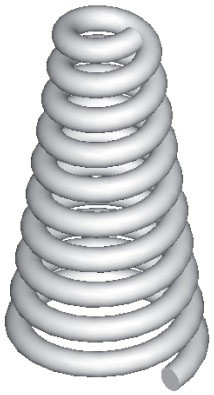


***Figure 8-17** The resultant helical feature*

Constant

The **Constant** option is used to create a helical feature with constant pitch as shown in Figure 8-18.

Variable

The **Variable** option is used to create a helical feature of varying pitch. While using this option you have to define the start pitch and the end pitch, you can also define a pitch between the start and the end point of the helix. The helical sweep feature with variable pitch is shown in Figure 8-19.

Figure 8-18 Helical sweep feature with constant pitch

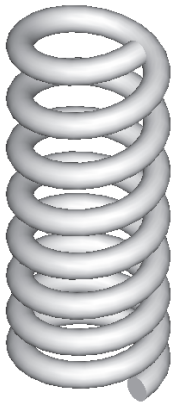


Figure 8-19 Helical sweep feature with variable pitch

Thru Axis

The **Thru Axis** option is used to create a helical feature around a axis.

Norm To Traj

The **Norm To Traj** option is used to create a helical feature perpendicular to the sketched trajectory.

Right Handed

The **Right Handed** option is used to create a helical feature in which the section is swept in the counterclockwise direction from the start sketch.

Left Handed

The **Left Handed** option is used to create a helical feature in which the section is swept in the clockwise direction from the start sketch.

Sect to Srfs or Blend Section to Surfaces

The **Sect to Srfs** (Section to Surface) option is used to blend a selected set of tangential surfaces with a sketched contour. You can invoke this options by choosing **PART > Feature > Create > Solid > Protrusion > Advanced > Solid > Done**. The contour sketched to create the feature using this option must be closed. This option can be used to create surface or a solid feature. Figure 8-20 shows the section to be blended with the selected set of tangential surfaces and Figure 8-21 shows the resultant blended feature.

Srfs to Srfs or Blend Between Surfaces

The Srfs to Srfs (Surface to Surface) option is used to create a blending between two curved quilts, two curved solid surfaces, or the combination of the both. The curved surfaces may be revolved features or spheres. You can invoke this option by choosing **PART > Feature > Create > Solid > Protrusion > Advanced > Solid > Done**. Figures 8-22 and 8-24 show the two curved surfaces to be blended and Figures 8-23 and 80-25 show the resultant blended feature using this option.

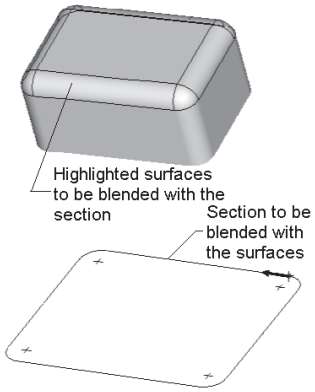


Figure 8-20 *Highlighted set of tangential surfaces and the contour sketch.*

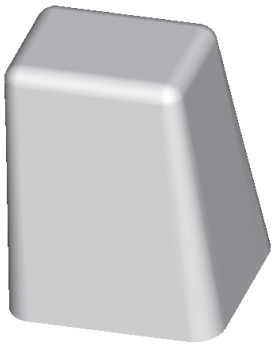


Figure 8-21 *Feature created using the* ***Sect to Srfs*** *option*

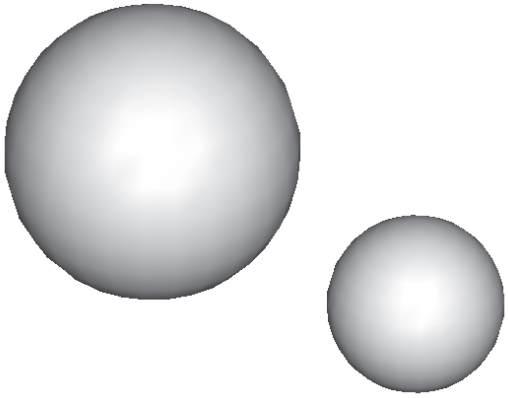


Figure 8-22 *Two spheres to be blended*

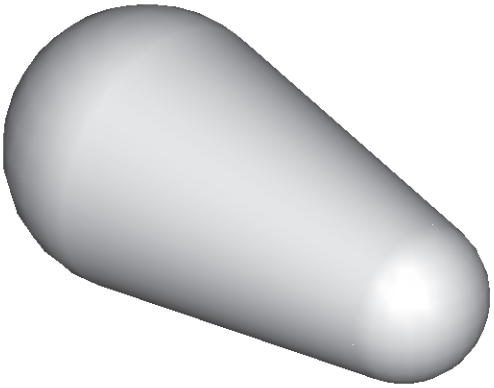


Figure 8-23 *Feature created using the* ***Srfs to Srfs*** *option*

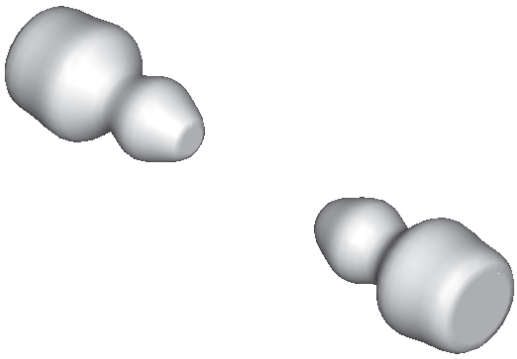


Figure 8-24 *Two curved features to be blended*

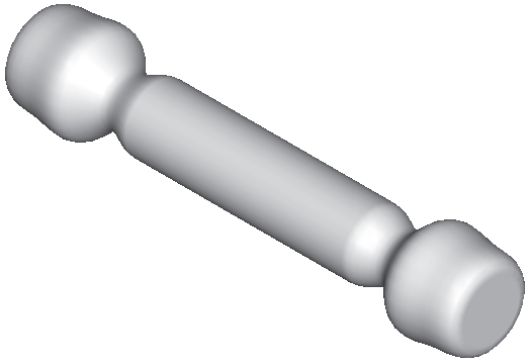


Figure 8-25 *Feature created using the* ***Srfs to Srfs*** *option*

From File

The **From File** option in the **ADV FEAT OPT** submenu reads the data points from an ASCII file for the creation of a blended feature. The type of blend and the cartesian coordinates of all the points are defined by the data file while using this option. A single coordinate system is used to locate the different blend points. You must import the data points for the curves before creating the blended feature as the data points for the curves ensure the smoothness of the blend created.

USING RELATIONS

Relations are user-defined equations that relate the dimension symbols or values. This option is used to create relations between various dimensions of a model and to keep the design intent of a model. For example, if it is required that the centre of the circle should lie at a distance of 1/3 from the vertex of a triangle, then it can be achieved using the relations. Relations can also be used to control the effect of modification on a model.

When you choose the **PART > Relations** from the **Menu Manager**, the **RELATIONS** submenu is displayed as shown in Figure 8-26.

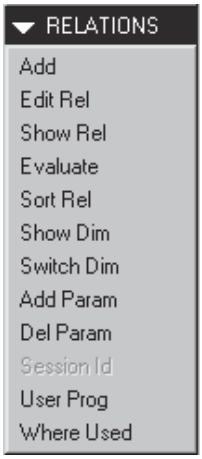


***Figure 8-26 RELATIONS** submenu*

Relations applied to a sketch can be modified and used in the part or assembly mode also. In the Sketch mode, the **RELATIONS** submenu can be invoked by choosing **Sketch > Relations** from the menu bar. Relations are of two types and are discussed next.

Equality Relations

The Equality relations are used to assign a value to a parameter or to a symbolic dimension. The variable on left of the equation is called driven and the variable on right of the equation is called driver. Another way of representing this is

driven=driver

For example,

d1=25
d2=d1+10
d5=d3*sin(d4)

In the above equations, symbol **d** represents the dimension created in the Part mode. Dimensions in the Sketch mode are represented by symbol **sd**, where **s** represents the Sketch mode and **d** the dimension.

In all of the above equations the expression on the right hand side of the equation determine the value of the variable on the left.

Comparison Relations

The Comparison relation compares the two sides of the equation. This type of relations are used in two ways: by giving conditional statements or by defining constrains.

For example,

As conditional statements

```
If sd1>=10
sd2=20
else
sd2=30
```

As a constraint

```
sd1+sd2>sd3+10
sd3>sd2/2
```

Note

The assignment operator '=' assigns a value to the driven that is on the left of the equation, whereas in case of the comparison operator the driven is compared with the driver and the value is returned as True or False. Hence, the assignment operator '=' (equal to) is different from the comparison operator '=' (equal to). The comparison operator '=' is always used with 'IF', '>', or '<'.

For example, sd1>=10

If sd1 is greater than or equal to 10 then the value returned is True and if sd1 is less than or equal to 10 then the value returned is False.

If you use the IF condition in the ***Message Input Window*** *while entering relations, then you can exit the* ***Message Input Window*** *only when you type ENDIF in the window. This means that the IF and ENDIF statements form a loop. In case you use the IF condition and press ENTER, the* ***Message Input Window*** *will again appear to let you enter the next relation. This will continue until you enter ENDIF in the window.*

For example,

```
IF sd1>=10
   sd2=20
ELSE
   sd2=30
ENDIF
```

Add

When you invoke the **Add** option of the **RELATION** submenu, you are prompted to enter the relation in the message box. Remember to use the appropriate dimension symbols while giving relations. The relations you enter are saved in **rel.ptd** file.

1. Choose **RELATION** from the **SKETCHER** menu. The dimensions are displayed in a symbolic form.

2. Choose **Add** from the **RELATIONS** submenu. You will be prompted to enter the relation.

3. The relations are added using the sketcher dimension symbols or the known dimension symbols. Press ENTER after adding all the relations. Regenerate the sketch. The changes will appear in the sketch as per the relations added.

Note

*When the **Intent Manager** is off, the relations are not evaluated by Pro/ENGINEER until you regenerate the sketch.*

Edit Rel

The **Edit Rel** (Edit Relations) option of the **RELATIONS** submenu allows you to edit the relations you have entered or the relations associated with the current sketch.

When you choose this option, the rel.ptd file is opened in the system editor and you can edit the relations there. If the relation contains an error you can rectify it. If before rectifying the errors you close the notepad window, a message box prompts you if you want to leave the relation as it is or if you want to correct it. If you choose **YES**, then the same file is again opened and now you can make the corrections. You can also write new relations in this file in lieu of using the **Add** option in the **RELATIONS** submenu.

The obsolete relations are not automatically deleted by Pro/ENGINEER but have to be deleted manually. This can be done using the **Edit Rel** option. After you make changes to the rel.ptd file and close it, you are prompted to save the changes you made to the file.

Note

*Although you can add relations using the **Edit Rel** option but this method is not recommended, the reason being that relations added using the **Edit Rel** option are not checked for errors instantaneously.*

Show Rel

You can view the relations of a sketch by choosing the **Show Rel** (Show Relations) option of the **RELATIONS** submenu. When you choose this option, all the relations associated with the current sketch are displayed with their values in the **Information Window**. This option behaves different with different modes of Pro/ENGINEER.

Evaluate

The **Evaluate** option gives you the value of any dimensional symbol used in a relation. When you choose this option in the **RELATIONS** submenu, the **Message Input Window** appears. Enter in this window the dimensional symbol whose value you want to evaluate.

Sort Rel

The **Sort Rel** (Sort Relations) option of the **RELATIONS** submenu sorts the relations in an order. The relations are ordered so that a relation that depends on the value of another relation is evaluated after this relation.

For example, consider relations

sd0=sd1+sd2*2
sd2=sd3 - 2*sd4

Now when you choose the **Sort Rel** option, the relation will be ordered as shown below.

sd2=sd3 - 2*sd4
sd0=sd1+sd2*2

Note
*The conditional relations are not sorted using the **Sort Rel** option.*

Switch Dim

The **Switch Dim** (Switch Dimensions) option of the **RELATIONS** submenu toggles between the symbols and the value mode of the dimensions. You can choose this option to display the dimensions as values or as symbols.

User Prog

The **User Prog** (User Program) option of the **RELATIONS** submenu allows you to link a C program module with the sketch. When you choose this option the **USER PROG** submenu displays three options: **Edit**, **Link**, and **Run**. Pro/ENGINEER provides a template C file that helps you to read and use the C programming environment.

TUTORIALS

Tutorial 1

In this tutorial you will create the model shown in Figure 8-27. This figure also shows the sectioned top, front, right-side, and isometric views of the model. **(Expected time: 45 min)**

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is composed of three features, see Figure 8-27.

b. The base feature is a swept feature with variable section, see Figure 8-34. First, the origin trajectory will be sketched, see Figure 8-28, and then the X trajectory will be sketched, see Figure 8-29. Two additional trajectories will be sketched that will sweep the section along their paths, see Figure 8-30 and Figure 8-31. Then the section that will vary with the shape of the trajectories will be sketched, see Figure 8-33.

c. The second feature is a round, see Figure 8-36.

d. The third feature is a shell of thickness 2, see Figure 8-38.

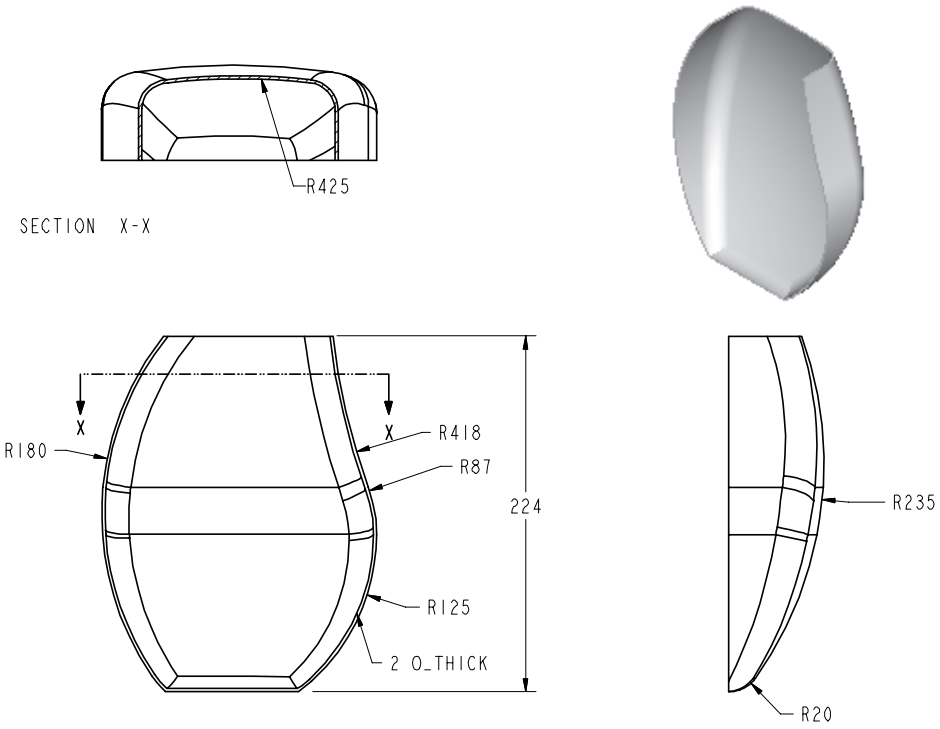


Figure 8-27 *Sectioned-top, front, right-side, and isometric views of the model*

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 directory named **c08**, if it does not exist. Choose the **New Directory** button in the **Select Working Directory** dialog box and create a directory named **c08** at **C:\ProE**.

Creating New Object File

1. Open a new part file and name it **c08tut1**. The three default datum planes will be displayed on the graphics screen. The **Model Tree** is also displayed on the graphics screen. Exit the **Model Tree** by choosing the **Model Tree on/off** button from the **Model Display** toolbar.

Creating the Base Feature

To create the sketch for the base feature, you first need to select the sketching plane to draw the origin trajectory. After selecting the sketching plane, draw the sketch for the origin trajectory and dimension it. Then you need to select the sketching plane for the X trajectory, draw the sketch for the X trajectory, and dimension it. The X trajectory defines the horizontal vector of a section. If the section is aligned with the X trajectory, the section varies along the path of the X trajectory. Similarly, two additional trajectories will be drawn to guide the section throughout the sweep. To create the base feature, you need to draw the origin trajectory on the **FRONT** datum plane.

1. Choose **Insert** > **Protrusion** > **Variable Section Sweep** from the menu bar.

   You can also invoke the **Variable Section Sweep** option from the **ADV FEAT OPT** submenu that is displayed when you choose **PART** > **Feature** > **Create** > **Protrusion** > **Advanced** > **Solid** > **Done** from the **Menu Manager**. The **Var Sec Swp** option is selected by default in the **ADV FEAT OPT** submenu. Choose **Done**.

   The **SWEEP OPTS** submenu is displayed. The **NrmToOriginTraj** option is selected by default. You will use this option because the transition of the sections is along the origin trajectory and the section is normal to the origin trajectory.

2. Choose **Done** from the **SWEEP OPTS** submenu. The **VAR SEC SWP** menu is displayed and you are prompted to specify the trajectory that defines the origin of the section.

3. Choose the **Sketch Traj** option from the **VAR SEC SWP** menu. You need to sketch the origin trajectory. The **SETUP PLANE** submenu is displayed and you are prompted to select the sketching plane.

4. Select the **FRONT** datum plane from the graphics screen. The **DIRECTION** submenu is displayed. Choose **Okay**. The **SKET VIEW** submenu is displayed.

5. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen.

6. Once you enter the sketcher environment, create the sketch of the origin trajectory. The origin trajectory is a straight line segment aligned to the **RIGHT** datum plane and the start point is aligned to the **TOP** datum plane. The sketch of the origin trajectory is shown in Figure 8-28.

Note

The start point of all the trajectories should be noted before exiting the sketcher environment for each trajectory. In the sketcher environment, you can change the start point by selecting the point where you want the start point. When the point turns red in color, press and hold down the right mouse button to invoke the shortcut menu and choose the ***Start Point*** *option from the shortcut menu.*

7. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

   The **VAR SEC SWP** menu is displayed and you are prompted to specify the trajectory that defines the section horizontal vector.

8. Choose the **Sketch Traj** option from the **VAR SEC SWP** menu. You need to sketch the X- trajectory. The **SETUP PLANE** submenu is displayed and you are prompted to select the sketching plane.

9. Select the **FRONT** datum plane from the graphics screen. The **DIRECTION** submenu is

displayed. Choose **Okay**. The **SKET VIEW** submenu is displayed.

10. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen.

11. Once you enter the sketcher environment, create the sketch of the of the X trajectory. The X trajectory is drawn using three arcs. It is recommended that you start the sketch from the top arc and then change the start point. To change the start point, select the point, it turns red in color. Press and hold the right mouse button and select **Start Point** option from the shortcut menu. The sketch of the X trajectory is shown in Figure 8-29.

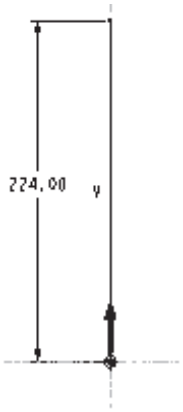


Figure 8-28 *Sketch with dimension of the origin trajectory*

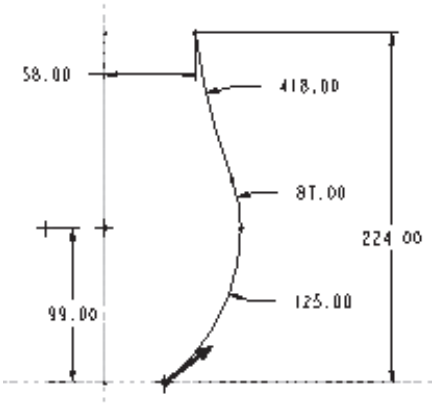


Figure 8-29 *Sketch with dimensions of the X trajectory*

As evident from the sketch of the X trajectory, the center points of the bottom and the second arcs are aligned horizontally.

12. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

    The **VAR SEC SWP** menu is displayed and you are prompted to specify another trajectory or choose Done.

13. Choose the **Sketch Traj** option from the **VAR SEC SWP** menu. You need to sketch a additional trajectory. This trajectory will guide the section along its path. The **SETUP PLANE** submenu is displayed and you are prompted to select the sketching plane.

14. Select the **FRONT** datum plane from the graphics screen. The **DIRECTION** submenu is displayed. Choose **Okay**. The **SKET VIEW** submenu is displayed.

15. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen.

16. Once you enter the sketcher environment, create the sketch of the trajectory as shown in Figure 8-30. This trajectory is created using a single arc. If the start point is on the upper

endpoint, change it to the lower endpoint by selecting the lower endpoint. When the point turns red in color, right-click to invoke the shortcut menu, and choose the **Start Point** option from the shortcut menu.

17. After the sketch is complete, choose the **Continue with the current section** button to exit the sketcher environment.

    The **VAR SEC SWP** menu is displayed and you are prompted to specify another trajectory or choose Done.

18. Similarly, create another trajectory on the **RIGHT** datum plane and keep the **TOP** datum plane at the left. The sketch of the trajectory is shown in Figure 8-31.

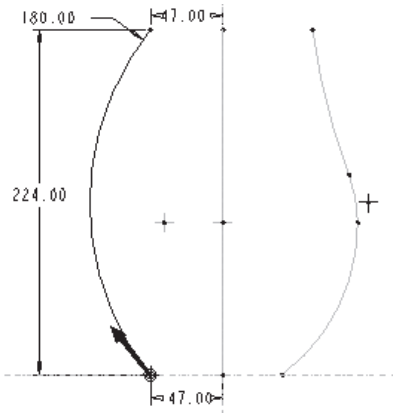


Figure 8-30 *Sketch with dimensions of the additional trajectory*

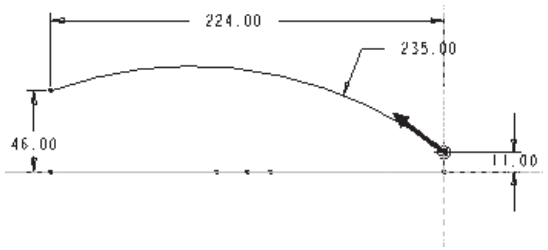


Figure 8-31 *Sketch with dimensions of the additional trajectory*

19. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

    The **VAR SEC SWP** menu is displayed and you are prompted to specify another trajectory or choose Done.

20. Choose the **Default** option from the **Saved view list** button in the **Top Toolbar** to view the trajectories you have created, shown in Figure 8-32. This view gives a better view of trajectories in 3D space.

21. Choose **Done** from the **VAR SEC SWP** menu. The system takes you to the sketcher environment.

22. Draw the closed sketch of the section and apply the dimensions as shown in Figure 8-33. Note that the section is aligned to all the start points of the trajectories that are displayed by crosses. The cursor will automatically snap to these points as you move it close to these points. In order to vary the section with the trajectories, it is necessary to align the section with the trajectories.

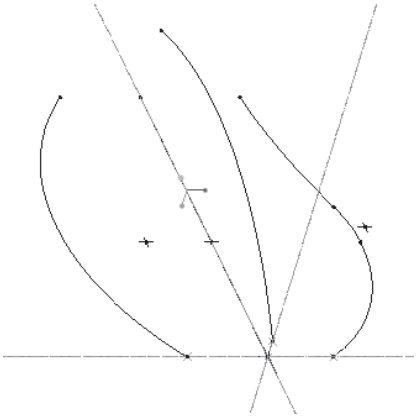


Figure 8-32 *Default view of the four trajectories*

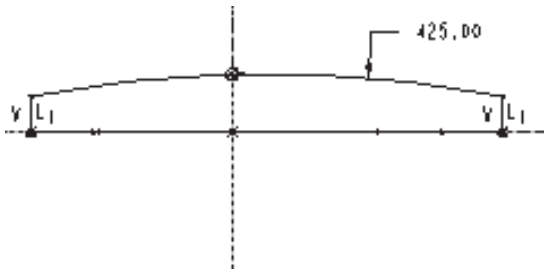


Figure 8-33 *Sketch of the section with dimensions*

23. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

24. Choose **OK** from the **PROTRUSION** dialog box. The variable section sweep feature is completed and is shown in Figure 8-34. You can use CTRL+middle mouse button to view the feature as shown in the figure.

Creating the Round Feature

You will create round on the edges of the base feature. The edges you need to round are shown in Figure 8-35 .

1. Choose **Insert** > **Round** from the menu bar. The **ROUND TYPE** menu is displayed.

2. Choose **Simple** > **Done**. The **RND SET ATTR** menu is displayed. The **Constant** > **Edge Chain** options are selected by default. Choose **Done**. The **CHAIN** menu is displayed.

3. The **Tangnt Chain** option is selected by default. Select the edges shown in Figure 8-35.

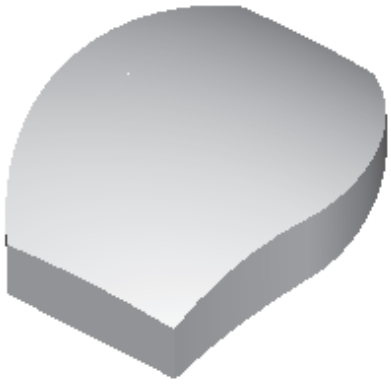


Figure 8-34 *Variable section swept feature*

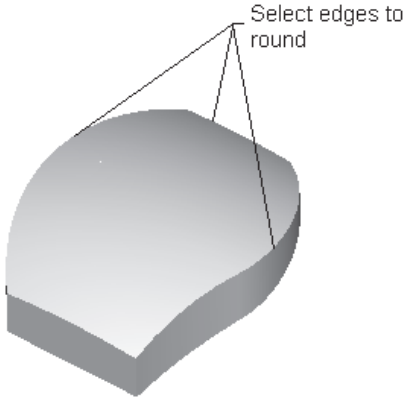


Figure 8-35 *Edges to be selected to round*

4. Choose **Done** from the **CHAIN** menu. The **Message Input Window** is displayed.

5. Enter **20** in the **Message Input Window** and press ENTER. Choose **OK** from the **ROUND** dialog box. The base feature after creating the round is shown in Figure 8-36.

Creating the Shell

Now, you need to create the shell and remove the front planar and bottom planar surfaces of the base feature.

1. Choose **Insert > Shell** from the menu bar. You are prompted to select the surfaces to be removed.

2. Select the surfaces shown in Figure 8-37 to remove.

Figure 8-36 *Model after creating round*

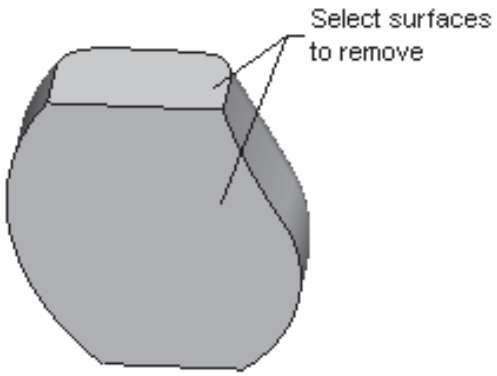


Figure 8-37 *Surfaces to be selected to remove*

3. Choose **Done Sel** from the **GET SELECT** submenu and choose **Done Refs** from the **FEATURE REFS** menu. The **Message Input Window** is displayed.

4. Enter **2** in the **Message Input Window** and press ENTER. Choose **OK** from the **SHELL** dialog box. The model after creating the shell feature is shown in Figure 8-38.

Saving the Model

You have to save the model because you may need it later.

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 8-39.

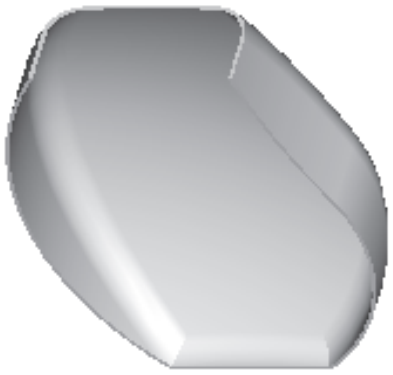


Figure 8-38 *Final model after shelling*

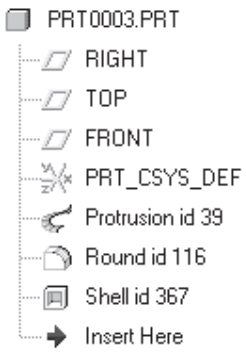


Figure 8-39 *Model Tree for Tutorial 1*

Tutorial 2

In this tutorial you will create the model shown in Figure 8-40 . This model is a Upper Housing of a motor blower assembly. Figure 8-41 shows the left-side view of the top view, top view, front view, and the sectioned left-side view of the model. **(Expected time: 1 hr)**

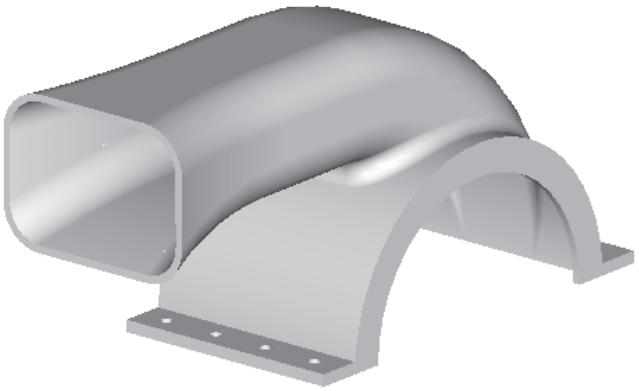


Figure 8-40 *Solid model of the Upper Housing*

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is composed of ten features, see Figure 8-40.

b. The base feature is an extruded feature, see Figure 8-41. Select the sketching plane for the base feature, draw the sketch using the sketcher tools, and apply dimensions, see Figure 8-42.

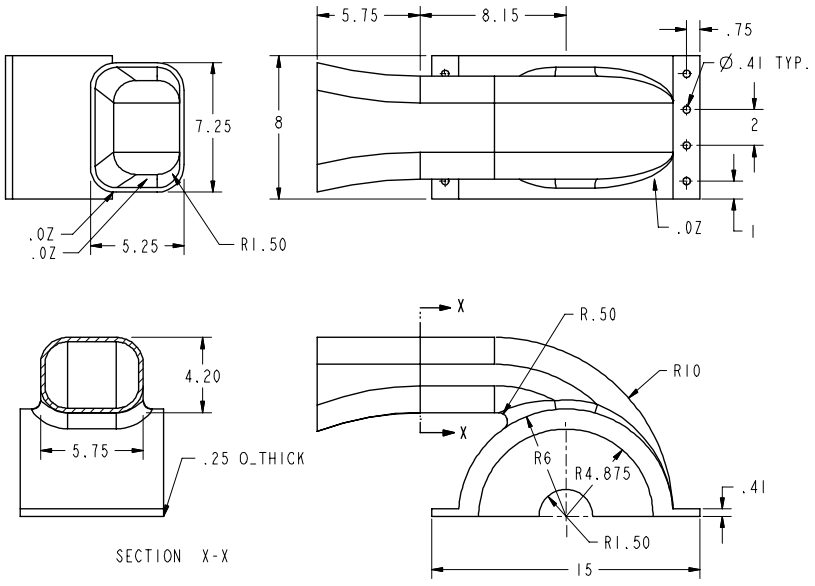


Figure 8-41 *Left-side view of the top view, top view, front view, and the sectioned left-side view of the model*

c. The second feature is a swept blend feature in which the section is normal to the origin trajectory, see Figure 8-49. First the sketch for the origin trajectory will be drawn and dimensioned and then three sections will be defined along the origin trajectory.

d. The third feature is a round of radius 1.5, see Figure 8-50.

e. The fourth feature is a round of radius 0.5, see Figure 8-52.

f. The fifth feature is a shell of thickness 0.25 that will be created on the front planar surface of the swept blend feature and on the bottom planar surface of the base feature, see Figure 8-53.

g. The sixth feature is an extruded cut, see Figure 8-55. Select the sketching plane for the cut feature, draw the sketch using sketching tools, and apply the dimensions, see Figure 8-54.

h. The seventh feature is also an extruded cut, see Figure 8-57. Select the sketching plane for the cut feature, draw the sketch using sketching tools, and apply the dimensions, see Figure 8-56.

i. The eighth feature is an extruded feature, see Figure 8-59. Select the sketching plane for the extruded feature, draw the sketch, and apply constraints and dimensions, see Figure 8-58.

j. The ninth feature is a copy of the eighth feature, see Figure 8-60.

k. The tenth feature is a hole, see Figure 8-61, and this hole will be patterned. After you pattern the hole, the hole and the pattern feature will be combined in a single feature.

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 **c08** in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it **c08tut2**. The three default datum planes and the **Model Tree** are displayed on the graphics screen. Exit the **Model Tree**. The **Model Tree** will not appear if it was previously turned off.

Creating the Base Feature

To create the sketch for the base feature, you first need to select the sketching plane for the base feature. In this model, you need to draw the base feature on the **RIGHT** datum plane.

1. Choose **Insert > Protrusion > Extrude** from the menu bar. The **ATTRIBUTES** menu is displayed. Choose **Both Sides > Done** from this menu.

2. Select the **RIGHT** datum plane as the sketching plane. The **DIRECTION** submenu is displayed.

3. Choose **Okay** from this submenu. The **SKET VIEW** submenu is displayed.

4. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen.

5. Once you enter the sketcher environment, create the sketch of the base feature and apply dimensions as shown in Figure 8-42.

6. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment. The **SPEC FROM** menu is displayed and you are prompted to specify the depth of extrusion.

7. The **Blind** option is selected by default in the **SPEC FROM** menu; choose **Done**.

8. Enter a depth of **8** in the **Message Input Window** that appears and press ENTER. Choose **OK** from the **PROTRUSION** dialog box.

   The base feature is completed and the default trimetric view is shown in Figure 8-43.

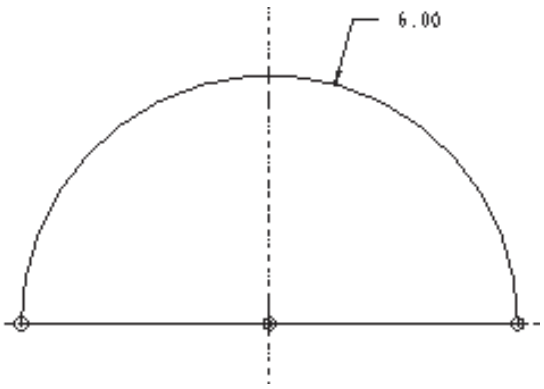


Figure 8-42 Sketch for the base feature

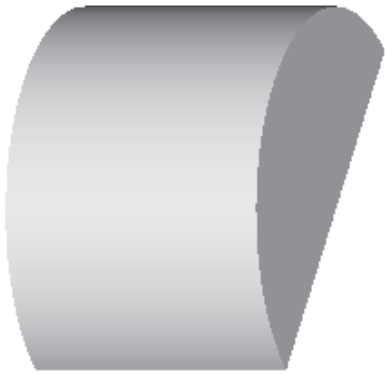


Figure 8-43 Default trimetric view of the base feature

Creating the Swept Blend

The second feature is a swept blend. You will sketch the trajectory for the blend on the **RIGHT** datum plane. Note that the trajectory is tangent to the curve of the base feature and the start point of the trajectory is aligned with the **TOP** datum plane.

1. Choose **Insert** > **Protrusion** > **Swept Blend** from the menu bar. The **BLEND OPTS** submenu is displayed. The **Sketch Sec** and the **NrmToOriginTraj** options are selected by default.

2. Choose **Done** from the **BLEND OPTS** submenu. The **SWEEP TRAJ** menu is displayed.

3. Choose the **Sketch Traj** option from the **SWEEP TRAJ** menu. You are prompted to select a sketching plane.

4. Select the **RIGHT** datum plane from the graphics screen. The **DIRECTION** menu is displayed. Choose **Okay**. The **SKET VIEW** submenu is displayed.

5. Select the **Top** option from the submenu and choose the **TOP** datum plane.

6. Once you enter the sketcher environment, draw the sketch of the trajectory and add constraints and dimensions as shown in Figure 8-44.

   As evident from Figure 8-44, the start point of the trajectory is aligned with the **TOP** datum plane and the trajectory is tangent to the curve of the base feature.

7. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

   The **CONFIRM SEL** menu is displayed. Note that on the origin trajectory one red, one white, and two green points are displayed. The green points indicate that these are fixed points for defining the sections. The location for red point can be changed by using the

options in the **CONFIRM SEL** menu.

Note

The two green points are fixed at the start point and at the endpoint of the trajectory, indicating that it is necessary to draw sections at these points in order to complete the swept blend feature. This is because swept blend is created by blending the sections along the trajectory. There can be any number of sections that can be possible between the start and the endpoint of the trajectory.

8. Choose **Next** from the **CONFIRM SEL** menu to shift the position of the red point. You need to shift the position of the red point because the section of the swept blend feature (SECTION X-X) that is given in Figure 8-44 is at the point where the red point is shifted now. Hence, now you will need to define three sections at three points.

9. Choose the **Accept** option from the **CONFIRM SEL** menu. The **Message Input Window** is displayed and you are prompted to enter the z-axis rotation angle for section 1. Section 1 is at the start point of the origin trajectory.

10. The value **0** is entered by default in the **Message Input Window**; press ENTER.

    The system takes you to the sketcher environment. Now, you need to sketch the first section of the blend at the start point of the trajectory. You will notice a coordinate system is placed at the start point of the trajectory. You need to align the sketch of the section to this coordinate system.

11. Use CTRL+middle mouse button to spin the view of the sketch as shown in Figure 8-45 to have a better understanding of the sketch in 3D space.

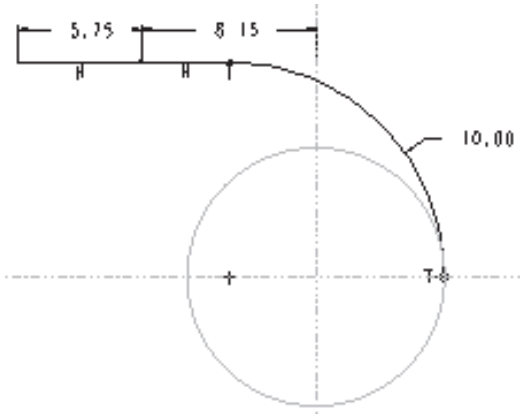


Figure 8-44 *Sketch of the origin trajectory*

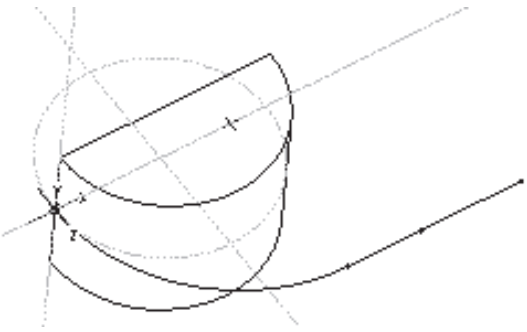


Figure 8-45 *Spined view of the sketch showing the origin trajectory, base feature, and the default coordinate system*

12. Choose the **Orient the sketching plane parallel to the screen** button from the **Top Toolchest**. The sketching plane reorients parallel to the graphics screen.

13. Draw the sketch of the first section and add dimensions as shown in Figure 8-46.

14. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment. The **Message Input Window** is displayed and you are prompted to enter the z-axis rotation angle for section 2.

15. The value **0** is entered by default in the **Message Input Window**; press ENTER.

16. Draw the sketch of the second section and add dimensions as shown in Figure 8-47. This section is at the point between the first and the third section and in Figure 8-41, this section is named as X-X.

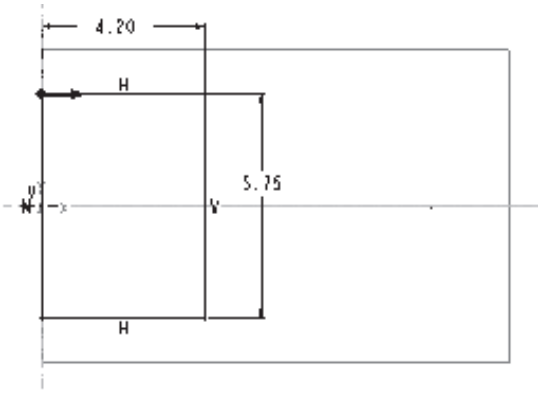


Figure 8-46 *Sketch for first section with dimensions*

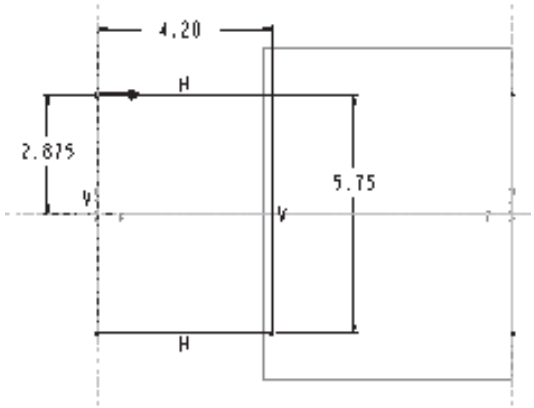


Figure 8-47 *Sketch for second section with dimensions*

17. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment. The **Message Input Window** is displayed and you are prompted to enter the z-axis rotation angle for section 3.

16. The value **0** is entered by default in the **Message Input Window**; press ENTER.

17. Draw the sketch of the third section and add dimensions as shown in Figure 8-48. This is the last section on the origin trajectory. Remember to align the section to the coordinate system.

18. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

19. Choose **OK** from the **PROTRUSION** dialog box. The default trimetric view of the swept blend feature with the base feature is shown in Figure 8-49.

Creating the Round Features

The two round features are created before shelling because the rounds are also to be shelled with other surfaces. The third feature in the model is a round of radius 1.5 and the fourth

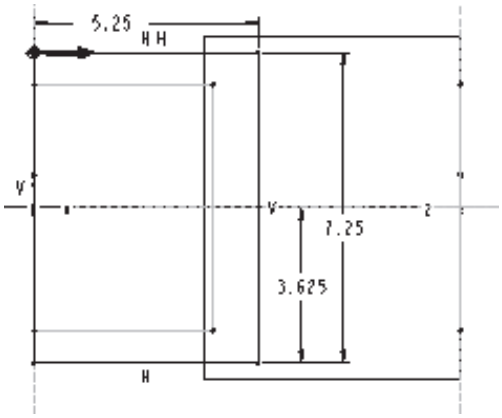


Figure 8-48 *Sketch of the third section with dimensions*

Figure 8-49 *Default view of the swept blend*

feature is a round of radius 0.5.

1. Choose **Insert > Round** from the menu bar. The **ROUND TYPE** menu is displayed.

2. Choose **Simple > Done**. The **RND SET ATTR** (Round Set Attributes) menu is displayed. The **Constant** and **Edge Chain** options are selected by default.

3. Choose **Done** from the **RND SET ATTR** menu. The **CHAIN** menu is displayed. The **Tangnt Chain** option is selected by default and you are prompted to select the edges to round.

4. Select all the four longer edges of the swept feature shown in Figure 8-50 and choose **Done** from the **CHAIN** menu. The **Message Input Window** is displayed with a default value of radius.

5. Enter **1.5** in the **Message Input Window** and press ENTER.

6. Choose **OK** from the **ROUND** dialog box. The default view after creating the round is shown in Figure 8-51.

   The third feature of the model is completed and now you will create the fourth feature, that is, a round of radius 0.5.

7. Choose **Insert > Round** from the menu bar. The **ROUND TYPE** menu is displayed.

8. Choose **Simple > Done**. The **RND SET ATTR** menu is displayed. The **Constant** and **Edge Chain** options are selected by default.

9. Choose **Done** front the **RND SET ATTR** menu. The **CHAIN** menu is displayed. The **Tangnt Chain** option is selected by default and you are prompted to select the edges to round.

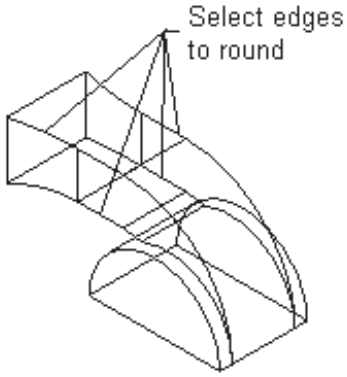


Figure 8-50 *Edges to round*

Figure 8-51 *Default view of the model after creating a round feature of radius 1.5.*

10. Select the edge where the base feature and the swept feature joins as shown in Figure 8-52.

    The **Message Input Window** is displayed with a default value of radius.

11. Enter a value of **0.5** in the **Message Input Window** and press ENTER.

12. Choose **OK** from the **ROUND** dialog box. The round feature is shown in Figure 8-53. You can use the CTRL+middle mouse button to orient the model as shown in Figure 8-53.

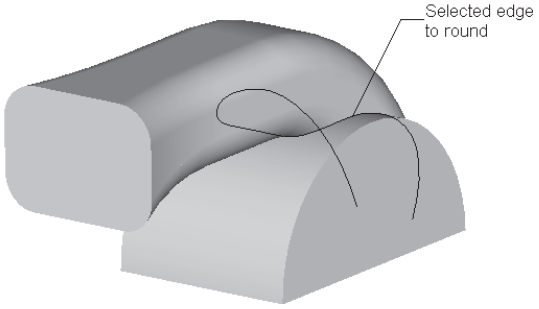


Figure 8-52 *Figure showing a solid model*

Creating the Shell

You need to create a shell of thickness 0.25 on the swept blend feature.

1. Choose **Insert > Shell** from the menu bar. You are prompted to select the surface to remove.

2. Select the front planar surface of the swept blend feature and bottom planar surface of the base feature. Choose **Done Sel** from the **GET SELECT** submenu or use the middle mouse button to confirm the selection.

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

4. Enter a value of **0.25** in the **Message Input Window** and press ENTER.

5. Choose **OK** from the **SHELL** dialog box. The shell feature is completed and is shown in Figure 8-54.

Figure 8-53 *Model after creating a round feature of radius 0.5*

Figure 8-54 *Default trimetric view after shelling*

Creating the Extruded Cut

The extruded cut is the sixth feature of this model. You need to create the cut on the front planar surface of the base feature.

1. Choose **Insert > Cut > Extrude**. The **ATTRIBUTES** menu is displayed.

2. Choose **One Side > Done**. Select the front planar surface of the base feature as the sketching plane and choose **Okay** from the **DIRECTION** submenu.

3. Select the **Top** option from the **SKET VIEW** submenu and choose the **TOP** datum plane.

4. Draw the sketch of the cut feature and apply dimensions as shown in Figure 8-55. You can use the **Create concentric arc** button from the **Right Toolchest** to draw the sketch.

5. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment. The **DIRECTION** menu is displayed. Choose **Okay**. The **SPEC TO** menu is displayed.

6. Select the **Thru Next** option from the **SPEC TO** menu and choose **Done**.

7. Choose **OK** from the **CUT** dialog box. The cut feature is shown in Figure 8-56.

   Similarly, create the next extruded cut feature at the back planar surface of the base feature. Select the sketching plane, draw the sketch, apply dimensions, and extrude the sketch using the **Thru Next** option. The sketch for the cut feature is shown in Figure 8-57.

   The seventh feature of the model, that is, cut feature is shown in Figure 8-58.

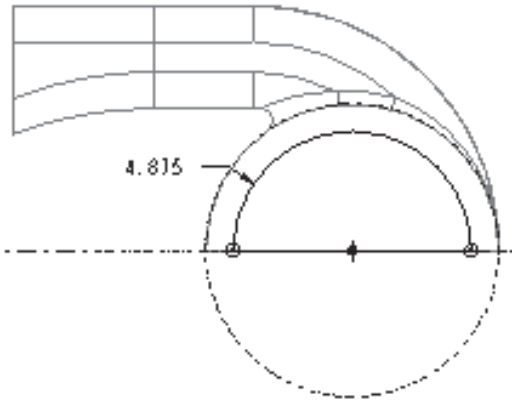


Figure 8-55 *Sketch with dimension for cut feature*

Figure 8-56 *Model with cut feature*

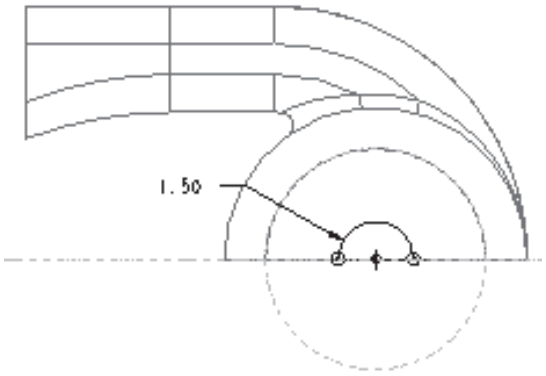


Figure 8-57 *Sketch with dimension for cut feature*

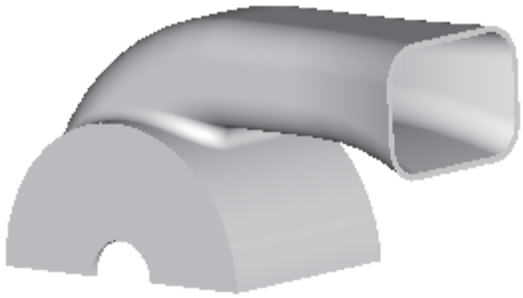


Figure 8-58 *Model with the cut feature*

Creating the Extruded Feature

The eighth feature of the model is an extruded feature and will be created on the front planar surface of the base feature.

1. Select the front planar surface of the base feature as the sketching plane for the extruded feature. Once you enter the sketcher environment, draw the sketch and apply constraints and dimensions as shown in Figure 8-59.

2. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

3. From the **SPEC TO** menu, select the **UpTo Surface** option and choose **Done**. You are prompted to select a surface to extrude upto.

4. Select the back surface of the base feature. Choose **OK** from the **PROTRUSION** dialog box; the extruded feature is completed and is shown in Figure 8-60.

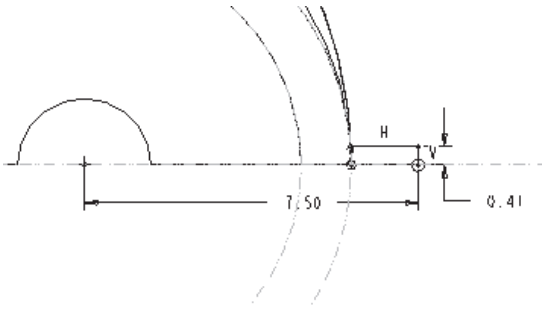


Figure 8-59 *Sketch for the extruded feature*

Figure 8-60 *Model after creating the extrude feature*

Creating the Copy of the Eighth Feature

You need to create a mirror copy of the extruded feature as shown in Figure 8-61 and this will be the ninth feature of the model. The extruded feature will be mirrored about the **FRONT** datum plane.

1. Choose **PART > Feature > Copy** from the **Menu Manager**. The **COPY FEATURE** submenu is displayed.

2. Choose **Mirror > Select > Dependent > Done** from the **COPY FEATURE** submenu. The **SELECT FEAT** submenu is displayed and you are prompted to select the features to be mirrored.

3. Select the previous extruded feature from the graphics screen and choose **Done** from the **SELECT FEAT** submenu. You are prompted to select a plane or create a datum plane to mirror about.

4. Select the **FRONT** datum plane as the mirror plane. The protrusion feature is mirrored about the **FRONT** datum plane as shown in Figure 8-61.

Creating the Tenth Feature

The tenth feature is a through hole and will be created using the **HOLE** dialog box.

1. Choose **Insert > Hole** from the menu bar. The **HOLE** dialog box is displayed. The **Straight hole** radio button in the **Hole Type** area is selected by default.

   Create the hole as shown in Figure 8-62 by specifying the placement parameters. The placement parameters can be referred from Figure 8-41.

Creating a Pattern of the Hole Feature

As evident from Figure 8-41, you need to create eight instances of the hole. The first instance

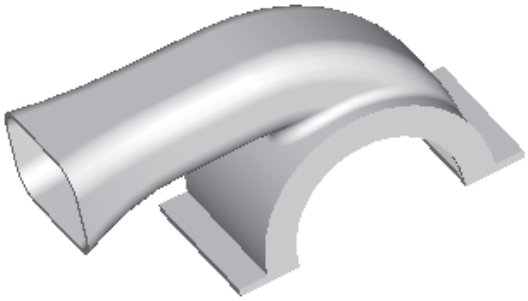


Figure 8-61 *Model after mirroring the extruded feature*

Figure 8-62 *Model after creating the hole*

is created by using the **HOLE** dialog box and you can use the **Pattern** option to create the remaining seven instances. You will create a rectangular pattern of the hole feature. You can also create all the holes using the **HOLE** dialog box and specifying the placement parameters for each of them. But to save time, you will create a pattern of the hole.

1. Choose **PART > Feature > Pattern** from the **Menu Manager**. You are prompted to select the feature to be patterned.

2. Select the hole that is on the base feature. The **PAT OPTIONS** submenu is displayed.

3. Select the **General** option from the **PAT OPTIONS** submenu and choose **Done**. The placement dimensions are displayed on the hole feature.

4. Select the dimension value **1** from the graphics screen. After you select the dimension in the first direction, the **Message Input Window** is displayed and you are prompted to specify the increment in dimension.

5. Enter a value of **2** in the **Message Input Window** and press ENTER. You are prompted to select a dimension in the same direction or choose Done.

6. Choose **Done** from the **EXIT** submenu. The **Message Input Window** is displayed and you are prompted to enter the total number of instances in that direction including the original.

7. Enter **4** in the **Message Input Window** and press ENTER. Now, you are prompted to enter dimension increment in the second direction.

8. Select the dimension value **0.75** from the graphics screen. After you select the dimension in the second direction, the **Message Input Window** is displayed and you are prompted to specify the increment in dimension value.

9. Enter a value of **13.5** in the **Message Input Window** and press ENTER. You are prompted to select another dimension in the same direction or choose Done.

10. Choose **Done** from the **EXIT** submenu. The **Message Input Window** is displayed and you are prompted to enter the total number of instances in that direction including the original.

11. Enter **2** in the **Message Input Window** and press ENTER. The rectangular pattern of holes will be displayed as shown in Figure 8-58. You can use CTRL+middle mouse button to spin the model and to display the model as shown in Figure 8-63.

Saving the Model

You have to save the model because you may need it later.

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 8-64.

Figure 8-63 *Completed model*

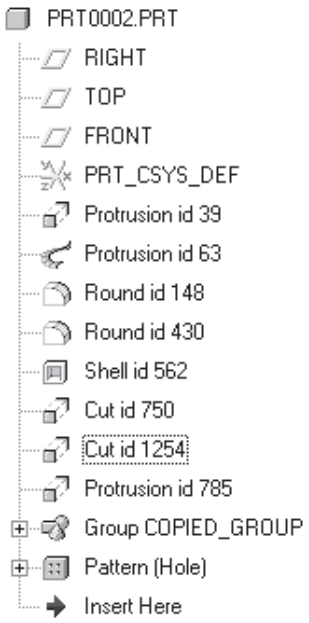


Figure 8-64 Model Tree *for Tutorial 2*

Tutorial 3

In this tutorial you will create the model shown in Figure 8-65. This model is a wheel of a car. Figure 8-66 shows the top view, sectioned front view, sectioned right view, detail view, and two blend sections with dimensions. **(Expected time: 45 min)**

The following steps outline the procedure for creating this model:

a. First examine the model and then determine the number of features in it. The model is

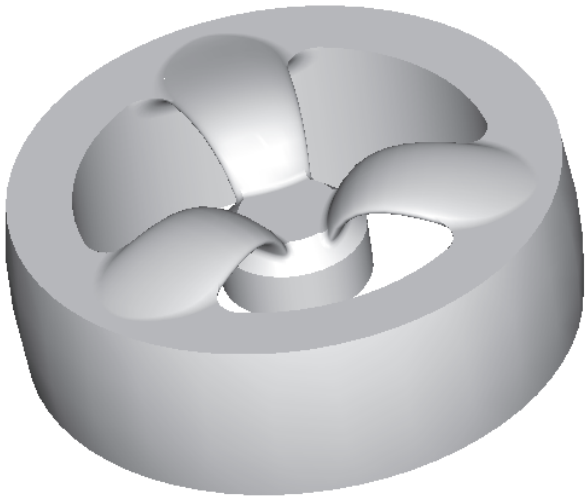


Figure 8-65 *Solid model of the wheel*

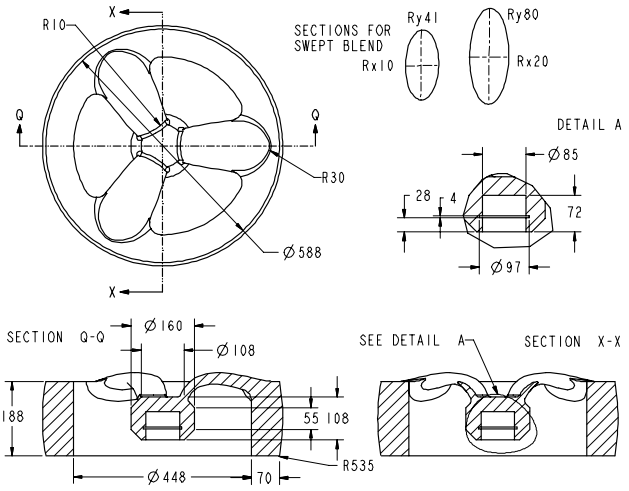


Figure 8-66 *Top view, sectioned front view, sectioned right view, detail view, and two blend sections with dimensions*

composed of five features, see Figure 8-65.

b. The base feature is a revolved feature, see Figure 8-68. Select the sketching plane for the base feature, draw the sketch using the sketcher tools, and apply constraints and dimensions, see Figure 8-67.

c. The second feature is a swept blend feature in which the section is normal to the origin

trajectory, see Figure 8-72. First the sketch for the origin trajectory will be drawn and dimensioned and then two sections at the start and endpoints will be defined along the origin trajectory. Then this feature will be patterned, see Figure 8-73.

d. The third feature is a round of radius 30 and the fourth feature is a round of radius 10, see Figure 8-75 and Figure 8-76 respectively.

e. The fifth feature is a revolved cut, see Figure 8-78. First you will draw the sketch of the cut feature on the sketching plane, apply constraint and dimensions, and revolve the sketch to 360-degree.

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 **c08** in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it **c08tut3**. The three default datum planes and the **Model Tree** are displayed on the graphics screen. Exit the **Model Tree**. The **Model Tree** will not appear if it has been previously turned off.

Creating the Base Feature

To create the sketch for the base feature, you first need to select the sketching plane. In this model, you need to draw the base feature on the **FRONT** datum plane.

1. Choose **Insert > Protrusion > Revolve** from the menu bar. The **ATTRIBUTES** menu is displayed. Choose **One Side > Done** from this menu.

2. Select the **FRONT** datum plane as the sketching plane. The **DIRECTION** submenu is displayed.

3. Choose **Okay** from this submenu. The **SKET VIEW** submenu is displayed.

4. Select the **Top** option from this submenu and choose the **TOP** datum plane from the graphics screen.

5. Once you enter the sketcher environment, create the sketch of the base feature and apply constraints and dimensions as shown in Figure 8-67. The sections must be closed.

6. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

   The **REV TO** menu is displayed and you are prompted to specify the angle of revolution.

7. Select the **360** option from the **REV TO** menu and choose **Done**.

8. Choose **OK** from the **PROTRUSION** dialog box.

The base feature is completed and the default trimetric view is shown in Figure 8-68.

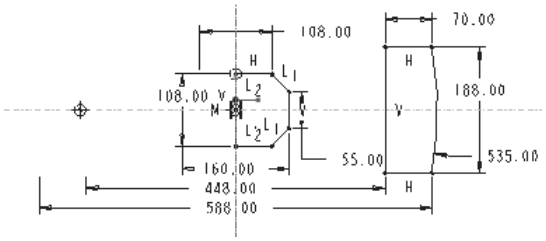


Figure 8-67 *Sketch with dimensions for the base feature*

Figure 8-68 *Default trimetric view of the base feature*

Creating the Swept Blend

The second feature of the model is a swept blend. You need to create a datum plane that is at an angle of 90-degree from the **RIGHT** datum plane. This is because later in the tutorial, you will create a rotational pattern of the swept blend feature. The 90-degree angle value will be used for the dimensional increment for creating the rotational pattern.

1. Choose **Insert** > **Protrusion** > **Swept Blend** from the menu bar. The **BLEND OPTS** submenu is displayed. The **Sketch Sec** and the **NrmToOriginTraj** options are selected by default.

2. Choose **Done** from the **BLEND OPTS** submenu. The **SWEEP TRAJ** menu is displayed.

3. Choose the **Sketch Traj** option from the **SWEEP TRAJ** menu. You are prompted to select a sketching plane or create a datum plane.

4. Choose **Make Datum** option from the **SETUP PLANE** submenu. **The DATUM PLANE** submenu is displayed.

5. Choose the **Through** option from the **DATUM PLANE** submenu and select the axis named **A_1** from the graphics screen. The **A_1** axis is the axis of revolution of the base feature.

6. Choose the **Angle** option from the **DATUM PLANE** submenu and select the **RIGHT** datum plane.

7. Choose **Done** from the **DATUM PLANE** submenu. The **OFFSET** submenu is displayed.

8. Choose the **Enter Value** option from the **OFFSET** submenu. The **Message Input Window** is displayed.

9. Enter **90** in the **Message Input Window** and press ENTER. The **DIRECTION** menu is displayed. Choose **Okay**. The **SKET VIEW** submenu is displayed.

10. Select the **Top** option from the submenu and choose the **TOP** datum plane.

11. Once you enter the sketcher environment, first you need to specify two references to dimension. Select the **TOP** and **RIGHT** datum planes.

12. Draw the sketch of the trajectory and add constraints and dimensions as shown in Figure 8-69. In Figure 8-69, the hidden lines are also shown because in this case the display of the sketch is more clear while drawing the sketch with display of hidden lines. The curve is drawn using the spline. The spline has only four control points: one start point, one endpoint, and two intermediate point. The dimensions shown in Figure 8-69 are enough to define the sketch.

Note

The shape of the spline will differ from user to user, and so the shape of the protrusion feature may vary.

13. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

    The **CONFIRM SEL** menu is displayed. Note that on the origin trajectory two green points are displayed. The green points indicate that they are fixed points for defining the sections.

14. Choose the **Next** option twice from the **CONFIRM SEL** menu. The **Message Input Window** is displayed and you are prompted to enter the z-axis rotation angle for section 1. Section 1 is at the start point of the origin trajectory.

15. The value **0** is entered by default in the **Message Input Window**; press ENTER.

    The system takes you to the sketcher environment. Now, you need to sketch the first section of the blend at the start point of the trajectory. You will notice a coordinate system is placed at the start point of the trajectory. You need to align the sketch of the section to this coordinate system.

16. Draw the sketch of the first section and add dimensions as shown in Figure 8-70.

17. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment. The **Message Input Window** is displayed and you are prompted to enter the z-axis rotation angle for section 2.

18. The value **0** is entered by default in the **Message Input Window**; press ENTER.

19. Draw the sketch of the second section and add dimensions as shown in Figure 8-71. The second section is also elliptical in shape, and therefore an ellipse is drawn in the sketch.

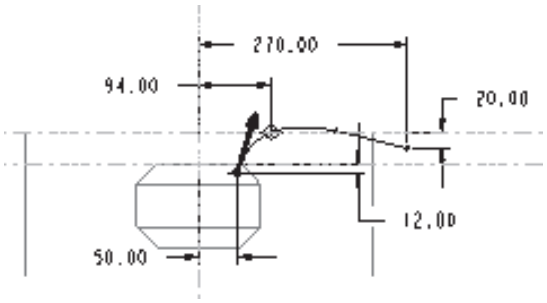


Figure 8-69 Sketch with dimensions for the origin trajectory

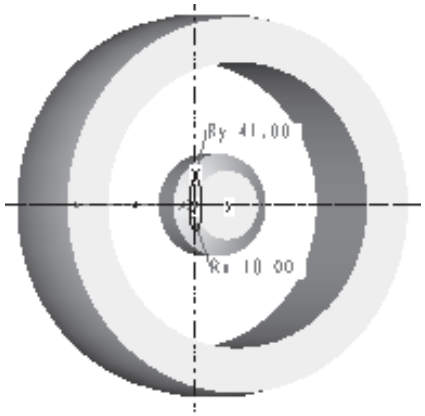


Figure 8-70 Sketch for section 1 with dimensions

The dimensions for this section are shown in Figure 8-66.

20. After the sketch is complete, choose the **Continue with the current section.** button to exit the sketcher environment.

21. Choose **OK** from the **PROTRUSION** dialog box. The default trimetric view of the swept blend feature with the base feature is shown in Figure 8-72.

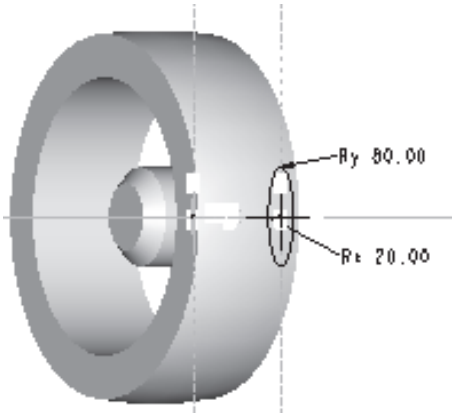


Figure 8-71 Sketch for section 2 with dimensions

Figure 8-72 Model after creating the swept blend

Creating the Rotational Pattern of the Swept Blend

Creating the swept blend features individually on the base feature will consume a lot of time. Therefore, you need to create a rotational pattern of the hole feature.

1. Choose **PART > Feature > Pattern** from the **Menu Manager**. You are prompted to select the feature to be patterned.

2. Select the swept blend feature from the graphics screen. The **PAT OPTIONS** submenu is displayed.

3. The **General** option is selected by default in the **PAT OPTIONS** submenu; choose **Done**.

4. The dimensions of the swept blend feature are displayed on the graphics screen. Select the angular dimension **90**. The **Message Input Window** is displayed.

Note

*You may need to zoom in the display of the model to select the **90** angular dimension value.*

5. Enter **120** in the **Message Input Window** and press ENTER. You have entered a value of 120 because there are three instances of the swept blend feature on a circular feature. You know that a circle makes an angle of 360-degree at the center; therefore when 360 is divided by 3 it is equal to 120.

6. Choose **Done** from the **EXIT** submenu. The **Message Input Window** is displayed and prompts you to specify the number of instances of the hole feature in the pattern.

7. Enter **3** in the **Message Input Window** and press ENTER. Choose **Done** from the **EXIT** submenu. The rotational pattern is created as shown in Figure 8-73.

Creating the Round Feature

The third feature is a round of radius 30.

1. Choose **Insert > Round** from the menu bar. The **ROUND TYPE** menu is displayed. The **Simple** option is selected by default; choose **Done** from this menu.

   The **RND SET ATTR** menu is displayed.

2. The **Constant > Edge Chain** options are selected by default. Choose **Done** from this menu. The **CHAIN** menu is displayed.

3. The **Tangnt Chain** option is selected by default and you are prompted to select an edge.

4. Select the edges shown in Figure 8-74 to round. You may need to spin the model to select the inner edges of the swept blend feature.

5. After selecting the edges, choose the **Done** option from the **CHAIN** menu. The **Message Input Window** is displayed and you are prompted to specify the radius of round.

6. Enter a value of **30** in the **Message Input Window** and press ENTER.

7. Choose **OK** from the **ROUND** dialog box.

   The round feature is completed. The default trimetric view of the round feature is shown

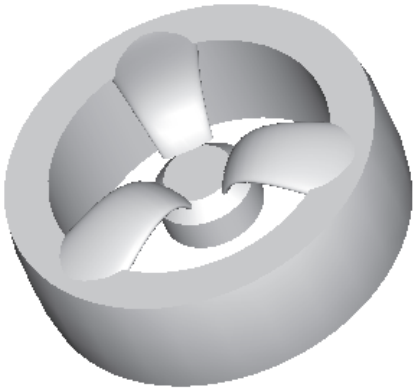


Figure 8-73 Model after creating pattern

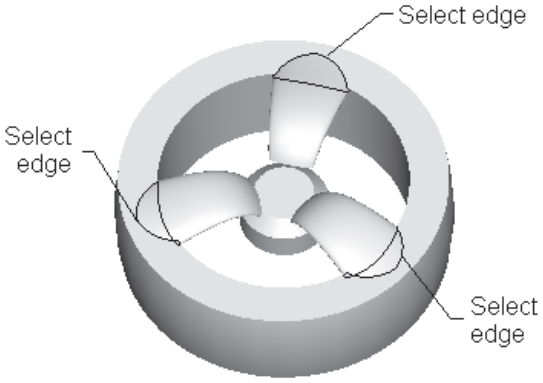


Figure 8-74 Edges to round

in Figure 8-75. Using the same options create a round feature of radius 10 highlighted in Figure 8-76.

Figure 8-75 Model after creating a round of radius 30

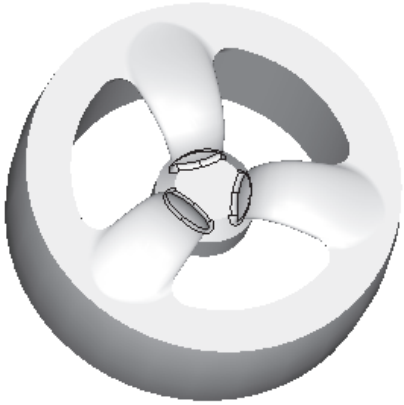


Figure 8-76 Model highlighting the round feature of radius 10

Creating the Revolve Cut Feature

You need to first select a sketching plane to draw the sketch of the cut feature and then specify the angle of revolution.

1. Choose **Insert > Cut > Revolve** from the menu bar. The **ATTRIBUTES** menu is displayed.

2. Choose **One Side > Done** from the **ATTRIBUTES** menu. You are prompted to select a sketching plane.

3. Select the **FRONT** datum plane. The **DIRECTION** submenu is displayed. Choose **Okay**. The **SKET VIEW** submenu is displayed.

4. Select the **Bottom** option and choose the **TOP** datum plane from the graphics screen.

5. Once you enter the sketcher environment, draw a center line aligned to the **RIGHT** datum plane. Draw the sketch of the revolved cut feature and add dimensions and constraints as shown in Figure 8-77.

6. After the sketch is complete, choose the **Continue with the current section**. button from the **Right Toolchest**.

   The **REV TO** menu is displayed and you are prompted to specify the angle of revolution.

7. Select the **360** option from the **REV TO** menu and choose **Done**.

8. Choose **OK** from the **CUT** dialog box. The revolve cut feature is shown in Figure 8-78.

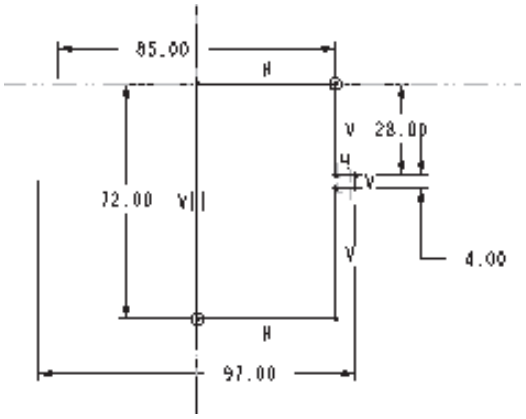


Figure 8-77 *Sketch for the revolve cut feature*

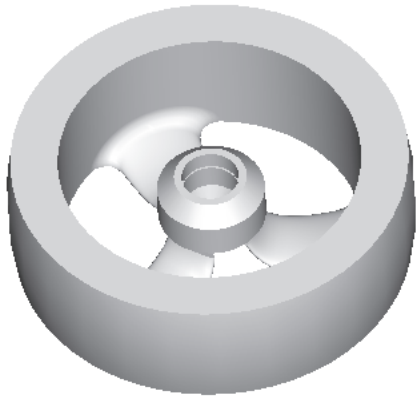


Figure 8-78 *Model after creating the revolve cut feature*

The model is completed and the isometric view of the model is shown in Figure 8-79.

Saving the Model

You have to save the model because you may need it later.

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 8-80. The feature id numbers displayed in the **Model Tree** may be different when you create the features.

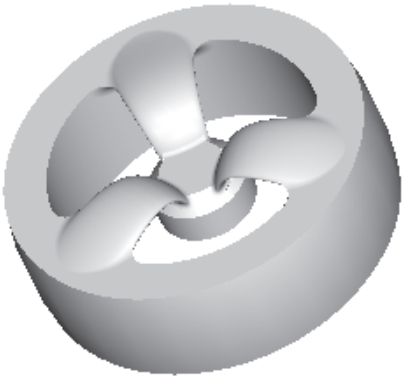


Figure 8-79 Isometric view of the model

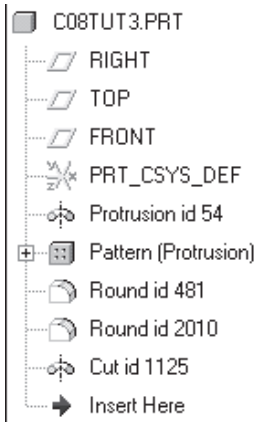


***Figure 8-80 Model Tree** for Tutorial 3*

Tutorial 4

In this tutorial you will create the spring shown in Figure 8-81. Figure 8-82 shows the front view with dimensions of the spring. **(Expected time: 20 min)**

***Figure 8-81** Isometric view of the spring*

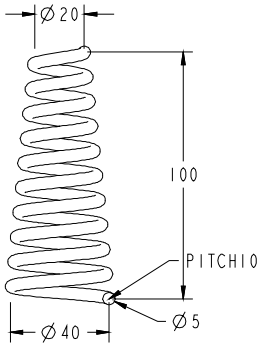


***Figure 8-82** Front view of the spring with dimensions*

The following steps outline the procedure for creating the constant pitch spring:

a. First examine the spring and then determine the specifications of the spring. The spring is right-handed, constant pitch, and is created by revolving through axis.

b. Select the sketching plane, draw the trajectory using the sketcher tools, and apply dimensions, see Figure 8-83.

c. Specify the pitch of the spring.

d. Draw the section of the spring using the sketcher tools and dimension it, see Figure 8-84.

After understanding the procedure for creating the model, you are now ready to draw 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 **c08** in the **Select Working Directory** dialog box.

Creating New Object File

1. Open a new part file and name it **c08tut4**. The three default datum planes and the **Model Tree** are displayed on the graphics screen. Exit the **Model Tree**. The **Model Tree** will not appear if it has been previously turned off.

Creating the Helical Sweep

The spring that you have to create is a right-handed spring of constant pitch.

1. Choose **Insert > Protrusion > Helical Sweep** from the menu bar. The **ATTRIBUTES** menu is displayed. The **Constant > Thru Axis > Right Handed** options are selected by default.

2. Choose **Done** from the **ATTRIBUTES** menu. You are prompted to select a sketching plane.

3. Select the **FRONT** datum plane from the graphics screen. The **DIRECTION** submenu is displayed. Choose **Okay**. The **SKET VIEW** submenu is displayed.

4. Select the **Top** option from the **SKET VIEW** submenu and choose the **TOP** datum plane.

5. Once you enter the sketcher environment, draw the sketch of the trajectory and dimension it as shown in Figure 8-83. As evident from Figure 8-83, the endpoint of the trajectory is aligned to the **TOP** datum plane.

   You need to draw a center line in the sketch about which the spring will be rotated. This is the axis of the spring and is the first trajectory that is drawn in the sketch.

6. After you complete the sketch of the trajectory, choose the **Continue with the current section.** button from the **Right Toolchest**. The **Message Input Window** is displayed and you are prompted to specify the pitch of the spring.

7. Enter **10** in the **Message Input Window** and press ENTER.

   Now, you enter the sketcher environment to draw the section of the spring. You will notice that 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.

8. Draw the section of the spring such that the center of the circle coincides with the intersection of the two perpendicular lines and dimension it as shown in Figure 8-84.

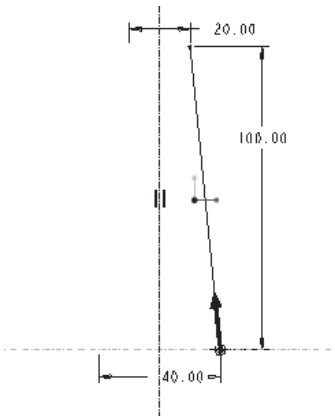


Figure 8-83 *Sketch of the trajectory with dimensions*

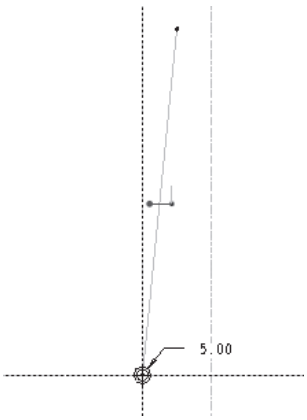


Figure 8-84 *Sketch of the section with dimensions*

9. After completing the sketch, choose the **Continue with the current section.** button to exit the sketcher environment.

10. Choose the **OK** button from the **PROTRUSION** dialog box.

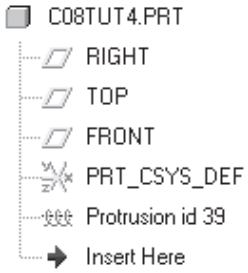
    The spring is created and is shown in Figure 8-85.

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 8-86.

Figure 8-85 *Isometric view of the spring*



8-86 Model Tree *for Tutorial 4*

Self Evaluation Test

Answer the following questions and then compare your answers with the answers given at the end of this chapter.

1. You can create Constant pitch as well as Variable pitch helical sweep features. (T/F)

2. The **Helical Sweep** option is used to create cut as well as the protrusion features. (T/F)

3. You can create a helical feature having 40 units pitch and 50 units as the diameter of the circle as the cross-section. (T/F)

4. The **Shoulder Crv** and the **Tangent Crv** options in the **BNDRS OPTS** submenu while using the **Boundaries** option are not available by default. (T/F)

5. The **Var Sec Sweep** option is available for both protrusion and cut options. (T/F)

6. While using the **Var Sec Sweep** option, the contour of the sweep feature depends on the shape of the _____.

7. While using the **Var Sec Sweep** option, when you draw the section for the sweep feature, the section should be _____ to the endpoints of the trajectories.

8. When you choose the **Done** option from the **SWEEP OPTS** submenu the _____ menu is displayed.

9. When you use the **Norm To Trj** option to create the variable sweep you are prompted to define an origin trajectory, a normal trajectory, and a _____.

10. The **Swept Blend** option is used to create a model that is a combination of _____ and _____.

Review Questions

Answer the following questions:

1. Which of the following menus is displayed when you choose the **Helical Sweep** option from the **Insert** menu in the menu bar?

(a) **ADV FEAT OPT** (b) **ATTRIBUTES**
(c) **SWEEP OPTS** (d) None

2. Which of the following options is used to create a helical feature with constant pitch throughout the sweep?

(a) **Right Handed** (b) **Constant**
(c) **Thru Axis** (d) None

3. Which of the following options in the **SWEEP OPTS** is used to create a sweep in which the swept section is normal to the trajectory defined as the origin trajectory?

(a) **Pivot Dir** (b) **NrmToOriginTraj**
(c) **Norm To Traj** (d) None

4. Which of the following options in the **RELATIONS** submenu is used to delete relations?

(a) **Add** (b) **Edit Rel**
(c) **Show Dim** (d) **Show Rel**

5. Which of the following options is used to create a variable section sweep in which you need to select a plane, edge, curve, or axis to which the section to be swept is perpendicular?

(a) **Pivot Dir** (b) **Nrm To Traj**
(c) **NrmToOriginTraj** (d) None

6. The **Var Sec Swp** option is used to create a sweep in which the section of the sweep is constant throughout the sweep. (T/F)

7. The trajectory called X trajectory is used to guide the section of the sweep feature along it. (T/F)

8. When you choose the **Edit Rel** option the relations are not checked for errors. (T/F)

9. The **Switch Dim** option of the **RELATIONS** submenu is used to toggle the dimension value mode and the symbols. (T/F)

10. **Helical Sweep** option is also used to create a cut. (T/F)

Exercise

Exercise 1

Create the model shown in Figure 8-87. The front view is shown in Figure 8-88.

(Expected time: 20 min)

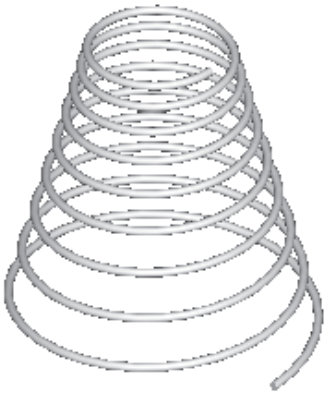


Figure 8-87 *Isometric view of the spring*

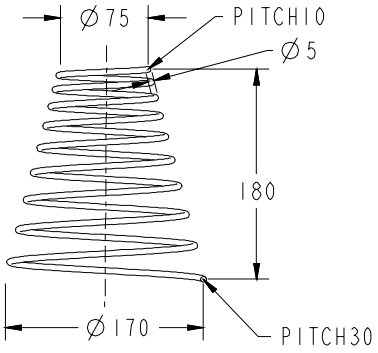


Figure 8-88 *Front view of the spring*

Answers to the Self-Evaluation Test

1 - T, **2** - T, **3** - F, **4** - T, **5** - T, **6** - X trajectory, **7**- aligned, **8** - **VAR SEC SWP**, **9** - section, **10** - sweep, blend.