Chapter 9

Advanced Modeling Tools-III

Learning Objectives

After completing this chapter you will be able to:

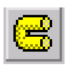
- *Create Sweep Feature.*
- *Create Loft Feature.*
- *Create 3D Sketches.*
- *Edit 3D Sketches.*
- *Create Curves.*
- *Create Draft Feature.*

ADVANCED MODELING TOOLS

Some of the advanced modeling tools were discussed in earlier chapters. The remaining advanced modeling tool are discussed in this chapter. The advanced modeling tools that are discussed in this chapter include sweep, loft, draft, extruding the text, curves, 3D sketches, and so on.

Creating the Sweep Feature

Toolbar:	Features > Sweep
Menu:	Insert > Bose/Base > Sweep



One of the most important advanced modeling tool is the **Sweep** tool. This tool is used to extrude a closed profile along an open or closed path. Therefore, to create a sweep feature you need at least two sketches. The first sketch is the section for the sweep feature and the second section is the path along which the section will be swept. An example of the sketches for creating the sweep feature is shown in Figure 9-1. Choose the **Sweep** button from the **Features** toolbar to invoke the **Sweep PropertyManager**. You can also invoke this tool by choosing **Insert** > **Bose/Base** > **Sweep** from the menu bar. The **Sweep PropertyManager** is shown in Figure 9-2.

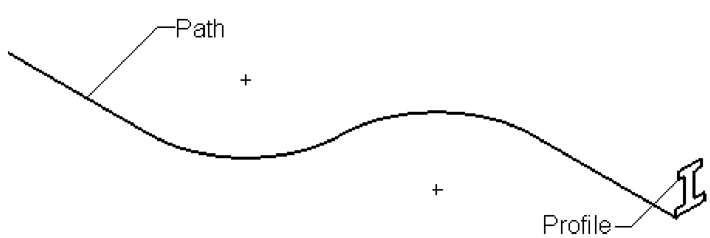


Figure 9-1 *Sketches to create a sweep feature*

After invoking the **Sweep PropertyManager**, you are prompted to select the sweep profile. Select from the drawing area the sketch that is created as the profile for the sweep feature. As soon as you select the sketch, the sketch is highlighted in green and the profile callout is displayed. Now, you are prompted to select the path for the sweep feature. Select the sketch that is created as the path of the sweep feature. When you select the sketch, it is highlighted in red and the path callout is displayed in the drawing area. The sweep feature is displayed in temporary graphics in the drawing area. Choose the **OK** button from the **Sweep PropertyManager** to end feature creation. Figure 9-3 shows a sweep feature.

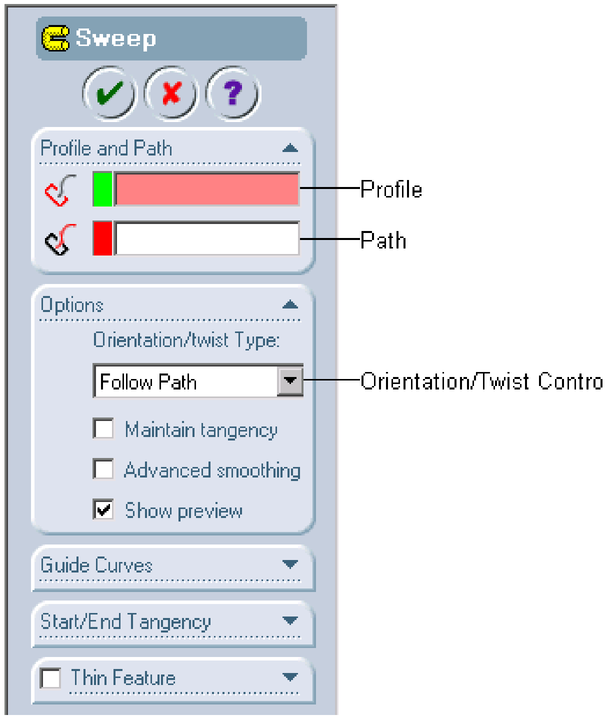


Figure 9-2** The **Sweep PropertyManager

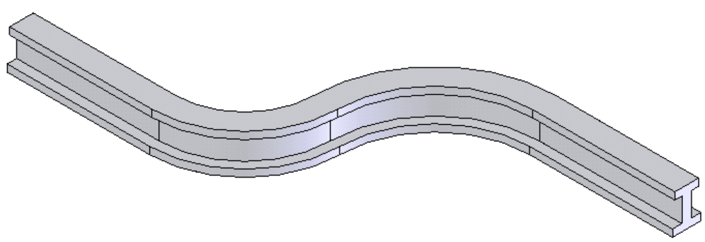


***Figure 9-3** Sweep feature*

It is not necessary that the sketch created for the profile of the sweep feature intersect the sketch created for the path of the sweep feature. However, the plane on which the profile is drawn should lie at one of the endpoints of the path. Figure 9-4 shows the nonintersecting sketches of profile and path. Figure 9-5 shows the resultant sweep feature. Figure 9-6 shows the sketch of the profile and the closed path. Figure 9-7 shows the resultant sweep feature.

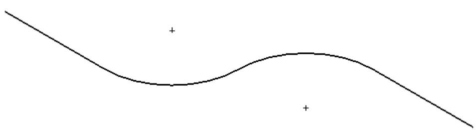


Figure 9-4 Nonintersecting sketches of profile and path

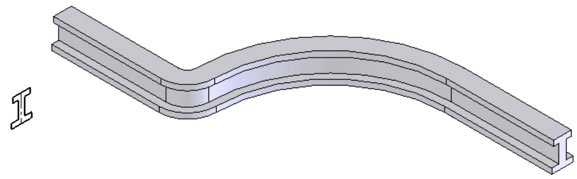


Figure 9-5 The resultant sweep feature

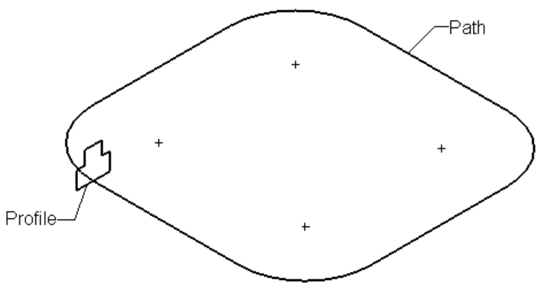


Figure 9-6 Sketch of the profile and the closed path

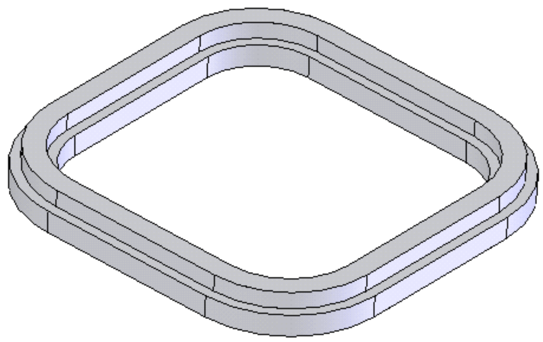


Figure 9-7 The resultant sweep feature

The various other options available in the **Sweep PropertyManager** to create advanced sweep features are discussed next.

Sweep Using the Follow Path and Keep Normal Constant options

When you are creating a sweep feature, by default, the **Follow Path** option is selected in the **Orientation/Twist Control** drop-down list available in the **Options** rollout. When you create a sweep feature using this option, the section will follow the path to create the sweep feature. If you select the **Keep normal constant** option from the **Orientation/Twist Control** drop-down list, the section will be swept along the path with a normal constraint. The section will not change its orientation along the sweep path. Therefore, the starting face and the end face of the sweep feature will be parallel. Figure 9-8 shows the sketches of the path and profile for the sketch feature. Figure 9-9 shows the sweep feature created using the **Follow Path** option. Figure 9-10 shows the sweep feature created using the **Keep normal constant** option. The other options available in the **Orientation/Twist Control** drop-down list are discussed later in this chapter.

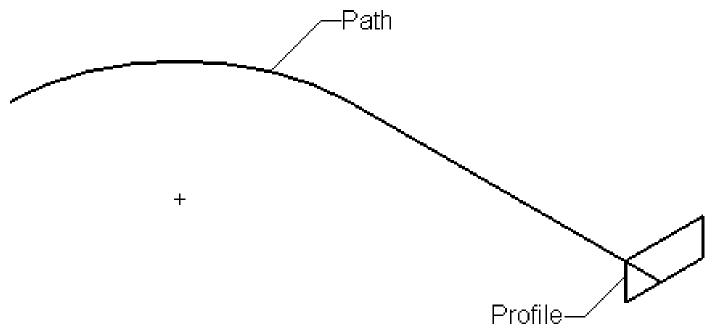


Figure 9-8 Sketches for the sweep feature

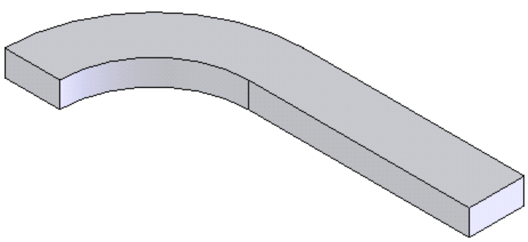


*Figure 9-7 Sweep feature with the **Follow Path** option selected from the **Orientation/Twist Control** drop-down list*

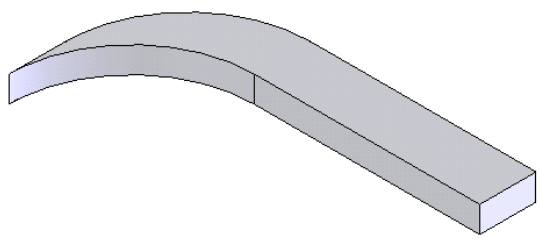
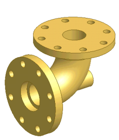


*Figure 9-10 Sweep feature with the **Keep normal constant** option selected from the **Orientation/Twist Control** drop-down list*

Maintain Tangency

The **Maintain Tangency** check box is available in the **Orientation/Twist Type** area of the **Options** rollout. This option is used when the sweep section has tangent entities and you want the corresponding surfaces to be tangent in the resultant sweep feature.



Tip. *The model edges can also be selected as a path for creating the sweep feature. When you select a model edge as the sweep path, the **Tangent Propagation** check box is displayed in the **Options** rollout. If this check box is selected, the edges tangent to the selected edge are selected automatically as the path of the sweep feature.*

Advanced Smoothing

The **Advanced smoothing** check box is available in the **Orientation/Twist Type** area of the **Options** rollout. This option is used if the sweep section has circular or elliptical arcs and you want a smooth surface to be created in the sweep feature.

Show Preview

The **Show preview** check box available in the **Options** rollout is used to display the preview of the sweep feature in the drawing area. This check box is selected by default. If you clear this check box, the preview of the sweep feature will not be displayed in the drawing area.

Merge Results

The **Merge results** check box is available only when you have at least one feature in the current document. This check box is selected by default. If you clear this check box, it will result in creating the sweep feature as a separate body.

Align with End Faces

The **Align with end faces** option is available in the **Options** rollout only when at least one feature has already been created in the current document. When this option is selected, the sweep feature is extended or trimmed to align with end faces. Figure 9-11 shows the profile and path for creating the sweep feature. Figure 9-12 shows the resultant sweep feature created with the **Align with end faces** check box cleared. Figure 9-13 shows the resultant sweep feature created with the **Align with end faces** check box selected.

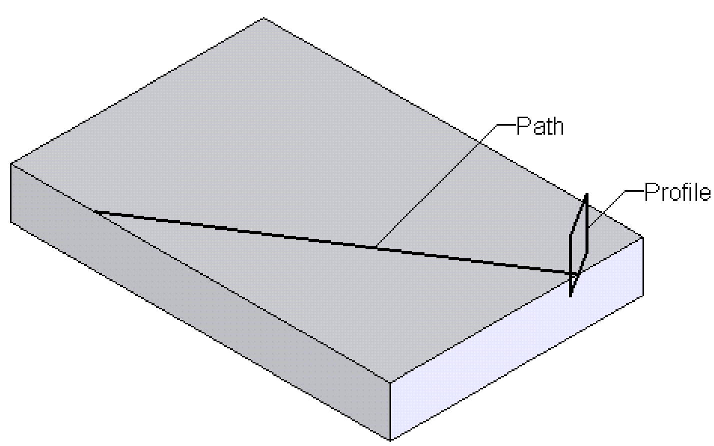


Figure 9-11 *Sketches for the sweep feature*

Note
If the sweep feature does not merge, you need to reduce the size of the profile.

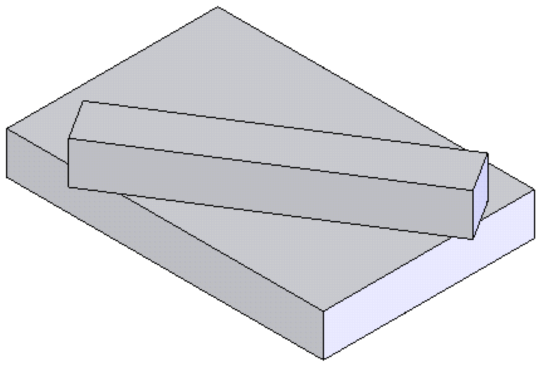


Figure 9-12 *Resultant sweep feature with the* ***Align with end faces*** *check box cleared*

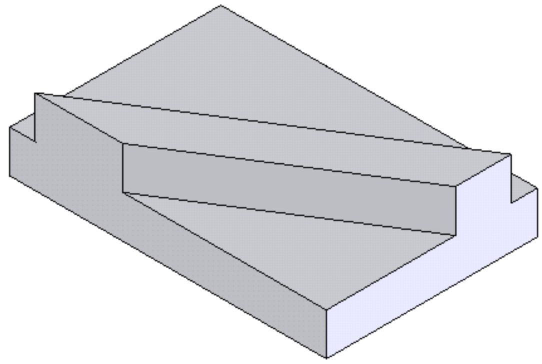


Figure 9-13 *Resultant sweep feature with the* ***Align with end faces*** *check box selected*

Sweep with Guide Curves

The sweep using guide curves is the most important option in the Advanced Modeling tools. In this sweep feature, the section of the sweep profile varies according to the guide curves along the sweep path. To create this type of feature, you need to create the sketch of the profile, path, and the guide curves. A **Pierce** relation must be applied between the guide curves and the profile of the sweep feature. The **Pierce** relation allows the profile to change shape and size along the sweep path. After creating the sketch of profile, path, and guide curves, invoke the **Sweep PropertyManager**. Select the sketches of the profile and the path; the preview of the sweep feature is displayed in the drawing area. Click on the black arrow on the right of the **Guide Curves** rollout to open this rollout. The **Guide Curves** rollout is shown in Figure 9-14.

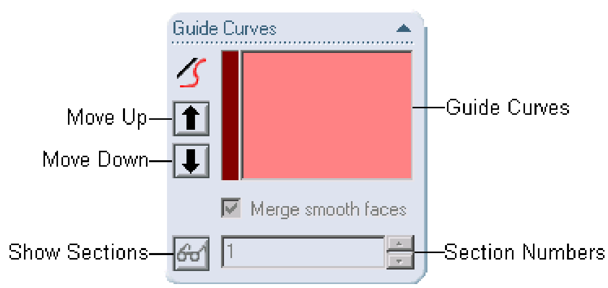


Figure 9-14 *The* ***Guide Curves PropertyManager***

Select the sketch of the guide curve; the selected guide curve is displayed in brown and a **Guide Curve** callout is also displayed attached to the guide curve. The preview of the sweep feature is also displayed in the drawing area in temporary graphics. Choose the **OK** button from the **Sweep PropertyManager**. Figure 9-15 shows the sketch for the sweep feature with guide curve. Figure 9-16 shows the resultant sweep feature creation.

In the previous case the path of the sweep feature is a straight line and the guide curve is an

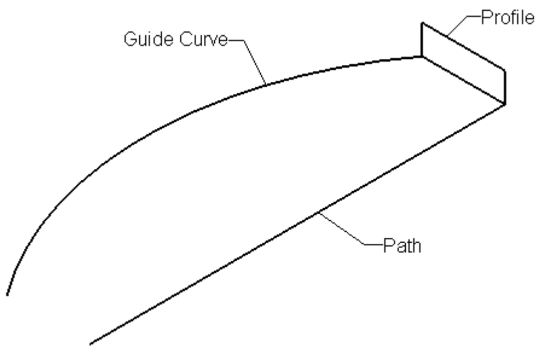


Figure 9-15 *Sketches for the sweep feature with guide curves*

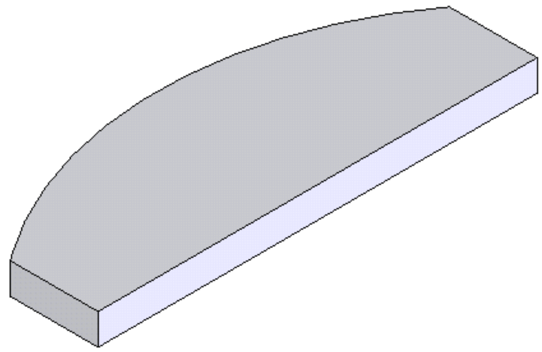


Figure 9-16 *Resultant sweep feature*

arc. In the next case, the arc is selected as the path of the sweep feature and the straight line is selected as the guide curve. Figure 9-17 shows the sketches for the sweep feature. Figure 9-18 shows the resultant sweep feature.

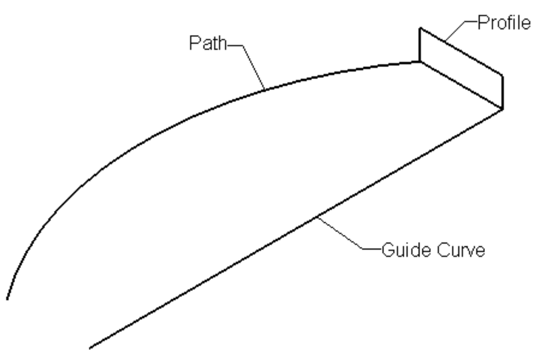


Figure 9-17 *Sketches for the sweep feature*

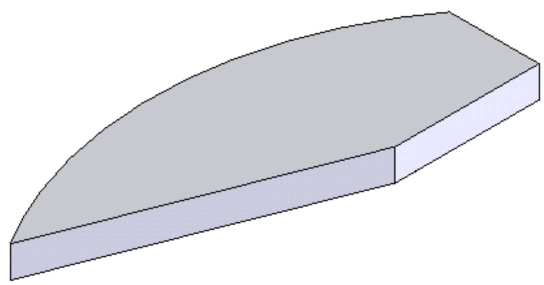


Figure 9-18 *Resultant sweep feature*

Move Up and Move Down

The **Move Up** button and the **Move Down** button available on the left of the **Guide Curves** display area are used to change the sequence of the selected guide curve.

Merge smooth faces

The **Merge smooth faces** check box available in the **Guide Curves** rollout is selected by default. This option is used to merge all the smooth faces together, resulting in a smooth sweep feature. When you clear this check box, the **Sweep Preview Warning** dialog box is displayed as shown in Figure 9-19. In this dialog box you are prompted that the feature you are creating may fail because of change in smooth face option. Choose **Yes** from this dialog box if you want to accept the change option. When you create a sweep feature with guide curves and this option cleared, the resulting feature do not merge the smooth faces

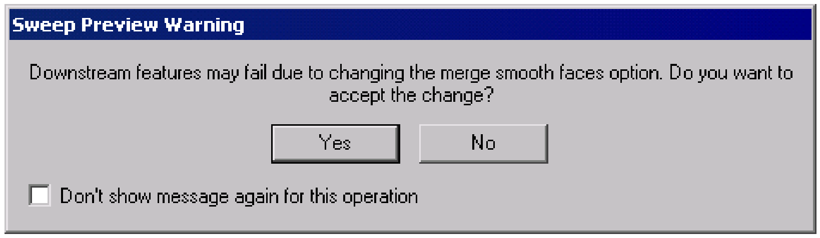


*Figure 9-19 The **Sweep Preview Warning** dialog box*

together. This results in a sweep feature with noncontinuous curvature surface. Figure 9-20 shows a sweep feature created with the **Merge smooth faces** check box selected. Figure 9-21 shows the same sweep feature with the **Merge smooth faces** check box cleared.

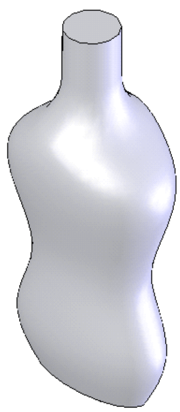


*Figure 9-20 Sweep feature with the **Merge smooth faces** check box selected*

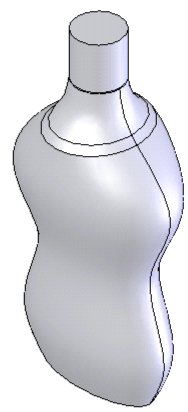
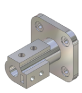


*Figure 9-21 Sweep feature with the **Merge smooth faces** check box cleared*



Note

*Remember that the if you create the sweep feature with the **Merge smooth faces** check box cleared, the resultant feature will be generated faster and the adjacent faces and edges are easily merged. Also the lines and arcs in the guide curve match accurately while creating the sweep feature.*

Show Sections

The **Show Sections** button available in the **Guide Curves** rollout is used to display the intermediate sections while creating the sweep feature with guide curves. To display the intermediate profiles or sections along the sweep path, choose the **Show Section** button from the **Guide Curves** rollout. The **Section Number** spinner is invoked. Using this spinner you can view the sections of the profile along the sweep path. Figure 9-22 shows a section being displayed using the **Show Section** tool along the sweep path.

Sweep Feature Using the Follow Path and 1st Guide Curve Option

When you create a sweep feature with guide curve using the **Follow path and 1st guide curve** option, the profile follows the path and the 1st guide curve to create the feature. For creating

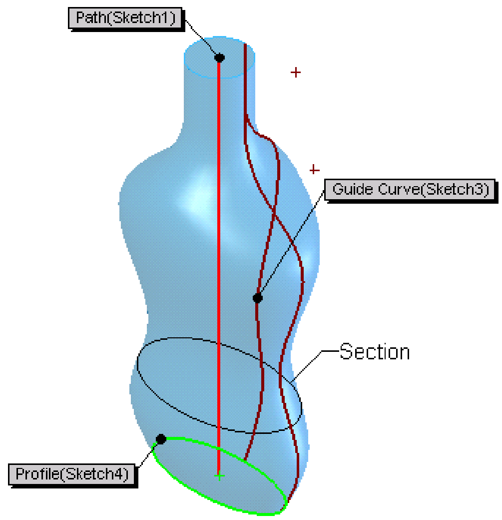


Figure 9-22 *Section being displayed using the* ***Show Section*** *tool*

a sweep feature using this option invoke the **Sweep PropertyManager** and select the profile, path, and guide curve(s). By default, the **Follow path** option is selected in the **Orientation/Twist Control** drop-down list. Select the **Follow path and 1st guide curve** option from the **Orientation/Twist Control** drop-down list. Choose the **OK** button from the **Sweep PropertyManager** to end feature creation.

Sweep Feature Using the Follow 1st and 2nd Guide Curves Option

Using this option, the profile of the sweep feature follows the 1st and 2nd guide curves to create the resultant sweep feature. For creating this type of sweep feature select the **Follow 1st and 2nd guide curves** option from the **Orientation/Twist Control** drop-down list. Choose the **OK** button from the **Sweep PropertyManager** to end feature creation.

Figure 9-23 shows the sketches of the profile, path, and guide curves for creating a sweep feature. Figure 9-24 shows the sweep feature created using the **Follow path and 1st guide curves** option. Figure 9-25 shows the sweep feature created using the **Follow 1st and 2nd guide curves** option.

Start/End Tangency

The **Start/End Tangency** rollout available in the **Sweep PropertyManager** is used to define the tangency conditions on the start and the end of the feature. This rollout is invoked by clicking one of the arrow provided on the right of the rollout. The **Start/End Tangency** rollout is displayed in Figure 9-26. The various options available in the **Start/End Tangency** rollout are discussed next.

Start tangency type

The **Start tangency type** drop-down list is used to specify the options to define the tangency at the start of the sweep feature. The various options available in this drop-down list are discussed next.

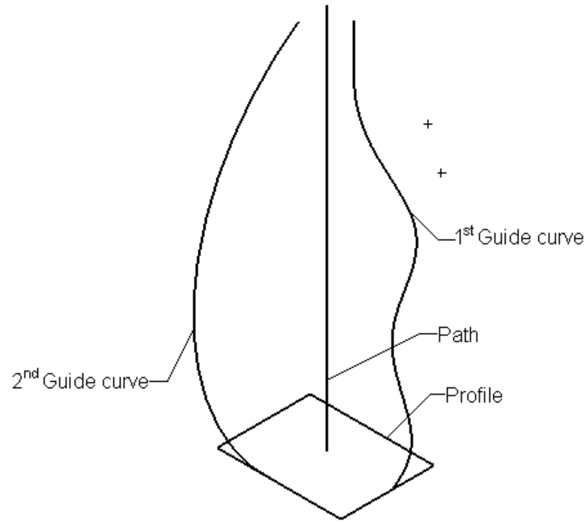


Figure 9-23 *Sketches or profile, path, and guide curves*

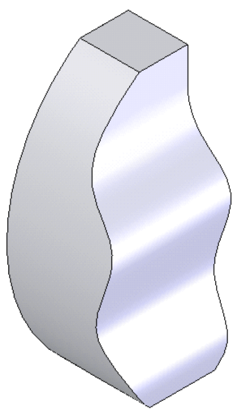


Figure 9-24 *Sweep feature using the* ***Follow path and 1st guide curve*** *option*

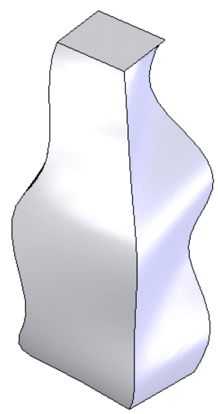


Figure 9-25 *Sweep feature using the* ***Follow 1st and 2nd guide curves*** *option*

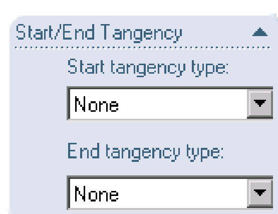


Figure 9-26 *The* ***Start/End Tangency*** *rollout*

None

The **None** option is selected by default and is used to create a sweep feature without applying any start tangency.

Path Tangent
The **Path Tangent** option is used to maintain the sweep feature normal to the path at the start.

Direction Vector
When you use the **Direction Vector** option, the starting of the sweep feature will be tangent to a virtual normal created from the selected entity. When you select this option a display area is also displayed and you need to select a linear edge, axis, planar face, or plane.

All Faces
The **All Faces** option is used to sweep the feature tangent to the adjoining faces of the existing geometry at the start. This option is available only when the sweep feature is attached to a surface or existing feature or geometry.

The options available in the **End tangency type** drop-down list are the same as those discussed above. The only difference is that the options in this drop-down list are applied to the end of the sweep feature. Figure 9-27 shows the sketches and the references to create the sweep feature with start and end tangency using the **Direction Vector** option. Figure 9-28 shows the resultant sweep feature.

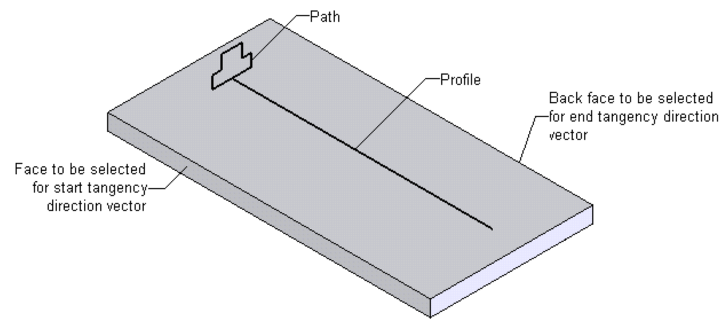


Figure 9-27 *Profile, path, and references for tangency using the* ***Direction Vector*** *option*

Creating a Thin Sweep Feature

You can also create a thin sweep feature by specifying the thickness using the **Thin Feature** rollout. This rollout is invoked by selecting the check box provided at the left of the **Thin Features** rollout. The **Thin Features** rollout is shown in Figure 9-29. The options available in this rollout are the same as those discussed in the earlier chapters in which extruding and revolving thin features have been discussed. Figure 9-30 shows a thin sweep feature.

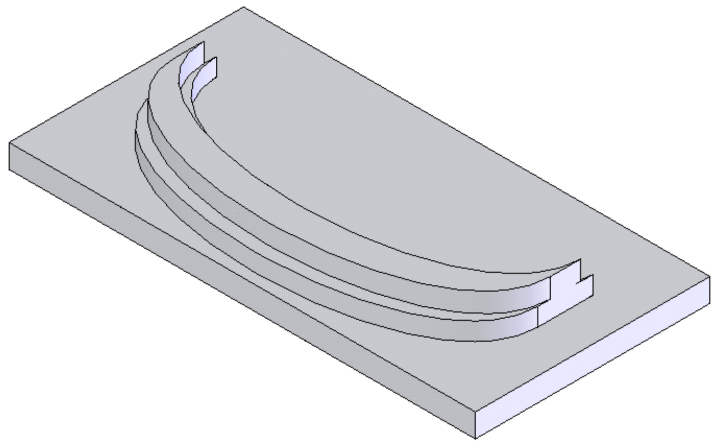


Figure 9-28 *Resultant sweep feature*

Figure 9-29 *The* ***Thin Feature*** *rollout*

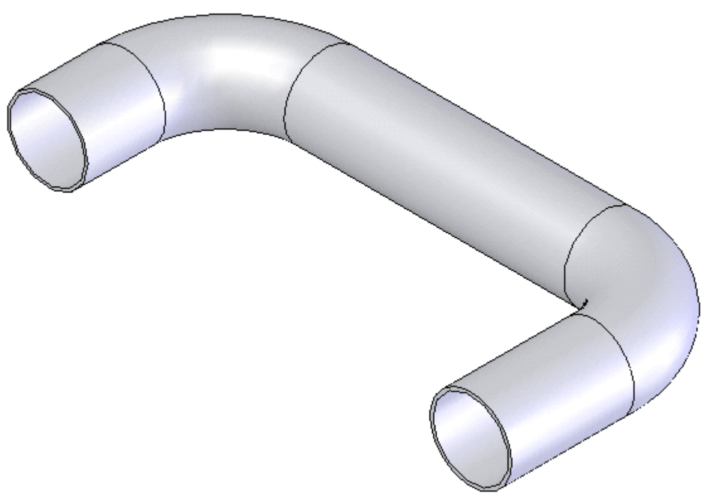
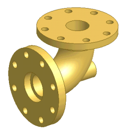


Figure 9-30 *Thin sweep feature*



Tip. *As discussed earlier, the **Pierce** relation should be applied between the profile and the guide curves. If the profile consists of a line or an arc then the coincident relation will also serve the purpose of the **Pierce** relation. If the profile is created using spline or ellipse then you have to apply the **Pierce** relation.*

Creating Cut Sweep Features

Menu: Insert > Cut > Sweep

You can also remove material from an existing feature or model using the sweep feature. To create a cut-sweep feature, choose **Insert > Cut > Sweep** from the menu bar to invoke the **Cut-Sweep PropertyManager** as shown in Figure 9-31. The options available in the **Cut-Sweep PropertyManager** are the same as those discussed in the **Sweep PropertyManager** with the only difference being that it is meant for cut operation. Figure 9-32 shows the sketched profile and path. Figure 9-33 shows the resultant sweep feature created using the **Cut-Sweep PropertyManager**.

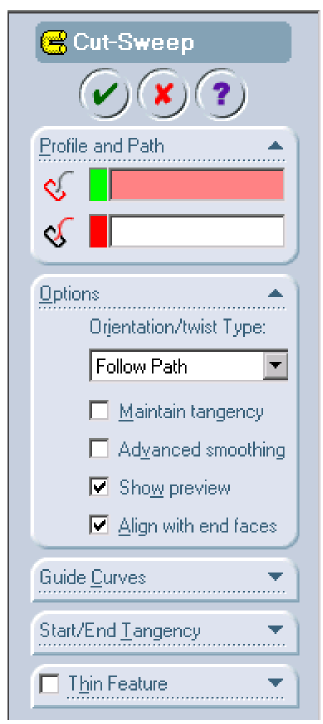


Figure 9-31 *The **Cut-Sweep PropertyManager***

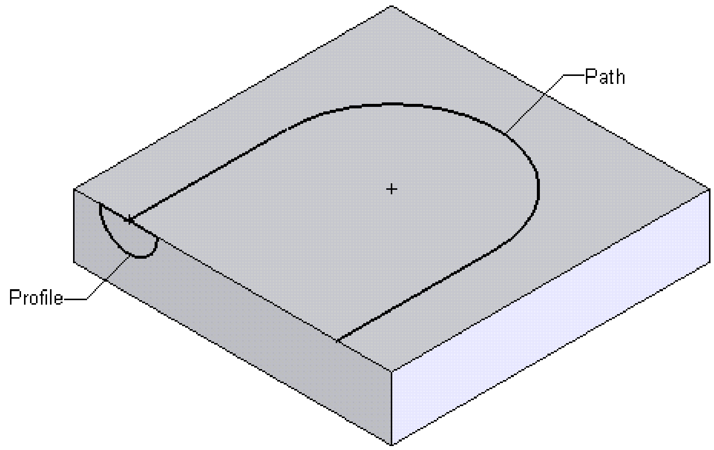


Figure 9-32 *Sketches for the cut sweep feature*

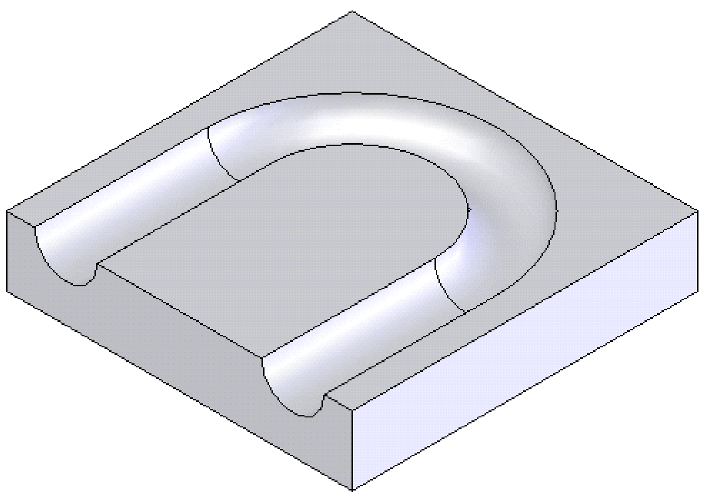


Figure 9-33 *Resultant cut sweep feature*

Creating the Loft Feature

Toolbar:	Features > Loft
Menu:	Insert > Bose/Base > Loft

The lofted features are created by blending more than one similar or dissimilar geometries together to get a free form type of shape. These similar or dissimilar geometries may or may not be parallel to each other. The sketches for lofts should be closed sketches.

In SolidWorks, the loft features are created using the **Loft PropertyManager**. The **Loft PropertyManager** is invoked by choosing the **Loft** button from the **Features** toolbar or by choosing **Insert** > **Boss/Base** > **Loft** from the menu bar. The **Loft PropertyManager** is shown in Figure 9-34.

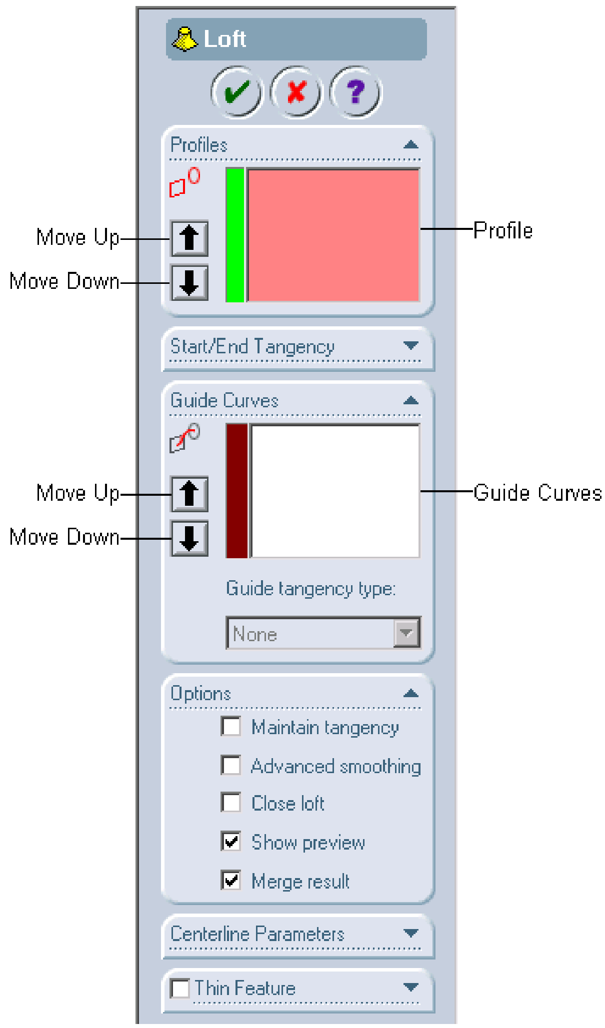


Figure 9-34 *The* ***Loft PropertyManager***

After creating the sketches when you invoke the **Loft PropertyManager**, you are prompted to select at least two profiles. Select the profiles from the drawing area. As you select the profiles, the preview of the loft feature is displayed in the drawing area in temporary graphics. Choose the **OK** button from the **Loft PropertyManager** to end feature creation.

Note

The geometry and the shape of the loft feature depends on the sequence of selection and the selection point of the sketches.

Figures 9-35 and 9-37 show the sequence and the selection point for selecting the sections to create a loft feature. Figures 9-36 and 9-38 show the resultant loft features.

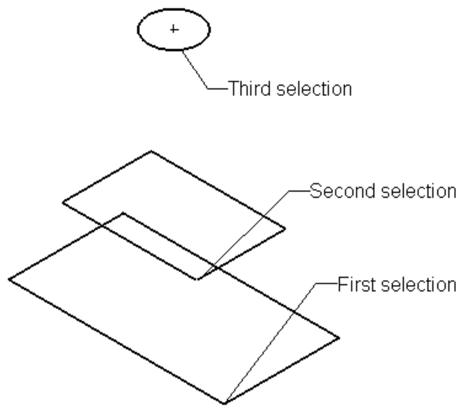


Figure 9-35 *Three sketches for the loft feature*

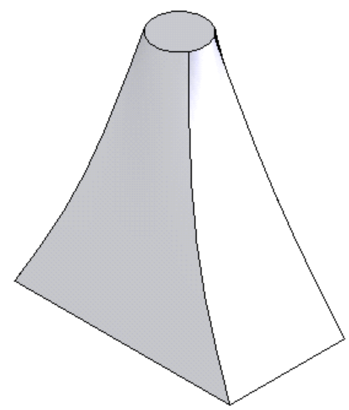


Figure 9-36 *Resultant loft feature*

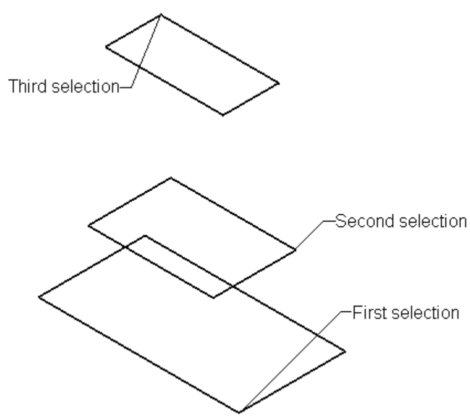


Figure 9-37 *Sketches for the loft feature*

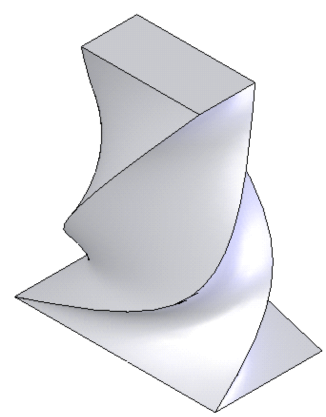


Figure 9-38 *Resultant loft feature*

Start/End Tangency

The **Start/End Tangency** rollout available in the **Loft PropertyManager** is used to define the tangency at the start and end sections of the loft feature. This rollout is invoked by clicking once on the black arrow provided at the left of this rollout. By default, the **None** option is selected. This means that tangency is not applied to the loft feature. The other options available in this rollout are discussed next.

Normal to Profile

The **Normal to Profile** option is used to define the tangency normal to profile. When you invoke this option, you are provided with a spinner to specify the length of tangent. A

Reverse Direction button is also provided to flip the direction of tangent. The **Start/End Tangency** rollout with **Normal to Profile** options selected in the **Start Tangency Type** and the **End Tangency Type** drop-down list is shown in Figure 9-39.

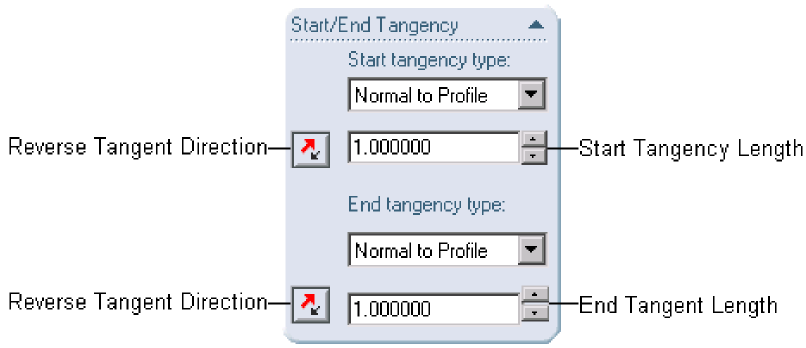


Figure 9-39 The ***Start/End Tangency*** *rollout with the* ***Normal to Profile*** *option selected*

Direction Vector

The **Direction Vector** option is used to define the tangency at the start and at the end of the loft feature by defining a direction vector. When you invoke this option you are provided with the **Direction Vector** display area and the spinners to define the length of tangents. You need to select the direction vectors to specify the tangent at the start and at the end of the loft feature. You can also specify the length of the tangents using the spinners provided in the **Start/End Tangent** rollout. The **Start/End Tangent** rollout with the **Direction Vector** option selected is displayed in Figure 9-40.

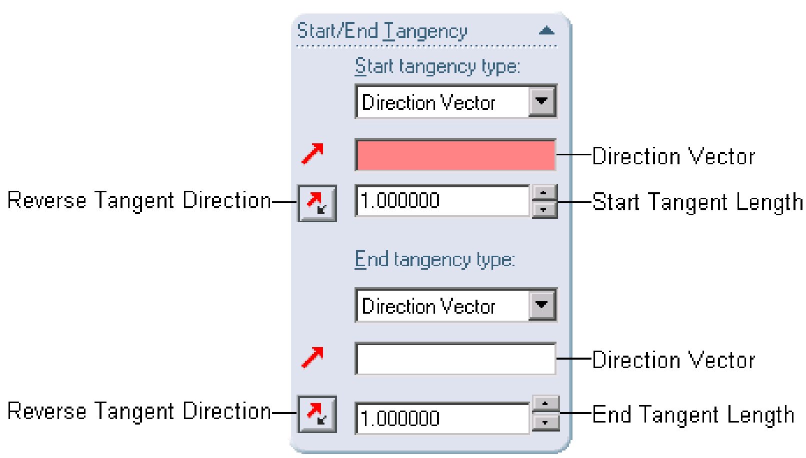


Figure 9-40 The ***Start/End Tangency*** *rollout with the* ***Direction Vector*** *option selected*

Figure 9-41 shows the section for the loft feature. Figure 9-42 shows the initial preview of the loft feature. Figure 9-43 shows the preview of the loft feature with tangent at the start of the loft. Figure 9-44 shows the tangent at the start and at the end of the loft. Figure 9-45 shows the final loft feature.

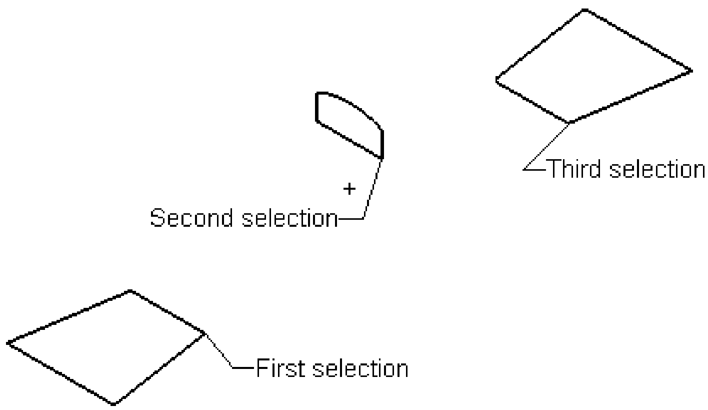


Figure 9-41 *Sections, selection points, and sequence of selection*

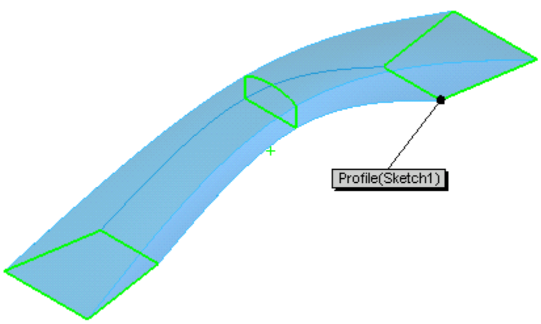


Figure 9-42 *Preview of the loft feature*

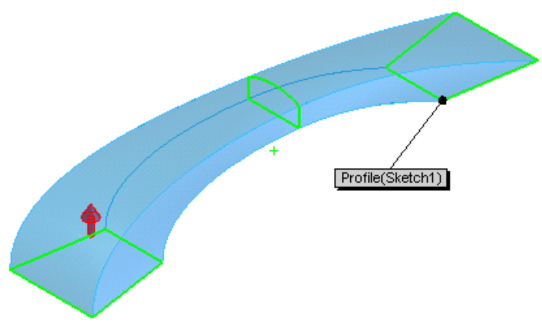


Figure 9-43 *Tangent applied at the start*

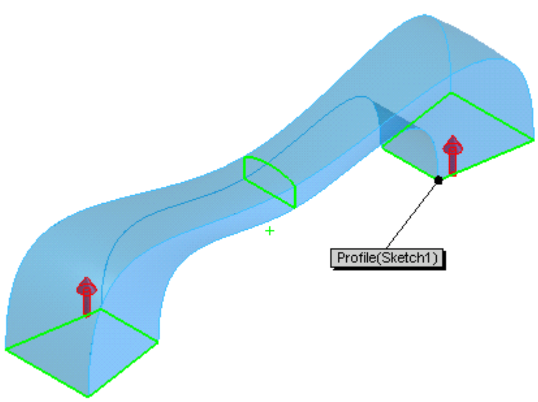


Figure 9-44 *Tangent applied at start and end*

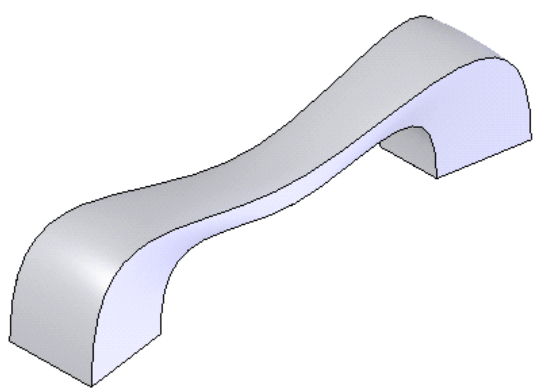
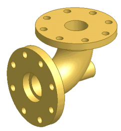


Figure 9-45 *The final loft feature*



Tip. *You can also define the tangent length dynamically by dragging the red arrows provided at the start and at the end of the loft feature. As you drag the arrow the preview of the tangent and the value in the spinner modify dynamically.*

Guide Curves

You can also define the guide curve between the profiles of the loft feature to define the path of transition of the loft feature. The sketches created for the guide curve must have a pierce relation with the sketches that define the loft section. All the other options available in the **Guide Curves** rollout are the same as those discussed earlier. The **Guide Curves** rollout is displayed in Figure 9-46.

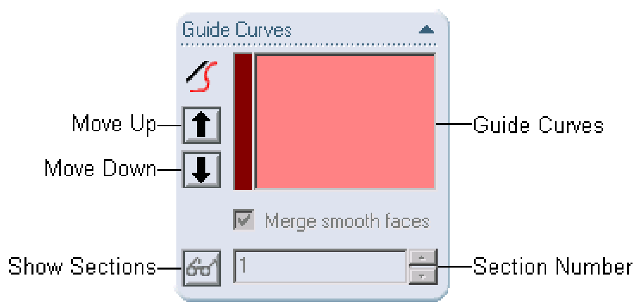


Figure 9-46 *The* ***Guide Curves*** *rollout*

Figure 9-47 shows the profiles and the guide curves for creating a loft feature with guide curves. Figure 9-48 shows the resultant loft feature created using the guide curves.

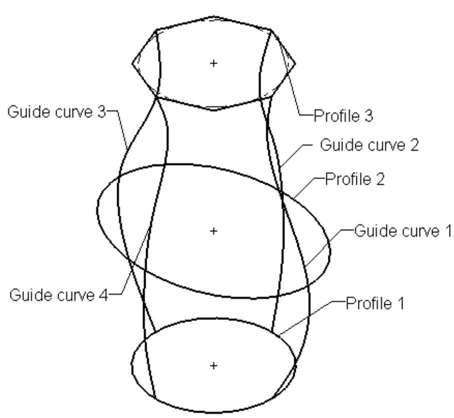


Figure 9-47 *Profiles and guide curves*

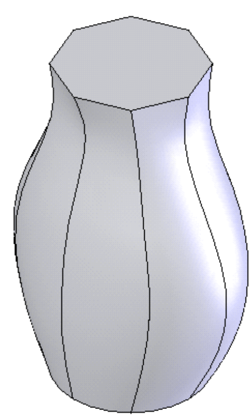


Figure 9-48 *Resultant loft feature*

Options

The **Options** rollout of the **Loft PropertyManager** is provided with many options to improve the creation of the loft feature. All the options available in this rollout are the same as those discussed earlier while discussing the sweep option. An additional option provided in this rollout is discussed next.

Close Loft

The **Close loft** option is used to created a close loft feature. A closed loft feature is created by joining the end section with the start section of the loft feature. Figure 9-49 shows a loft feature created with the **Close loft** check box cleared. Figure 9-50 shows the loft feature created with the **Close loft** check box selected.

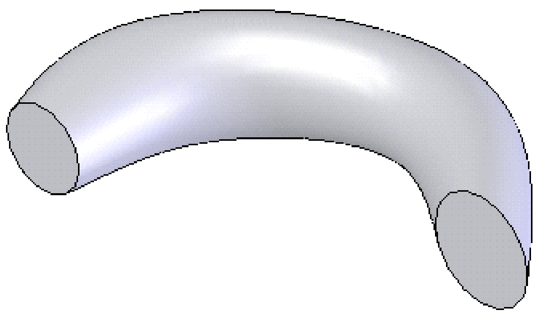


*Figure 9-49 Loft feature with the **Close Loft** check box cleared*

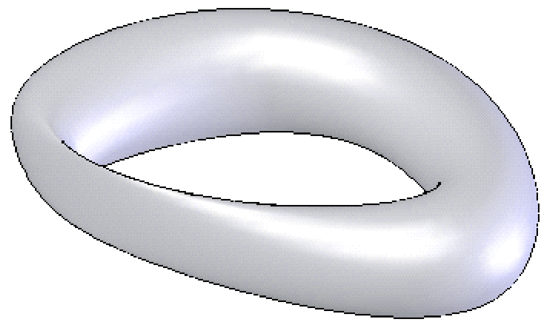


*Figure 9-50 Loft feature with the **Close loft** check box selected*

Centerline Parameters

The **Centerline Parameters** rollout is used to create a loft feature by blending two or more than two sections along a specified path. The path that specifies the transition is called centerline. You can invoke this rollout by clicking once on the arrow provided at the right of this rollout. The **Centerline Parameters** rollout is displayed in Figure 9-51.
The various options available in this rollout are discussed next.

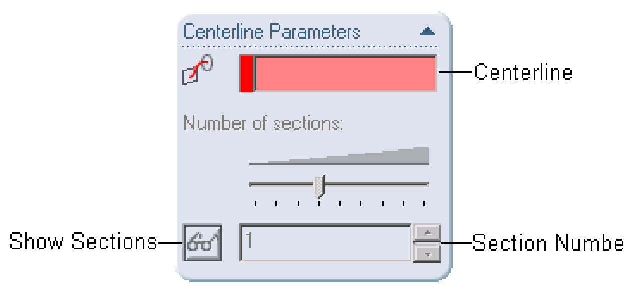


*Figure 9-51 The **Centerline Parameters** rollout*

Centerline

After invoking the **Centerline Parameters** rollout you need to define the centerline. Therefore, select the sketch that defines the centerline for the loft feature. The name of the sketch will be displayed in the **Centerline** display area.

Number of sections

The **Number of sections** slider bar provided in the **Centerline Parameters** rollout is used to define the number of intermediate sections. These intermediate sections define the accuracy and the smoothness of the surfaces generated to create the loft feature.

The **Show Sections** button and the **Section Number** spinner available in the **Centerline Parameters** rollout are used to display the intermediate sections as discussed earlier. Figure 9-52 shows the sketches of the profiles and the centerline used to create the loft feature. Figure 9-53 shows the resultant loft feature.

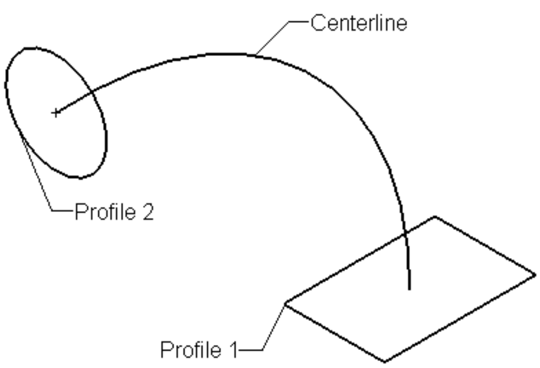


Figure 9-52 *Sketches of profiles and the centerline*

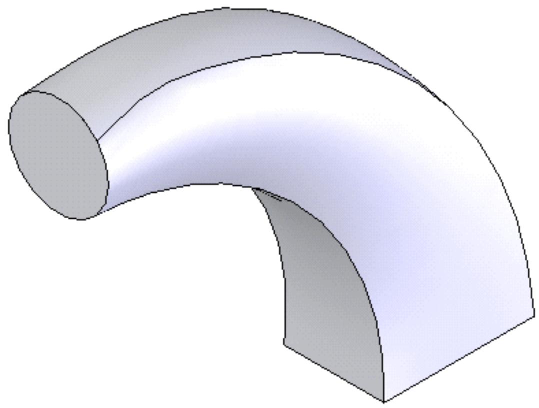
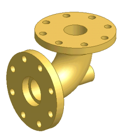


Figure 9-53 *The resultant loft feature*



Tip. *Create a loft feature such that one profile is created using the circle and the other is created using a polygon. Since, circles and ellipses do not have any endpoint and the polygons do have endpoints, the loft feature will blend the polygon and circle or ellipse approximately. But for better accuracy for blending you can split the sketch using the **Split Curve** option. After splitting the circle or ellipse, you can create a more accurate loft feature.*

You can also create a thin loft feature by defining the thin parameters using the **Thin Feature** rollout. Figure 9-54 shows a thin loft feature created using the **Thin Features** rollout available in the **Loft PropertyManager**.

The **Cut Loft PropertyManager** is used to create a cut loft feature. This option is invoked by choosing **Insert** > **Cut** > **Loft** from the menu bar.

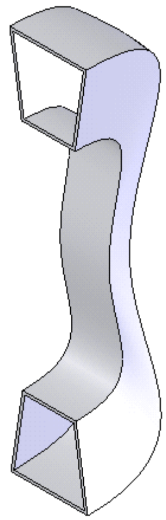
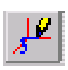


Figure 9-54 A thin loft feature

Creating 3D Sketches

Toolbar:	Sketch > 3DSketch
Menu:	Insert > 3D Sketch



In previous chapters you have learned how to create 2D sketches. In this chapter you will learn how to create 3D sketches. The 3D sketches are used to create 3D paths for the sweep features, 3D curves, and so on. Figure 9-55 shows a chair frame created by sweeping a profile along a 3D path.

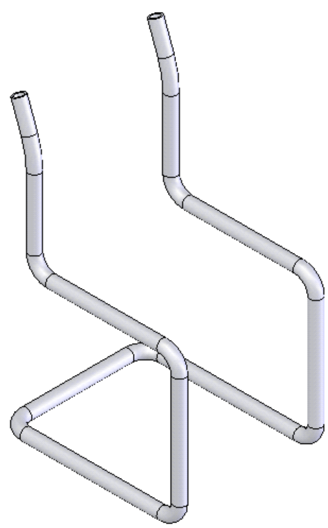
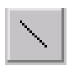


Figure 9-55 A chair frame created by sweeping a profile along a 3D path

To create a 3D sketch choose the **3D Sketch** button from the **Sketch** toolbar or choose **Insert > 3D Sketch** from the menu bar. When you choose this option, the 3D sketching environment is invoked and the origin is displayed in red color. For creating a 3D sketch, you do not need to select a sketching plane. When you invoke the 3D sketching environment some of the sketching tools are activated in the **Sketch Tools** toolbar. You can use only some of the sketching tools in the 3D sketching environment. The sketching tools that can be used in the 3D sketching environment are discussed next.

Line



For creating lines in the 3D sketching environment, first you have to orient the drawing area in the isometric view. Choose the **Isometric** button from the **Standard Views** toolbar to orient the drawing area to isometric. Now, choose the **Line** button from the **Sketching Tools** toolbar to invoke the line tool. The select cursor will be replaced by the line cursor with **XY** displayed at the bottom of the line cursor. This means that the sketch will be created in the XY plane by default. You can toggle between the planes using the TAB key from the keyboard. Move the cursor to the origin to start the sketch or to the location from where you want to start the sketching. Press and hold down the left mouse button at this location and you are provided with a space handle. The space handle includes a coordinate system in the current plane.

If you toggle the plane using the TAB key then the coordinate system will change with respect to the current plane. Now, drag the cursor to a location where you want to define the endpoint of the line. Release the left mouse button at this location. Once you complete sketching the first line, move the cursor to the endpoint of the line created earlier. When the line cursor turns yellow in color, toggle the plane using the TAB key. Drag the cursor to create another line. Figures 9-56 through 9-58 show sketching in different planes in the 3D sketching environment.

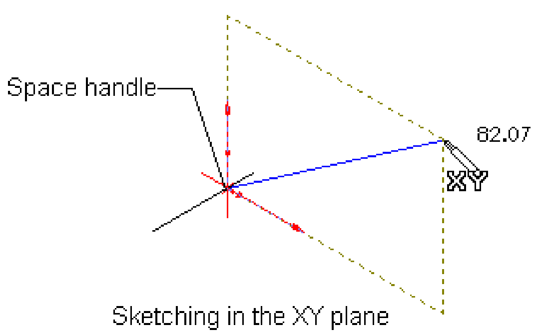


Figure 9-56 Sketching in the XY plane

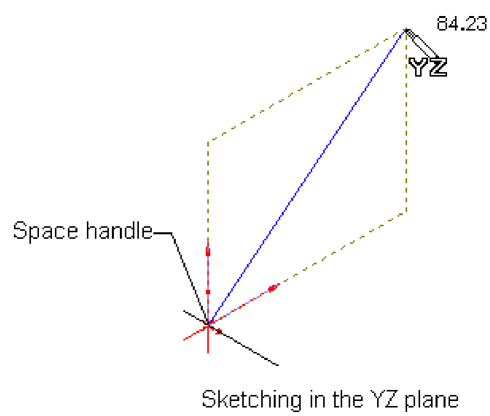


Figure 9-57 Sketching in the YZ plane

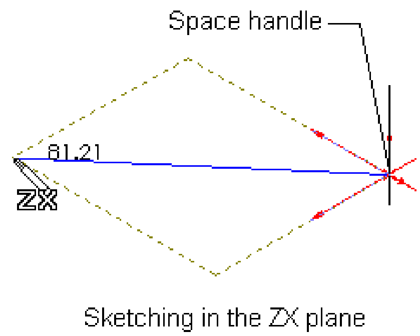


Figure 9-58 *Sketching in the ZX plane*

Figure 9-59 shows an example of a 3D sketch.

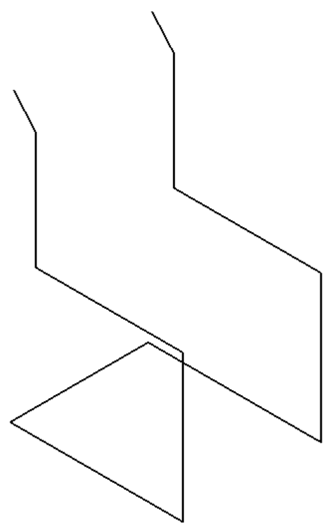
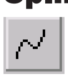


Figure 9-59 *A 3D sketch created using the line tool*

Spline



You can also create a spline in the 3D sketching environment. To create a 3D spline choose the **Spline** button from the **Sketching Tools** toolbar. The select cursor will be replaced by the spline cursor. Move the cursor to the desired location from where you want to start the sketching. Specify the start point of the spline; the space handle is displayed. You can toggle between the planes using the TAB key. Move the cursor where you want to specify the second point of spline and specify the second point of spline. As you specify the second point of spline, the space handle will be displayed and you toggle between the planes using the TAB key. After creating the spline, right-click and choose the **Select** option from the shortcut menu to end spline creation. Figure 9-60 shows a power cord with a knot; the cable of the power cord is created using the spline in the 3D sketching environment.

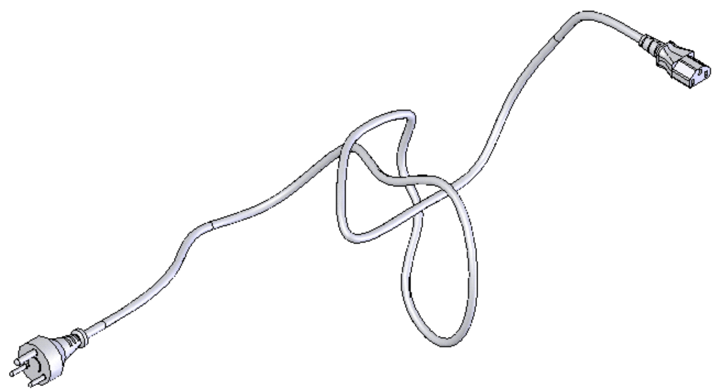
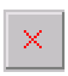


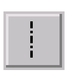
Figure 9-60 *Power cord with a knot*

Point



You can also create points in the 3D sketching environment of SolidWorks. To create a 3D point, first you have to invoke the 3D sketching environment and orient the drawing area to isometric view. Choose the **Point** button from the **Sketch Tools** toolbar to invoke the point tool. The select cursor will be replaced by the point cursor. Use the left mouse button to create points.

Centerline



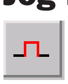
You can also create centerlines in the 3D sketching environment. Invoke the 3D sketching environment and orient the drawing view to isometric view. Choose the **Centerline** button from the **Sketch Tools** toolbar to create a centerline. The select cursor will be replaced by the line cursor. The procedure of creating the centerline is the same as that discussed for creating the lines.

The dimensioning of 3D sketches is the same as the dimensioning of 2D sketches.

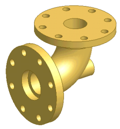
Editing the 3D Sketches

The editing operations that can be performed on 3D sketches are discussed next

Jog Line



The **Jog Line** tool is used to jog the sketched lines. When you create a jog line, automatic parallel and perpendicular relations are applied to the parent sketch. To create a jog line, choose the **Jog Line** button from the **Routing** toolbar or choose **Tools > Sketch Tools > Jog Line** from the menu bar. Now, select the start point of the jog line; a rectangle will be attached to the cursor. Using the TAB key you can toggle between the plane in which the jog line is being created. Move the cursor to define the size of the jog line and specify the endpoint of the jog line. You can also create jog lines in the 2D sketching



Tip. *If the **Routing** toolbar is not available in the SolidWorks window then you need to display the toolbar. To display the toolbar, choose **View** > **Toolbars** > **Routing** from the menu bar. The **Routing** toolbar will be displayed in the SolidWorks window. The other method of invoking the **Routing** toolbar is to move the cursor to any toolbar and right-click to invoke the shortcut menu. Choose the **Routing** toolbar from the shortcut menu.*

Using the above method you can invoke any toolbar that is not displayed in the SolidWorks window.

environment. Figure 9-61 shows the start point of the jog line. Figure 9-62 shows the cursor being moved to create the jog line. Figure 9-63 shows the resultant jog line.

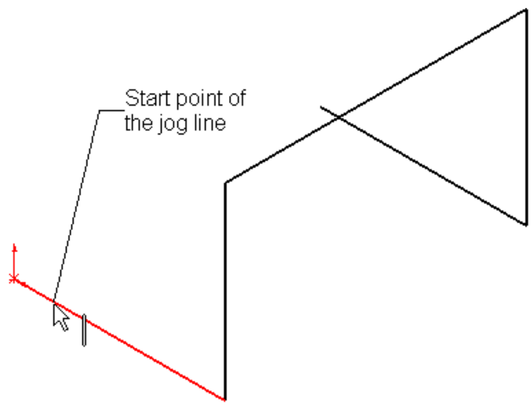


Figure 9-61 *Selecting the start point of the jog line*

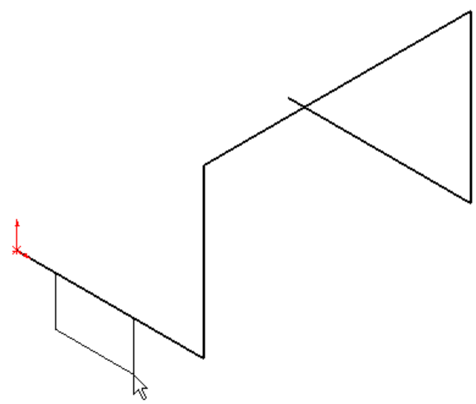


Figure 9-62 *Moving the mouse to specify the size of the jog line*

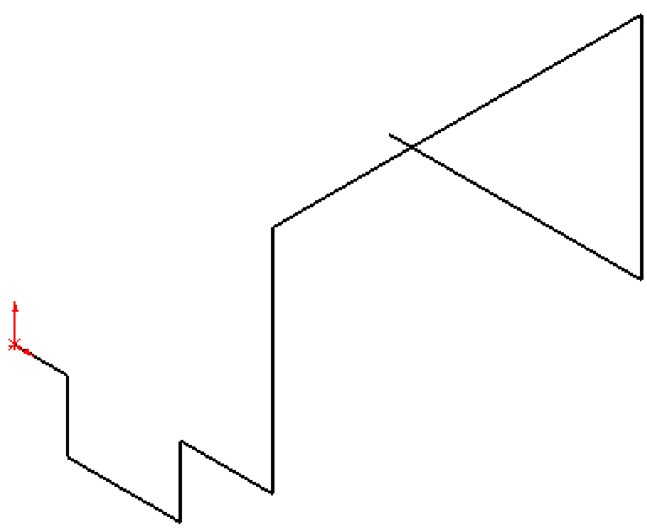


Figure 9-63 *The resultant jog line*

Other Editing Operations

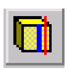
You can perform a number of editing operations on 3D sketches. These editing operations include **Convert Entities**, **Intersection Curves**, **Sketch Chamfer**, **Sketch Trim**, **Fit Spline**, **Sketch Trim**, **Sketch Extends**, and **Split Curve**. All these tools except **Intersection Curves** are discussed in the earlier chapters. The **Intersection Curves** tool will be discussed later.

Creating Curves

You can also create different types of curves in SolidWorks. Curves are used to create complex shapes generally using the sweep and loft tools. The types of curves that can be created in SolidWorks are discussed next.

Creating Projection Curve

Toolbar:	Curves > Projection
Menu:	Insert > Curve > Projected



This option allows you to project a sketched entity on one or more than one planar or curved faces. You can also project a sketched entity on another sketched entity to create a 3D curve. To create a projected curve, you first need to create at least two sketches or a sketch and at least one feature and then choose the **Projection** button from the **Curves** toolbar or choose **Inset** > **Curve** > **Projected** from the menu bar. When you choose the **Projection** button, the **Projected Curve PropertyManager** is displayed as shown in Figure 9-64. The confirmation corner is also displayed in the drawing area. The two different options to create projected curves are discussed next.

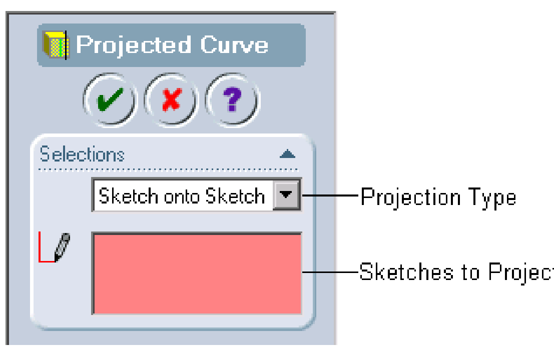


Figure 9-64** The **Projected Curve PropertyManager

Sketch onto Sketch

When you invoke the **Projected Curve PropertyManager**, the **Sketch onto Sketch** option is selected by default in the **Projection Type** drop-down list. You are prompted to select two sketches to project onto one another. Select two sketches to project them onto one another. When you select the sketches, the names of the sketches are displayed in the **Sketches to Project** display area. The preview of the projected curve is also displayed in the drawing area. Choose the **OK** button from the **Projected Curve PropertyManager** or choose the **OK** option from the confirmation corner. Figure 9-65 shows the two sketches selected to create a projected curve. Figure 9-66 shows the preview of the projected curve. Figure 9-67 show the resultant projected curve.

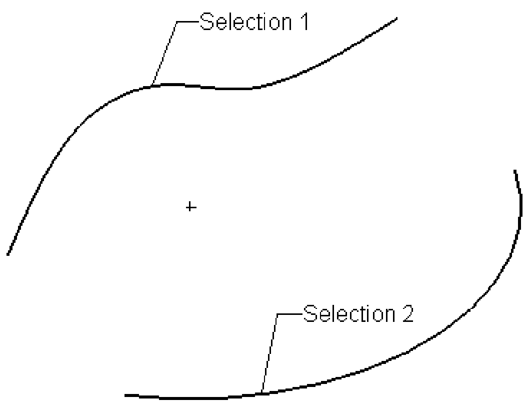


Figure 9-65 *Sketches to be selected*

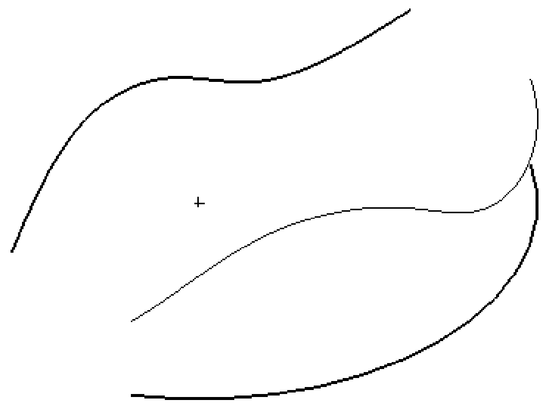


Figure 9-66 *Preview of the projected curve*

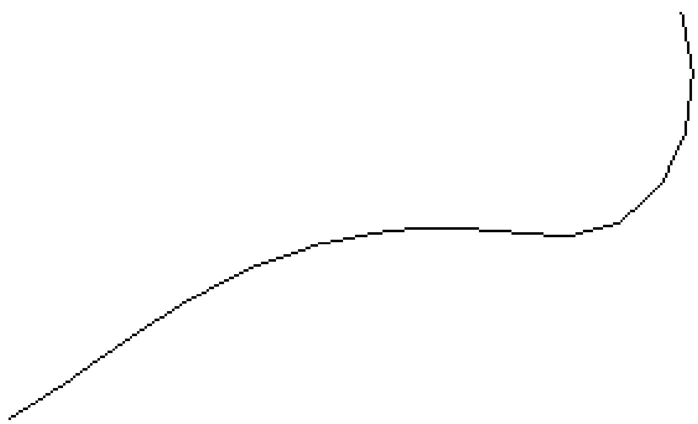


Figure 9-67 *Resultant projected curve*

Sketches onto Faces

The **Sketches onto Faces** option is available in the **Projection Type** drop-down list. This option is used to project a sketch on a planar or a curved face. When you choose this option, the **Sketch to Project** and the **Projection Faces** display areas are displayed in the **Selections** rollout. You are also provided with a **Reverse Projection** check box in the **Selections** rollout. The **Projected Curve PropertyManager** with **Sketches onto Faces** option selected is displayed in Figure 9-68. When you choose this option, you are prompted to select a sketch to project and the face on which to project. Now, you need to select the sketch from the drawing area and the face or faces on which you want to project the sketch. The selected sketch is highlighted in green and the selected face is highlighted in

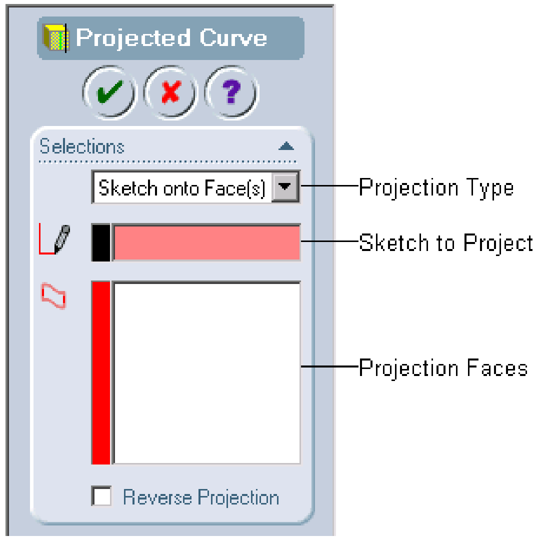


Figure 9-68 *The* ***Projected Curve PropertyManager*** *with the* ***Sketches onto Faces*** *option selected*

red. You are also provided with a **Reverse Projection** arrow in the drawing area. This arrow is used to reverse the direction of projection. You can also reverse the direction of projection using the **Reverse Projection** check box. Choose the **OK** button from the **Projected Curve PropertyManager** or choose the **OK** option from the confirmation corner. Figure 9-69 shows the sketch to be selected for projection and the face to be selected on which the sketch will be projected. Figure 9-70 shows the resultant projected sketch.

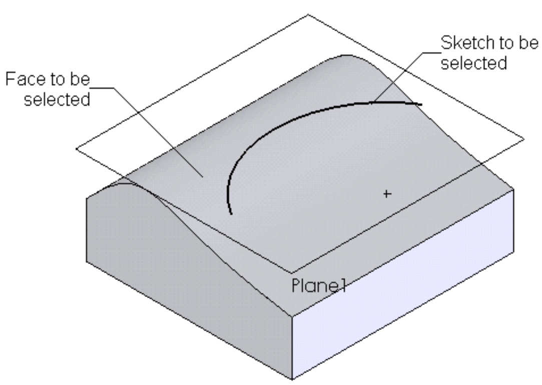


Figure 9-69 *Sketch and the face to be selected*

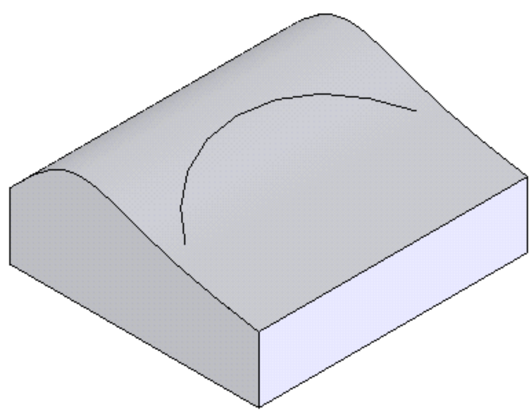


Figure 9-70 *Resultant projected curve*

Creating Split Lines

Toolbar:	Curves > Split Line
Menu:	Insert > Curve > Split Line

The **Split Line** tool is used to project a sketch on a planar or a curved face and in turn it splits or divides the single face into two or more than two faces. To create a split line to divide the faces, you need to invoke the **Split Line PropertyManager**. Choose the **Split Line** button from the **Curves** toolbar or choose **Insert** > **Curve** > **Split Line** from the menu bar to invoke the **Split Line PropertyManager** as shown in Figure 9-71. There are two methods of creating a split line, which are discussed next.

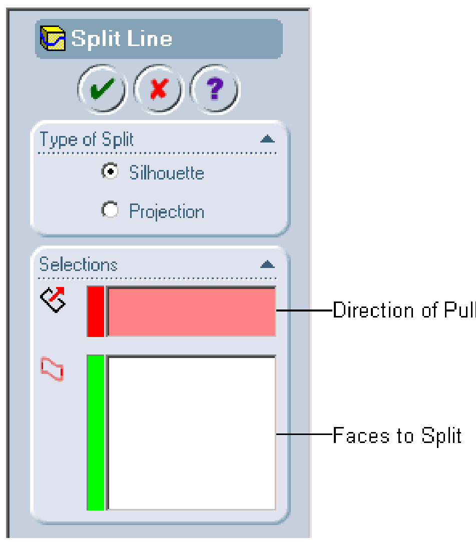


*Figure 9-71 The **Split Line PropertyManager***

Silhouette

Using this option you can split a cylindrical or circular face by creating the silhouette line at the intersection of the projection of plane and the cylindrical face. When you invoke the **Split Line PropertyManager**, the **Silhouette** option available in the **Types of Split** rollout is selected by default. You are prompted to change the type or select the direction of pull and faces to split. You need to select a plane that defines the direction of pull. Select the plane; the selected plane will be highlighted in red color. Now, select the cylindrical or circular face; the selected face will be highlighted in green color. Choose the **OK** button from the **Split Line PropertyManager** or choose the **OK** option from the confirmation corner. The cylindrical or circular face will be divided in two or more faces. Figure 9-72 shows the plane and the face to be selected. Figure 9-73 shows the resultant split line created to split the selected face.

Projection

Using this option you can project a sketched entity onto a planar or a curved face to create a split line on that face. The split line tends to split the selected face on which the sketch is projected. To use this option invoke the **Split Line PropertyManager** and choose

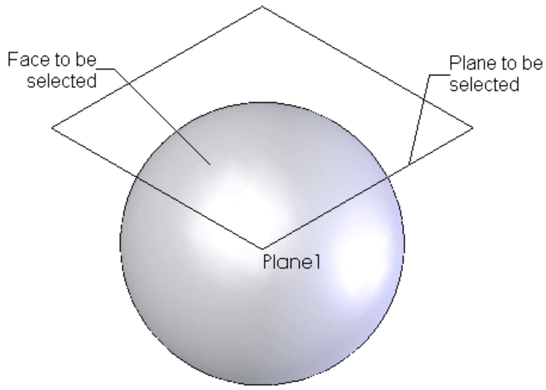


Figure 9-72 Plane and the face to be selected

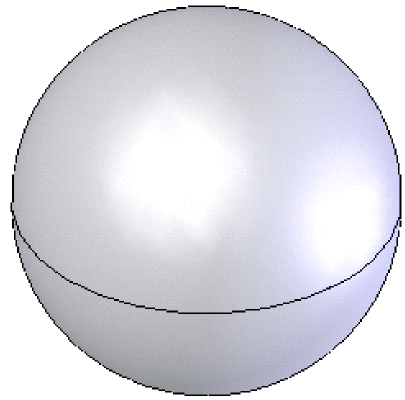


Figure 9-73 Resultant split line

the **Projection** option from the **Types to Split** rollout. The **Split Line PropertyManager** with the **Projection** option selected is shown in Figure 9-74. You are prompted to change

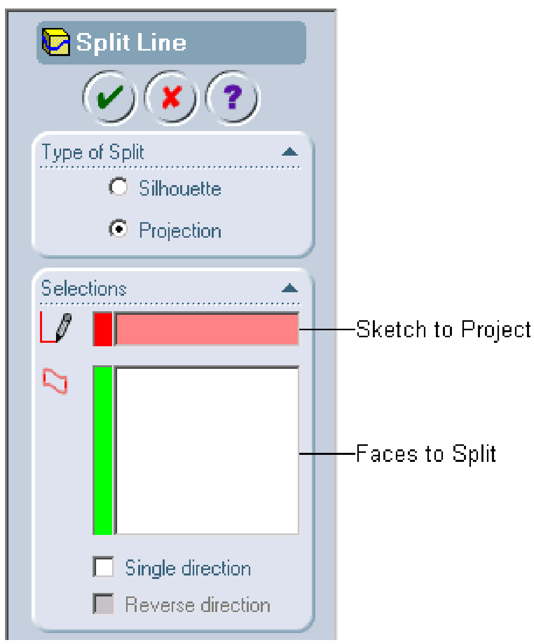


*Figure 9-74 The **Split Line PropertyManager** with the **Projection** option selected*

the type or select the sketch to project, direction, and faces to split. Select the sketch and the selected sketch will be displayed in green. Now, you need to select the face to split. Select the face; the selected face will be displayed in green and the preview of the split line will also be displayed in the drawing area. Choose the **OK** button from the confirmation corner. The selected face or faces will split into two or more than two faces. Figure 9-75 shows the sketch and face to be selected. Figure 9-76 shows the resultant split line created to split the selected face.

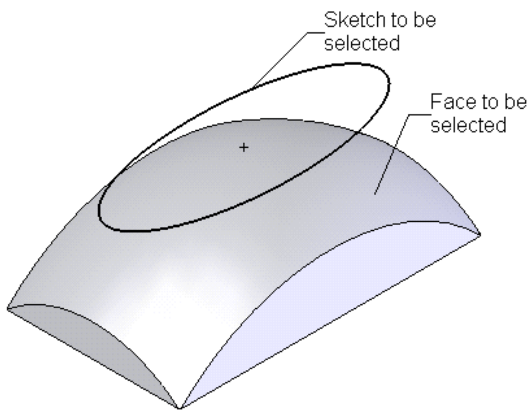


Figure 9-75 *Sketch and the face to be selected*

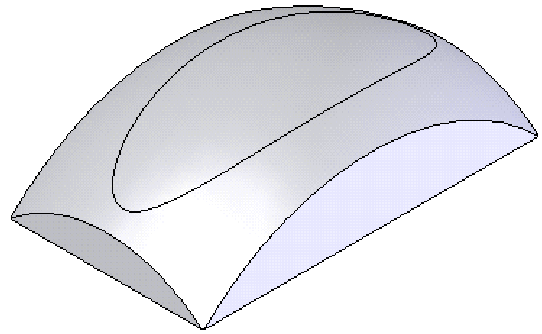


Figure 9-76 *Resultant split line*

The other options available in the **Selections** rollout of the **Split Line PropertyManager** are discussed next.

Single direction

While creating a split on a cylindrical face, if the sketching plane on which the sketch is created lies within the model, the split line will be created on two sides of the cylindrical face. The **Single direction** check box available in the **Selections** rollout is used to create the split line only in one direction. The **Reverse direction** check box is available only if the **Single direction** check box is selected. The **Reverse direction** check box is used to reverse the direction in which the split line should be created. Figure 9-77 shows the split line created on both sides of the model. Figure 9-78 shows the split line created on single side of the model.

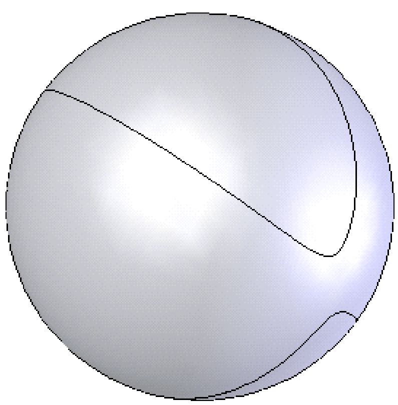


Figure 9-77 *Split line created on both sides*

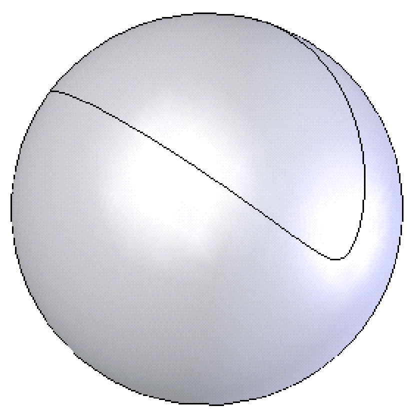
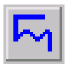


Figure 9-78 *Split line created on single side*

Creating Composite Curve

Toolbar:	Curves > Composite Curve
Menu:	Insert > Curve > Composite



The **Composite Curve** option is used to create a curve by combining other curves, sketched entities, and edges into a single curve. The composite curve is mainly used while creating a sweep or a loft feature. To create a composite curve you need to invoke the **Composite Curve PropertyManager**. Choose the **Composite Curve** button from the **Curves** toolbar or choose **Insert > Curve > Composite** from the menu bar to invoke the **Composite Curve PropertyManager**. The **Composite Curve PropertyManager** is shown in Figure 9-79.

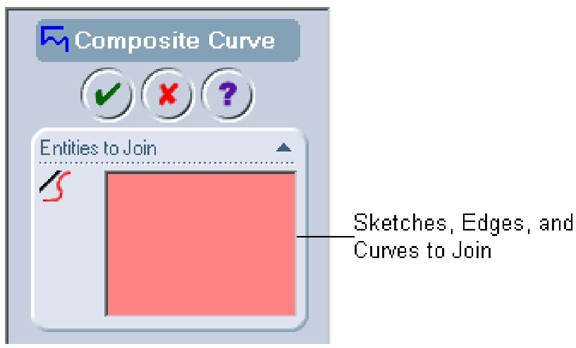
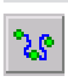


*Figure 9-79 The **Composite Curve PropertyManager***

When you invoke the **Composite Curve PropertyManager** you are prompted to select a continuous set of sketches, edges, and/or curves. Select the edges, curves, or sketched entities to create a continuous curve. Choose the **OK** button to end curve creation. The entities to be selected should form a continuous chain, otherwise the composite curve will not be created.

Creating Curve Through Free Points

Toolbar:	Curves > Curve Through Free Points
Menu:	Insert > Curve > Curve Through Free Points



The **Curve Through Free Points** option is used to create a curve by specifying the coordinate points. To create a curve using this option, choose the **Curve Through Free Points** button from the **Curves** toolbar or choose **Insert > Curve > Curve Through Free Points** from the menu bar. The **Curve File** dialog box is displayed as shown in Figure 9-80. Double-click the first column under the X area to invoke the first row to enter the coordinates for the start point for the curve. Double-click in the column below the first column to enter the coordinates for the second point for creating the curve. Similarly, specify the coordinates of the other points of the curve, see Figure 9-81. When you enter the coordinates of the points the preview of the curve is displayed in the drawing area. Choose the **OK** button from the **Curve File** dialog box to complete the feature creation, see Figure 9-82.

You can also save the current set of coordinates using the **Save** button available in the **Curve File** dialog box. When you choose this button to save the current set of coordinates, the **Save**

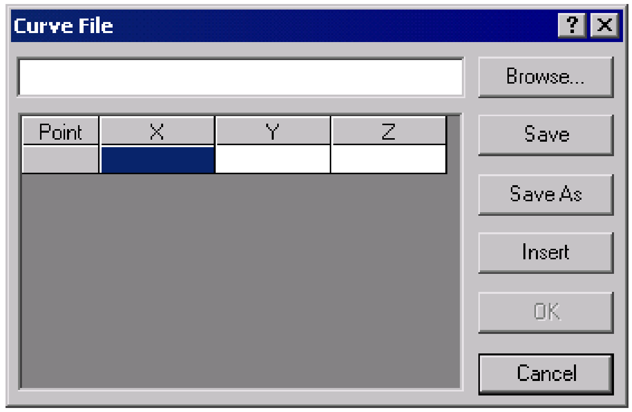
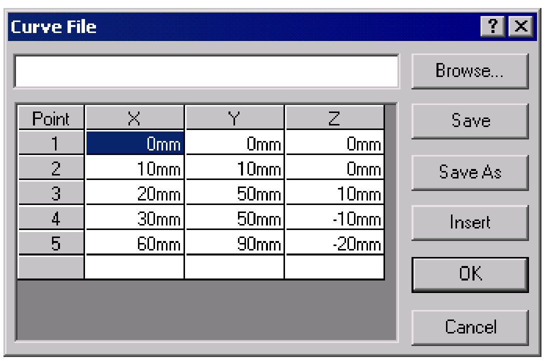


Figure 9-80 *The **Curve File** dialog box*



Point	X	Y	Z
1	0mm	0mm	0mm
2	10mm	10mm	0mm
3	20mm	50mm	10mm
4	30mm	50mm	-10mm
5	60mm	90mm	-20mm

Figure 9-81 *Coordinates entered in the **Curve File** dialog box*

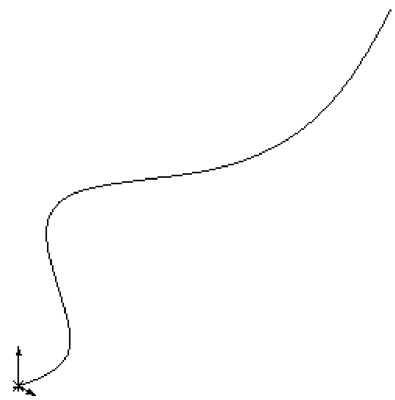
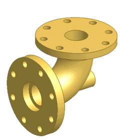


Figure 9-82 *Resultant 3D curve*

As dialog box is displayed. Browse the directory in which you need to save the coordinates and enter the name of the file in the **File name** message area and choose the **Save** button. The curve file is saved with extension *.sldcrv*. Using the **Save As** button you can save the current set of coordinates with some other name.

Using the **Browse** button, you can open an existing curve file. Invoke the **Curve File** dialog box and choose the **Browse** button. The **Open** dialog box will be displayed. You can browse the previously saved curve file to specify the coordinate points. You can also write the coordinates in a text (notepad) file and save it. In the **Open** dialog box, choose the **Text Files (*.txt)** option from the **Files of type** drop-down list and browse the text file to specify the coordinates.



Tip. *You can select a row and using the DELETE key from the keyboard you can delete the entire row. Using the SHIFT key select the entire row and using the **Insert** button, you can add a row between two rows.*

Creating Curve Through Reference Points

Toolbar:	Curves > 3D Curve
Menu:	Insert > Curve > Curve Through Reference Points

The **Curve Through Reference Points** option enables you to create a curve by selecting the sketched points, vertices, origin, endpoints, or center points. To create a curve through reference points, you have to invoke the **Curve Through Reference Points PropertyManager**. Choose the **3D Curve** button from the **Curves** toolbar or choose **Insert > Curve > Curve Through Reference Points** from the menu bar. The **Curve Through Reference Points PropertyManager** is shown in Figure 9-83.

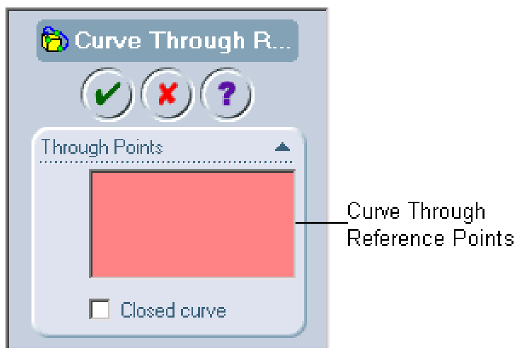


Figure 9-83** The **Curve Through Reference Points PropertyManager

When you invoke this tool, you are prompted to select vertices to define the through points for the curve. Select the points to define the curve. As you define the points, the preview of the resultant curve created using the selected points is displayed in the drawing area. After specifying all the points, choose the **OK** button or choose the **OK** option from the confirmation corner. You can also use the **Closed curve** check box to create a closed curve. Figure 9-84 shows the vertices to be selected to create the curve through reference points. Figure 9-85 shows the resultant 3D curve.

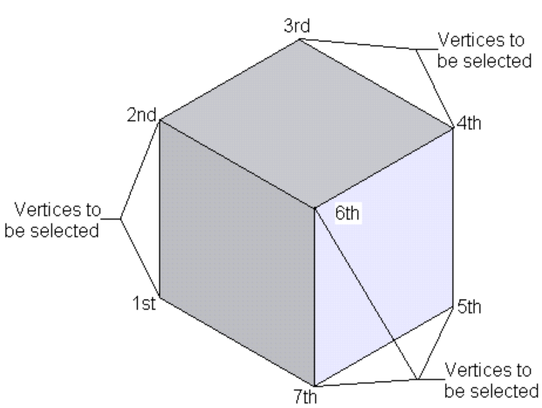


***Figure 9-84** Vertices to be selected*

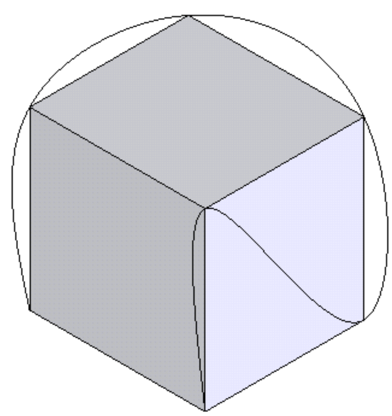
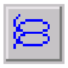


***Figure 9-85** Resultant 3D curve*

Creating Helical Curve

Toolbar:	Curves > Helix
Menu:	Insert > Curve > Helix/Spiral



The **Helical Curve** option is used to create a helical curve or a spiral curve. The helical or spiral curve is used as the sweep path to create springs, threads, spiral coils, and so on. Figure 9-86 shows a spring created by sweeping a profile along a helical path. Figure 9-87 shows a spiral coil created by sweeping a profile along a spiral path.

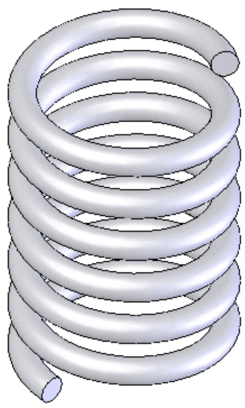


Figure 9-86 Spring

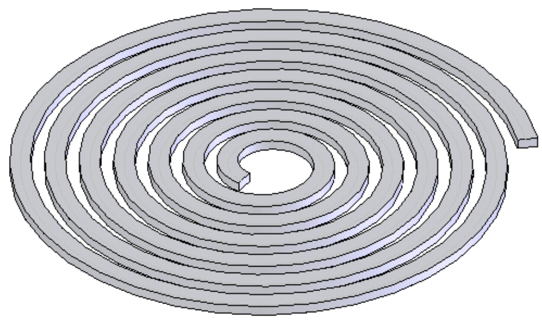


Figure 9-87 Spiral coil

To create a helix in SolidWorks you first have to create a sketch of the circle that defines the diameter of the helical curve. If you are creating a spiral, the sketch will define the starting diameter of the spiral curve. You can invoke the **Helix** tool while you are in the sketching environment. If you are not in the sketching environment, you first have to select the sketch and then invoke this tool. Choose the **Helix** button from the **Curves** toolbar or choose **Insert > Curve > Helix/Spiral** from the menu bar. The **Helix Curve** dialog box will be displayed as shown in Figure 9-88.

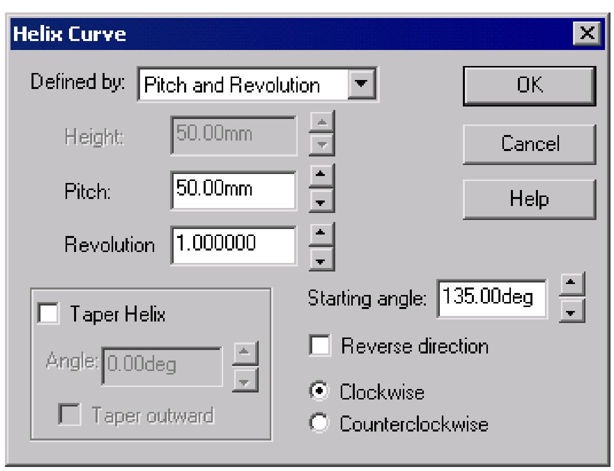


*Figure 9-88 The **Helix Curve** dialog box*

The preview of the helix curve is displayed in the drawing area with the default values. There are various methods to specify the parameters of the helical curve. These methods are discussed next.

Pitch and Revolution

The **Pitch and Revolution** option available in the **Defined by** drop-down list is selected by default. Using this option, you can specify the pitch of the helical curve and the number of revolutions. When this option is selected, the **Pitch** spinner and the **Revolution** spinner are available in the **Helix Curve** dialog box to define the value of pitch and number of revolutions.

Height and Revolution

The **Height and Revolution** option available in the **Defined by** drop-down list is used to define the parameters of the helix curve in the form of the total helix height and the number of revolutions. When you choose this option, the **Height** and **Revolution** spinners are displayed to specify the required parameters.

Height and Pitch

The **Height and Pitch** option available in the **Defined by** drop-down list is used to define the parameters of the helix curve in terms of the height and the pitch of the helix. When you select this option, the **Height** and **Pitch** spinners are displayed to specify the required parameters.

When you specify the parameters to create the helix curve, the preview in the drawing area modifies dynamically. Figure 9-89 shows a helix.

You can also create a tapered helix using the **Taper Helix** check box available in the **Helix Curve** dialog box. The procedure of creating a taper helical curve is discussed next.

Taper Helix

The **Taper Helix** check box is selected to create a tapered helical curve. To create a tapered helical curve, select the **Taper Helix** check box from the **Helix Curve** dialog box. The **Angle** spinner and the **Taper outward** check box are enabled. Using the **Angle** spinner you can specify the value of the angle of taper. The **Taper outward** check box is used to create an outward taper. When you specify the parameters to create a tapered helical curve, the preview of the helical curve updates automatically in the drawing area. Figure 9-90 shows a tapered helical curve. Figure 9-91 shows a tapered helical curve created with the **Taper outward** check box selected.

Using the **Start angle** spinner you can specify the start angle of the helical curve. The **Reverse direction** check box is selected to reverse the direction of helical curve creation.

By default, the helical curve is created in the clockwise direction. Therefore, the **Clockwise** radio button is selected in the **Helix Curve** dialog box. If you need to create the helical curve in the counterclockwise direction, you need to select the **Counterclockwise** radio button. After setting all the parameters, choose the **OK** button from the **Helix Curve** dialog box.

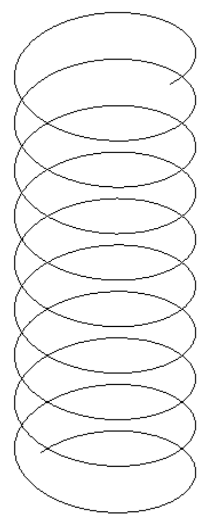


Figure 9-89 Helical Curve

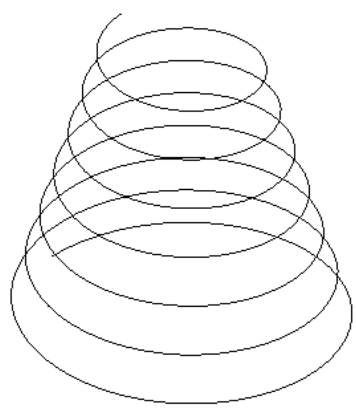


Figure 9-90 *Tapered helical curve*

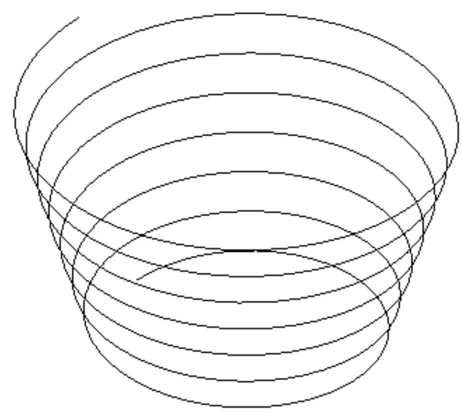


Figure 9-91 *Tapered helical curve with the* ***Taper outwards*** *option selected*

Creating the Spiral Curve

For creating a **Spiral Curve** first you have to create a sketched circle. The circle will define the inner diameter of the spiral coil. Select the **Spiral** option from the **Defined by** drop-down list available in the **Helix Curve** dialog box. The preview of the spiral curve will be displayed in the drawing area. You can define the pitch and the number of revolutions in the **Pitch** spinner and the **Revolution** spinner, respectively. The other options except **Taper Helix** are available while creating a spiral curve. These options are the same as those discussed earlier. After specifying all the required parameters, choose the **OK** button. Figure 9-92 shows a spiral curve.

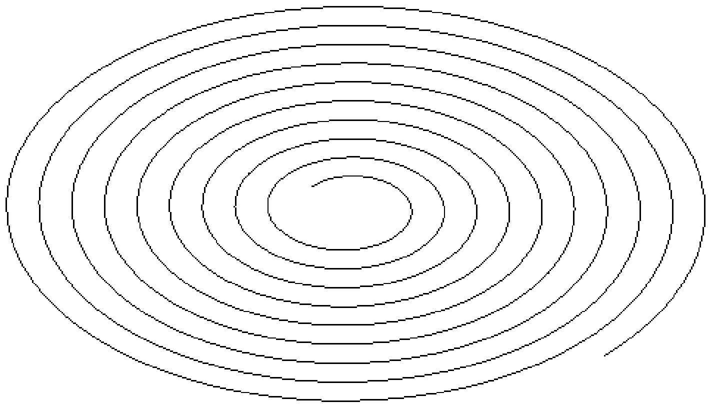
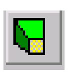


Figure 9-92 Spiral curve

Creating Draft Features

Toolbar:	Features > Draft
Menu:	Insert > Features > Draft



The **Draft** tool is used to taper the selected faces of the model. One of the main application of the draft feature is to taper the faces of the parts to be moulded or casted so that it is easier to remove them from the mould or die. To create a draft feature you need to invoke the **Draft PropertyManager**. Choose the **Draft** button from the **Features** toolbar or choose **Insert > Features > Draft** from the menu bar to invoke the **Draft PropertyManager**. The **Draft PropertyManager** is shown in Figure 9-93.

After invoking this tool, you are prompted to select a neutral plane and the faces to draft. You will notice that the **Neutral Plane** option is selected by default in the **Types of Draft** drop-down list in the **Types of Draft** rollout. Therefore, you need to select a neutral plane for creating the draft feature. Select a planar face or a plane that acts as a neutral plane. The selected face will be displayed in red with the **Neutral Plane** callout. A **Reverse Direction** arrow is also displayed in the drawing area. Now, click in the **Faces to Draft** display area in the **Faces to Draft** rollout to activate the selection mode. Select the faces to apply the draft. The selected faces will be displayed green in color with the **Draft Face** callout. Now, set the value of the draft angle in the **Draft Angle** spinner available in the **Draft Angle** rollout. Choose the **OK** button from the **Draft PropertyManager** or choose the **OK** option from the confirmation corner.

Figure 9-94 shows the neutral face and the faces to be selected to add the draft. Figure 9-95 shows the resultant draft feature. The other options in the **Draft PropertyManager** are discussed next.

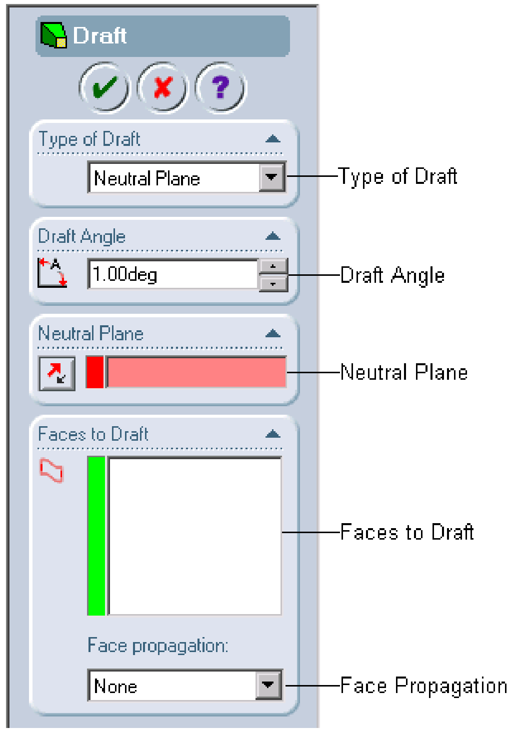


Figure 9-93 *The* ***Draft PropertyManager***

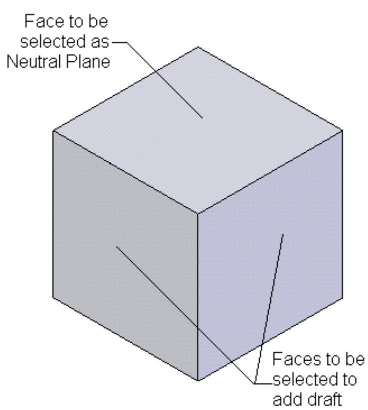


Figure 9-94 *Faces to be selected*

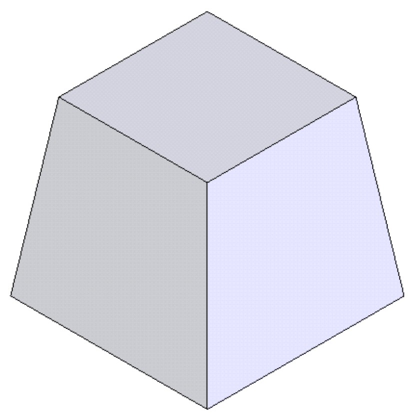


Figure 9-95 *Resultant draft feature*

Reverse Direction

The **Reverse Direction** button available on the left of the **Neutral Plane** display area in the **Neutral Plane** rollout is used to reverse the direction of draft creation. You can also reverse the direction of draft creation by selecting the **Reverse Direction** arrow from the drawing area.

Face propagation

The options available in the **Face Propagation** drop-down list are used to extend the draft feature to the other faces. The options available in this drop-down list are discussed next.

None

The **None** option is selected by default. This option is used when you do not need to apply any type of face propagation.

Along Tangent

The **Along Tangent** option is used to apply the draft to the faces tangent to the selected face.

All Faces

The **All Faces** option is used to apply the draft to all the faces attached to the neutral plane or face.

Inner Faces

This option is used to draft all the faces inside the model that are attached to the neutral plane or face.

Outer Faces

This option is used to draft all the outside faces of the model that are attached to the neutral plane or face.

Creating the Draft Using the Parting Line

To create a draft feature using the **Parting Line** option, you first need to create a parting line using the split curve option. The **Split Curve** option was discussed earlier in this chapter. You can also select the model edges as split curve. Now, invoke the **Draft PropertyManager** and select the **Parting Line** option from the **Type of Draft** drop-down list. The **Draft PropertyManager** with the **Parting Line** option selected is displayed in Figure 9-96. You are prompted to select the direction of pull and the parting lines.

Select a planar face, plane, or an edge as the pull direction for creating the draft feature. The **Direction of Pull** callout and the **Reverse Direction** arrow are displayed in the drawing area. Click once in the **Parting Line** area to invoke the selection mode and select the parting lines. The direction arrows will be displayed with the selected parting lines. Using the **Other Face** button, you can reverse the direction of the selected parting line. Set the value of the angle in the **Draft Angle** spinner and end feature creation.

Figure 9-97 shows the pull direction and the parting lines to be selected. Figure 9-98 shows the resultant draft feature creation.

Allow reduced angle

When applying the draft feature using the **Parting Line**, a smaller draft angle is applied to some portions of the draft feature because of geometric conditions. Therefore, the **Allow reduced angle** check box available in the **Type of Draft** rollout is used to maintain the consistency in the draft feature.

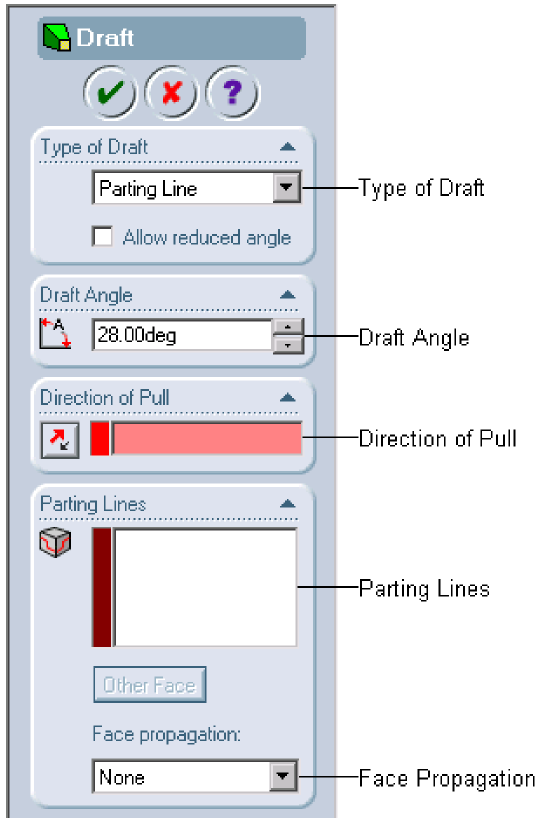


Figure 9-96 *The* ***Draft PropertyManager*** *with the* ***Parting Line*** *option selected from the* ***Type of Draft*** *drop-down list*

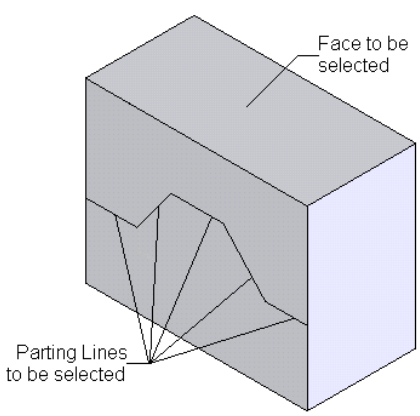


Figure 9-97 *Face and the parting lines to be selected*

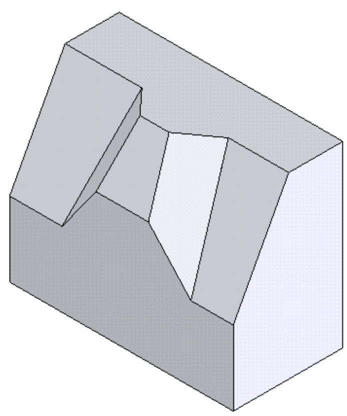


Figure 9-98 *Resultant draft feature*

Creating the Step Draft

The **Step Draft** option available in the **Type of Draft** drop-down list is used to create a step draft. The **Draft PropertyManager** with the **Step Draft** option selected is shown in Figure 9-99.

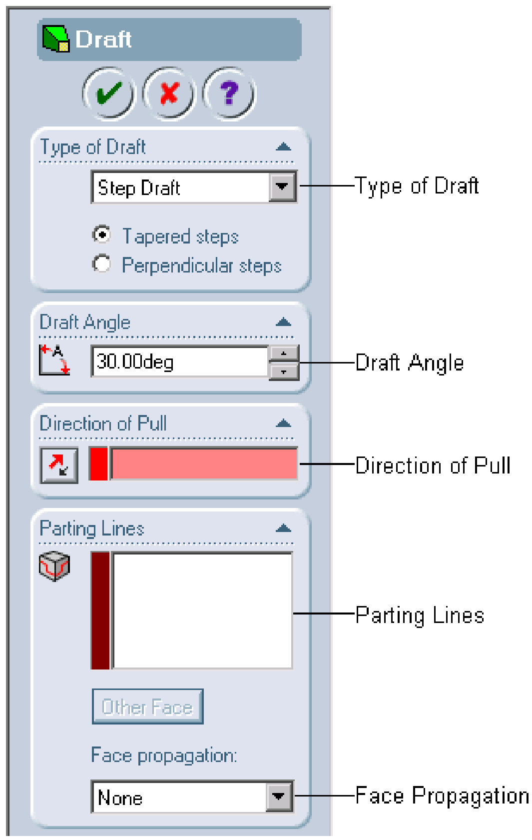


Figure 9-99 *The **Draft PropertyManager** with the **Step Draft** option selected from the **Type of Draft** drop-down list*

You are prompted to select the direction of pull and the parting line. Select the direction of pull; the **Reverse Direction** arrow and the **Direction of Pull** callout are displayed. Select the parting lines to create the step draft. As discussed you can change the direction of the parting lines using the **Other Face** option. Set the value of the angle in the **Draft Angle** spinner and choose the **OK** button from the **Draft PropertyManager**. Figure 9-100 shows the face selected to define the direction of pull. Figure 9-101 shows the step draft feature created.

You will notice that the **Tapered steps** radio button is selected by default in the **Type of Draft** rollout. Some of the faces of the step draft created will include a taper. If you select the **Perpendicular steps** radio button, the steps created will be perpendicular. Figure 9-102

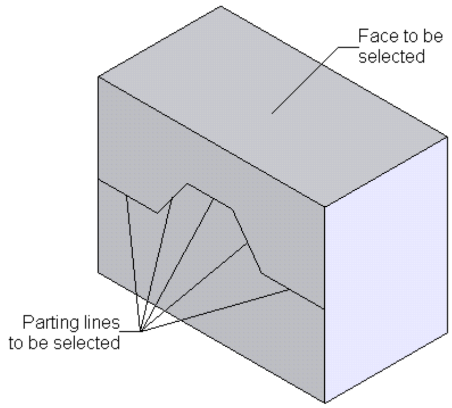


Figure 9-100 *Face and the parting lines to be selected*

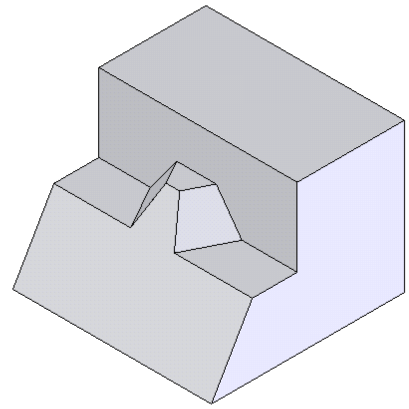


Figure 9-101 *Resultant step draft feature*

shows the step draft created with the **Tapered steps** radio button selected. Figure 9-103 shows the step draft created with the **Perpendicular steps** radio button selected.

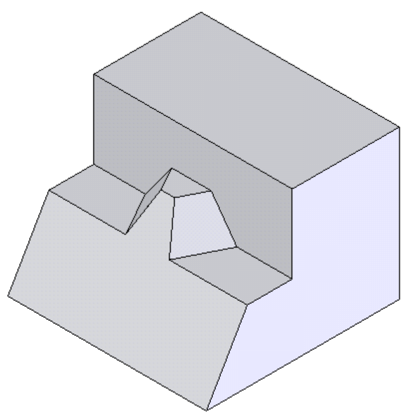


Figure 9-102 *Step draft created with the* ***Tapered steps*** *radio button selected*

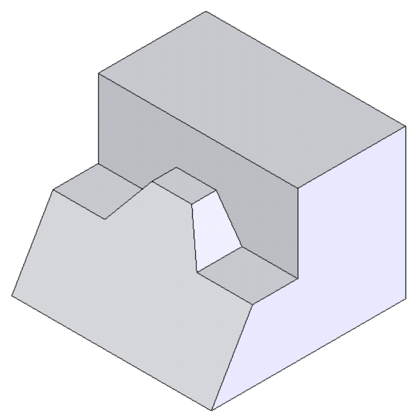


Figure 9-103 *Step draft created with the* ***Perpendicular steps*** *radio button selected*

TUTORIALS

Tutorial 1

In this tutorial you will create the model shown in Figure 9-104. The dimensions of model are shown in Figure 9-105. **(Expected time: 45 min)**

The steps to be followed to complete this tutorial are given next:

a. The base feature of the model is a sweep feature. First, you need to create the path of the sweep feature on the front plane. Next, you need to create a plane normal to the path of the sweep feature. Select the newly created plane as the sketching plane and create the

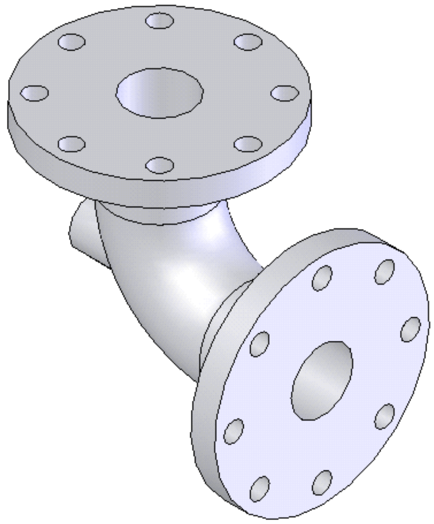


Figure 9-104 *Model for Tutorial 1*

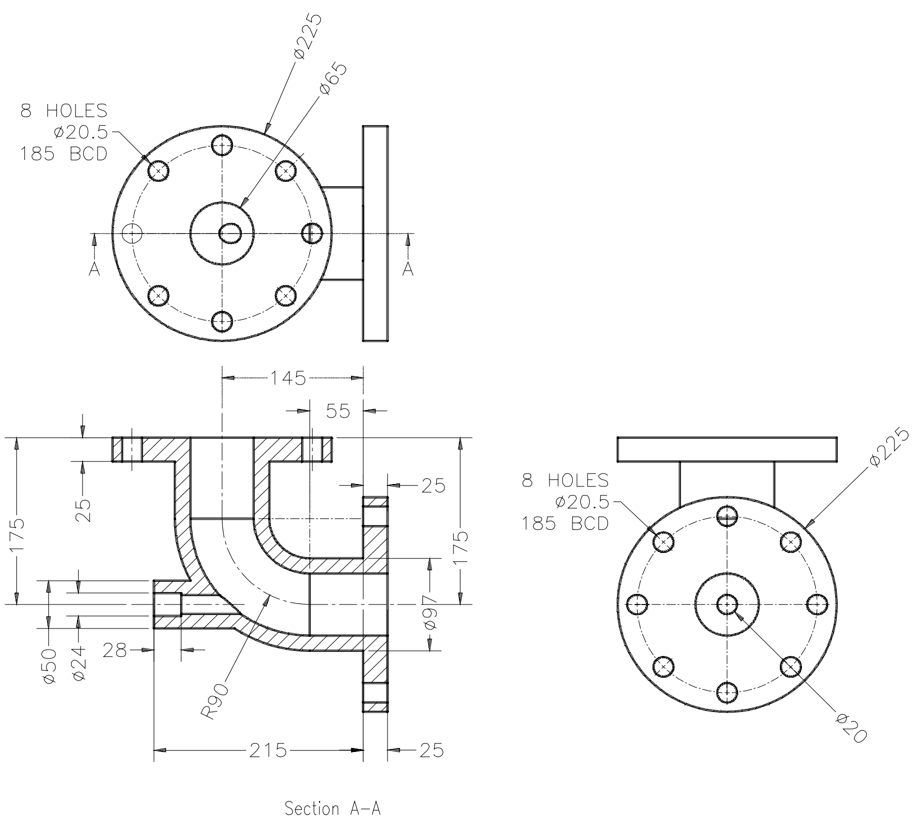


Figure 9-105 *Views and dimensions of the model for Tutorial 1*

profile of the sweep feature. You will create a thin sweep feature because the base feature of the model is a hollow feature, refer to Figures 9-106 through 9-108.

b. Create the extrude features on both ends of the sweep feature, refer to Figure 9-109.

c. Create a plane at an offset distance from the right face of the model. Create the circular feature by extruding the feature using the **Up To Next** option.

d. Create the hole using the **Simple Hole PropertyManager**, refer to Figure 9-109.

e. Create the pattern of the hole feature, refer to Figure 9-109.

f. Create the counterbore hole using the cut revolve option, refer to Figure 9-109.

Creating the Path for the Sweep Feature

As discussed earlier, the base feature of the model is a sweep feature. To create the sweep feature, you first need to create the path of the sweep feature. This path will be created on the **Front** plane.

1. Start SolidWorks and open a new part document from the **Template** tab of the **New SolidWorks Document** dialog box.

2. Draw the sketch of the path of the sweep feature on the **Front** plane and add the required relations and dimensions to the sketch as shown in Figure 9-106. Exit the sketching environment and change the view to isometric view.

Creating the Profile of the Sweep Feature

After creating the path of the sweep feature, you will create the profile of the sweep feature. For creating the profile, first you need to create a reference plane normal to the path. The newly created plane will be selected as the sketching plane for creating the profile of the sweep feature.

1. Invoke the **Plane PropertyManager** and using the **Normal to Curve** option, create a plane normal to the path as shown in Figure 9-107.

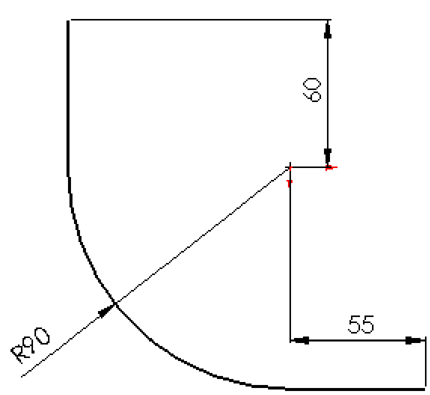


Figure 9-106 *Sketch of the path*

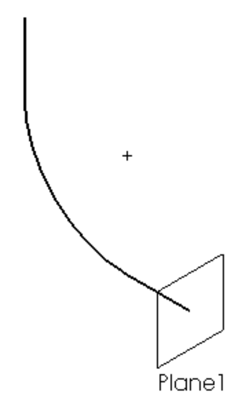


Figure 9-107 *Plane created normal to path*

2. Invoke the sketching environment by selecting the newly created plane as the sketching plane.

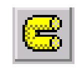
3. Create the sketch of the profile of the sweep feature using the circle tool. The diameter of the profile of the sweep feature is 97.

4. After creating the profile of the sweep feature exit the sketching environment.

Creating the Sweep Feature

The sweep feature that you are going to create is a thin sweep feature. You will use the **Thin Feature** rollout to specify the parameters of the thin feature.

1. Choose the **Sweep** button from the **Features** toolbar. The **Sweep PropertyManager** is invoked and you are prompted to select the sweep profile.



2. Select the profile of the sweep feature. The selected profile will be highlighted in green and the **Profile** callout will also be displayed.

   After selecting the profile, you are prompted to select the path.

3. Select the path of the sweep feature. The selected path will be highlighted in red and the **Path** callout is also displayed. The preview of the sweep feature is also displayed in the drawing area.

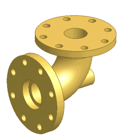
4. Select the **Thin Feature** check box available on the left of the rollout. The **Thin Feature** rollout is invoked.

5. Set the value of thickness in the **Thickness** spinner to **16**.

   Since the wall thickness added to the model is reverse to the required direction, therefore, you need to reverse the direction of creation of thin feature.

6. Choose the **Reverse Direction** button from the **Thin Features** rollout. Choose the **OK** button from the **Sweep PropertyManager** or choose **OK** from the confirmation corner.

   The base feature created by sweeping a profile along a path is shown in Figure 9-108.



Tip. *Instead of creating a thin sweep feature, you can also create a solid sweep feature and then add a shell feature to hollow the base feature.*

Creating the Remaining Features

Create the remaining join features of the model using the extrude option. Using the **Simple Hole PropertyManager** create the holes and pattern them using the circular pattern tool. Create the counterbore hole using the **Hole Wizard** or using the revolve cut option.

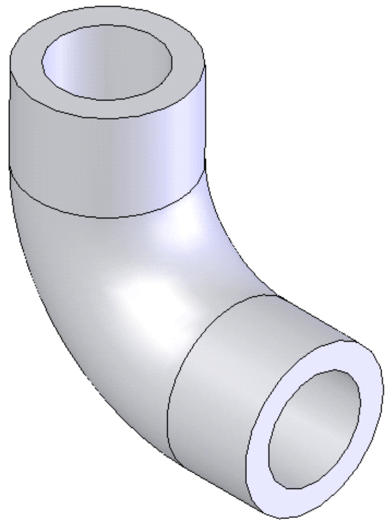


Figure 9-108 *Base feature of the model*

The final solid model of Tutorial 1 is shown in Figure 9-109. The model tree of the model is shown in Figure 9-110.

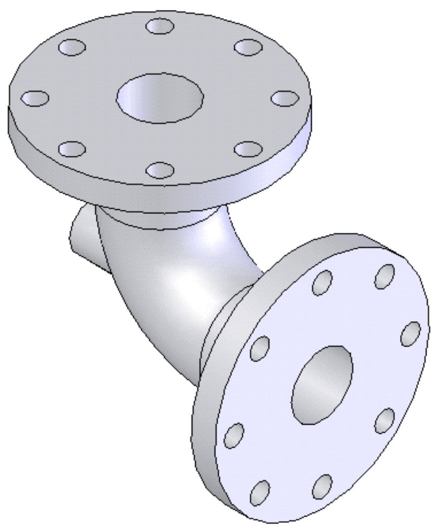
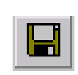


Figure 9-109 *Final model of Tutorial 1*

Saving the Model

Next, you need to save the model.

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below:



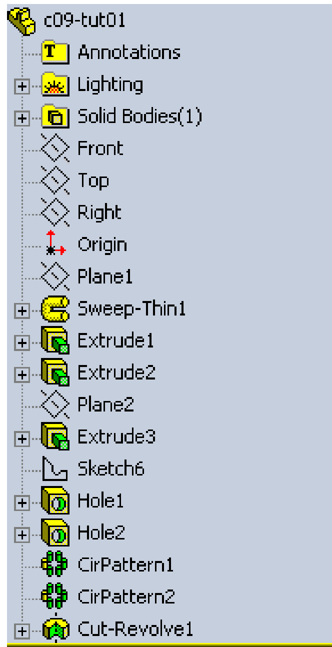


*Figure 9-110 The **FeatureManager Design Tree** for the model*

\My Documents\SolidWorks\c09\c09-tut01.SLDPRT.

2. Choose **File** > **Close** from the menu bar to close the file.

Tutorial 2

In this tutorial you will create the chair frame shown in Figure 9-111. The dimensions of the chair frame are shown in Figure 9-112. **(Expected time : 30 min)**

The steps to be followed to complete this tutorial are discussed next:

a. The chair frame is created by sweeping a profile along a 3D path. The 3D path will be created in the 3D sketching environment. Therefore, you need to invoke the 3D sketching environment and then create the sketch of the 3D path. You will create only the left half of the 3D path in the 3D sketching environment, refer to Figure 9-113.
b. Create a plane normal to the 3D path. Selecting the newly created plane as the sketching environment create the sketch of the profile.
c. Sweep the profile along the 3D path using the **Thin Feature** option, refer to Figure 9-114.
d. Mirror the sweep feature using the **Front** plane, refer to Figure 9-115.

Creating the Path of Sweep Feature Using 3D Sketching Environment

It is evident from Figure 9-111 that the model is created by sweeping a profile along a 3D path. Therefore, you need to create a path of the sweep feature in the 3D sketching environment.

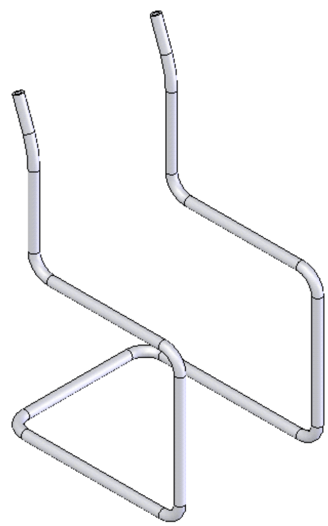


Figure 9-111 *Model of Tutorial 2*

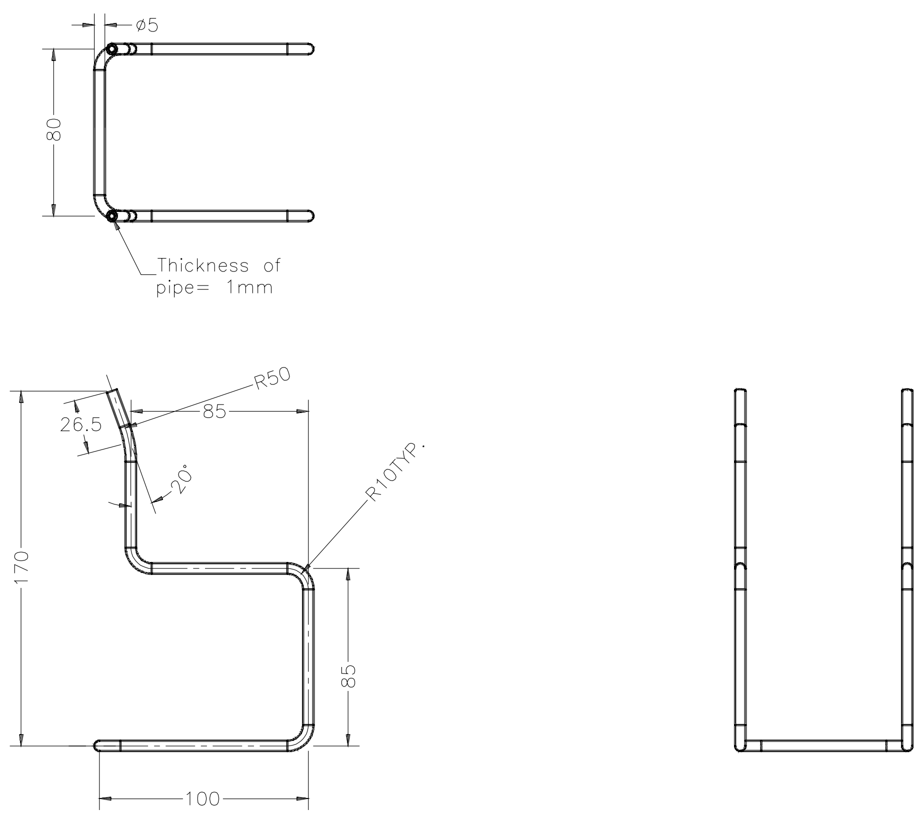
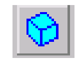


Figure 9-112 *Views and dimensions of the model for Tutorial 2*

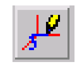
1. Create a new document in the **Part** mode.

2. Create a plane at an offset distance of 40mm from the **Front** plane.

3. Change the current view to isometric using the **Isometric** button from the **Standard Views** toolbar.



4. Choose **Insert > 3D Sketch** from the menu bar to invoke the 3D sketching environment. You can also use the **3D Sketch** button from the **Sketch** toolbar to invoke the 3D sketching environment. You will need to customize this button using the **Customize** toolbar.



   The 3D sketching environment is invoked and the sketch origin is displayed in red. You are also provided with the confirmation corner on the top right of the drawing area. The sketching tools that can be used in the 3D sketching environment are also invoked.

5. Invoke the **Line** tool. The select cursor is replaced by the line cursor. The **XY** that is displayed below the line cursor suggests that by default it will sketch in the **XY** plane.

   The first line you need to create is in the **ZX** plane. Therefore, you will toggle the plane before you start creating the sketch.

6. Choose the TAB key from the key board twice to switch to **ZX** plane.

7. Move the line cursor to the origin. When the cursor turns yellow in color, press and hold down the left mouse button. A space handle will be displayed at the origin that indicate the direction of **X**, **Y**, and **Z**.

8. Drag the cursor toward the left. You will notice that the **Z** symbol is displayed below the cursor as you drag the cursor. This indicates that you are creating the line in the **Z** direction. Release the left mouse button when the value of the length of line displays a value close to **40**.

9. Move the cursor to the endpoint of the previous line and when the cursor turns yellow in color, press and hold down the right mouse button.

10. Drag the cursor to the right. The **X** symbol is displayed below the cursor, which indicates that the line is being created in the **X** direction. Release the left mouse button when the length of the line above the line cursor shows a value close to **100**.

11. Move the cursor to the endpoint of the previous line. When the cursor snaps to the endpoint, press and hold down the left mouse button.

12. Press the TAB key to switch to the **XY** plane. Drag the cursor vertically upwards. The **Y** symbol will be displayed below the cursor, suggesting that the line is being created in the **Y** direction.

13. Release the left mouse button when the value above the cursor shows a value close to **85**.

14. Similarly, create the remaining sketch and add the required relations and dimensions to the sketch. The final sketch is displayed in Figure 9-113.

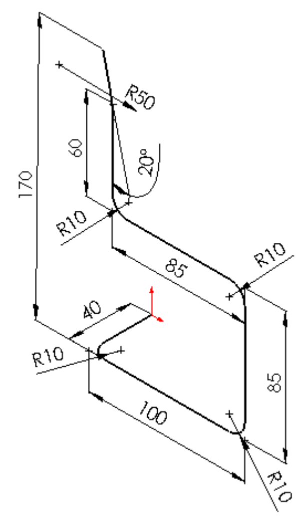


Figure 9-113 *Sketch of the 3D path*

15. Add a **Coincident** relation between the upper point of the 3D sketch and **Plane1**.

16. Exit the sketching environment.

Creating the Profile of the Sweep Feature

As discussed earlier, the sweep feature will be created by sweeping the profile along a 3D path. You have created the path and now you need to create the profile of the sweep feature.

1. Select the **Front** plane as the sketching plane and invoke the sketching environment.

   Since the **Front** plane is normal to the 3D path, therefore, you do not need to create a reference plane. If any one of the default plane is not normal to the sweep profile, then you need to create a reference plane normal to the path.

2. Create the sketch of the profile of sweep feature and add the required dimension to the sketch. Refer to Figure 9-112.

3. Exit the sketching environment.

Sweeping the Profile Along the 3D Path

After creating the 3D path and the profile of the sweep feature, you need to sweep the profile along the 3D path using the **Sweep** tool.

1. Choose the **Sweep** button from the **Features** toolbar. The **Sweep PropertyManager** is invoked and you are prompted to select the sweep profile.

2. Select the sketch of the profile. The selected profile will be displayed green in color and the profile callout is also displayed.

   You are prompted to select the path of the sweep feature.

3. Select the path of the sweep feature. The path will be displayed in red color and the path callout is also displayed. The preview of the sweep feature is also displayed in the drawing area.

   As evident from Figure 9-112, the frame of the chair is made from a hollow pipe. Therefore, you need to create a thin sweep feature to create a hollow chair frame.

4. Invoke the **Thin Feature** rollout and set the value of the **Thickness** spinner to **1**.

5. Choose the **Reverse Direction** button from the **Thin Features** rollout to reverse the direction of thin feature creation.

6. Choose the **OK** button from the **Sweep PropertyManager** to end feature creation.

   The model after creating the sweep feature is displayed in Figure 9-114.

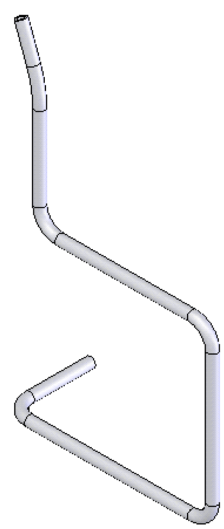


Figure 9-114 *Sweep feature created by sweeping a profile along a 3D path*

7. Using the mirror tool, mirror the sweep feature along the **Front** plane. The model after mirroring is displayed in Figure 9-115. The **FeatureManager Design Tree** of the model is shown in Figure 9-116.

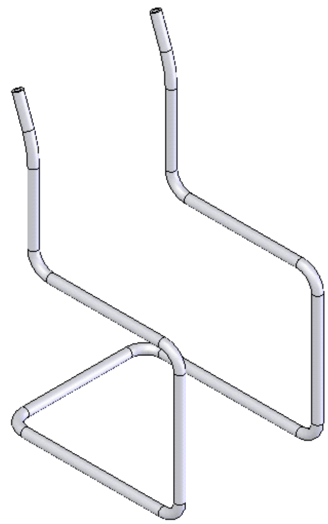


Figure 9-115 *The final model*

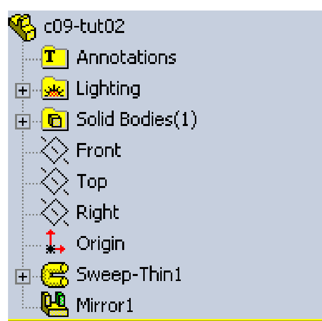
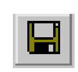


Figure 9-116 *The **FeatureManager Design Tree** for Tutorial 2*

Saving the Model

Next, you need to save the model.



1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below and close the file

   \My Documents\SolidWorks\c09\c09-tut02.SLDPRT.

Tutorial 3

In this tutorial you will create the spring shown in Figure 9-117. The dimensions of the spring are shown in Figure 9-118. **(Expected time: 45 min)**

The steps to be followed to complete this tutorial are discussed next:

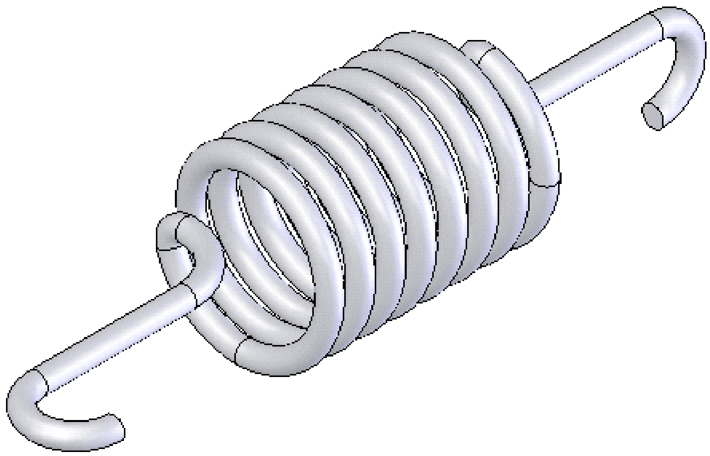


__Figure 9-117__ Model for Tutorial 3

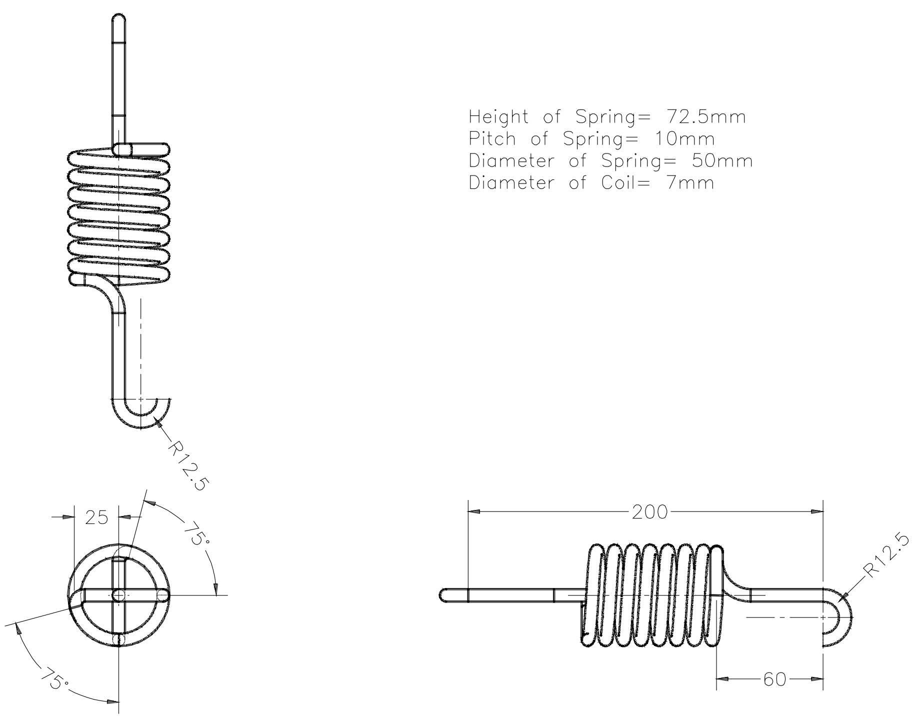


__Figure 9-118__ Views and dimensions of the model for Tutorial 3

a. First you need to create the helical path of the spring, refer to Figure 9-119.
b. Create the side clips of the spring, refer to Figure 9-122.
c. Combine the end clips and the helical curve to create a single curve using the **Composite Curve** option, refer to Figure 9-123.
d. Create the profile on the plane normal to the curve and sweep the profile along the curve, refer to Figures 9-124 and 9-125.

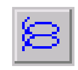
Creating the Helical Curve

For creating this model first you need to create the helical path of the spring. For creating the helical path first you need to create a circular sketch. This sketch will define the diameter of the spring.

1. Start a new SolidWorks document in the Part mode and invoke the sketching environment.

2. Draw a circle of diameter 50. Change the view to isometric view.

3. Choose the **Helix** button from the **Curves** toolbar or choose **Insert > Curve > Helix** from the menu bar.



The **Helix Curve** dialog box is displayed and the preview of the helical curve with the default values is displayed in the drawing area.

4. Select the **Height and Pitch** option from the **Defined by** drop-down list of the **Helix Curve** dialog box.

5. Set the value of the **Height** spinner to **72.5** and the value of the **Pitch** spinner to **10**.

You will observe that the preview of the helical curve is updated automatically when you modify the values in the spinners.

6. Set the value of the **Starting angle** spinner to **0** and choose the **OK** button from the **Helix Curve** dialog box.

The helical curve created using the above steps is shown in Figure 9-119.

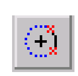
Creating the Sketch of the End Clips of the Spring

After creating the helical curve, you need to create the sketch that defines the path of the end clips. There are two end clips in this spring, and each end clip is created using two sketches. First, you will create the right end-clip and then the left end-clip.

1. Select the **Front** plane as the sketching plane and invoke the sketching environment.

The first sketch of the right end-clip consists of two arcs. The first arc will be created using the **Centerpoint Arc** tool and the second arc will be created using the **3 Pt Arc** tool.

2. Choose the **Centerpoint Arc** button from the **Sketch Tools** toolbar and specify the centerpoint of the arc at the origin.



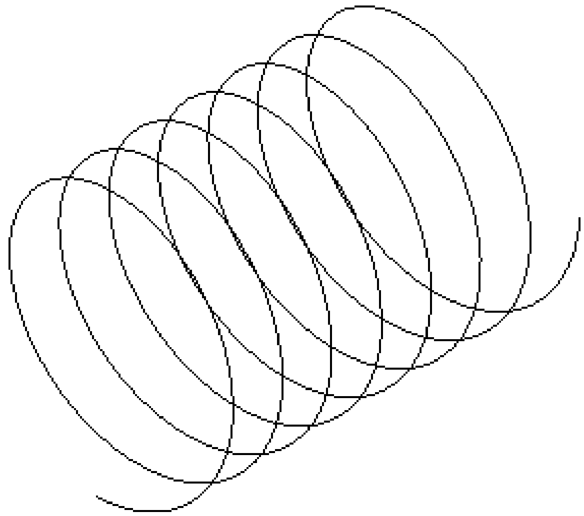
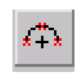


Figure 9-119 Helical curve

3. Move the cursor toward the right and specify the start point of the arc when the radius of the reference circle shows a value close to **25**.



4. Move the cursor in the counterclockwise direction. Specify the endpoint of the arc when the value of the angle above the cursor shows a value close to **75**.

5. Choose the **3 Pt Arc** button from the **Sketch Tools** toolbar. Specify the start point of the arc on the endpoint of the previous arc. Create the arc as shown in Figure 9-120.

6. Add the **Pierce** relation between the start point of the first arc and the helical path. Add the other relations and the dimensions to fully define the sketch. The fully defined sketch is shown in Figure 9-120.

7. Exit the sketching environment.

   After creating the first sketch of the right end-clip, you need to create the second sketch of the right end-clip. The second sketch of the right end-clip will be created on the **Right** plane.

8. Select the **Right** plane and invoke the sketching environment.

9. Create the sketch as shown in Figure 9-121. You need to apply the **Pierce** relation between the helical curve and the end point of left arc of the sketch.

10. Add a **Tangent** and **Coincident** relation between the left arc of the sketch and the sketch created previously. Add the required relations and dimensions to the sketch. Sketch after applying all the relations and dimensions is shown in Figure 9-121.

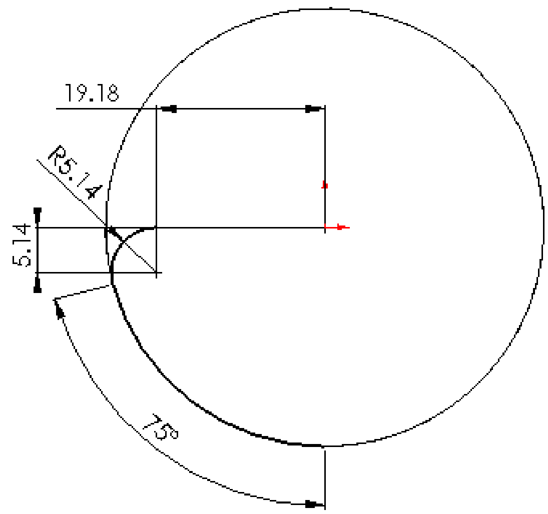


Figure 9-120 *First sketch of right end-clip*

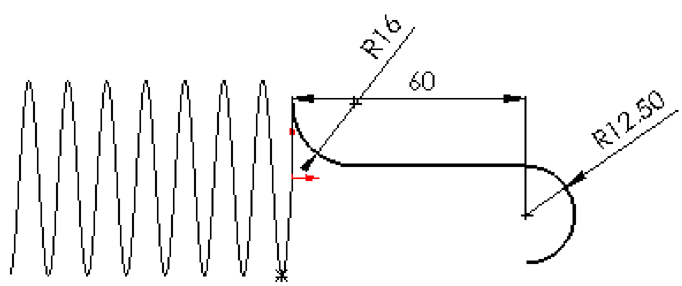


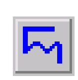
Figure 9-121 *Second sketch of right end-clip*

Similarly, create the sketch of the left end-clip. Figure 9-122 shows the path of the spring after creating the helical curve, right end-clip, and the left end-clip.

Creating the Composite Curve

After creating all the required sketches and the helical curve, you need to combine them from a single curve. This is done because while creating the sweep feature you cannot select more than one curve or sketch as a path.

1. Choose the **Composite Curve** button from the **Curves** toolbar or choose **Insert > Curve > Composite** from the menu bar. The **Composite Curve PropertyManager** is displayed and the confirmation corner is also available.



2. Select all the sketches and the helical curve either from the drawing area or from the **FeatureManager Design Tree** flyout.

3. Choose the **OK** button from the **Composite Curve PropertyManager** or choose **OK** from the confirmation corner.

   The composite curve is created. The composite curve created is displayed in Figure 9-123.

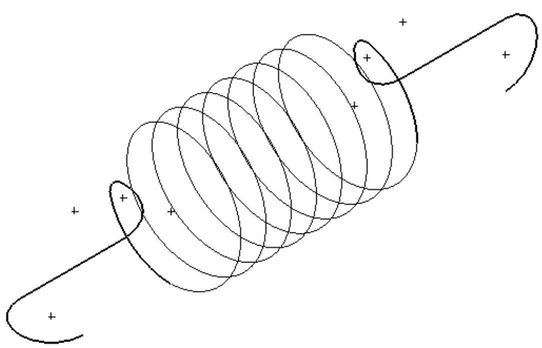


Figure 9-122 *Final path of the spring*

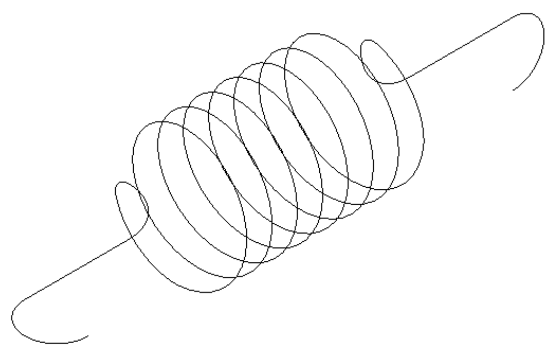


Figure 9-123 *Composite curve*

Creating the Profile for the Sweep Feature

Now, you need to create the profile of the sweep feature. The profile of the sweep feature will be created on a plane normal to the curve on the endpoint of the right end-clip.

1. Create a plane normal to the path and at the endpoint of the right end-clip.

2. Create the profile of the sweep feature and add the **Pierce** relation between the center of the circle and the composite curve. Add the required dimension to the sketch.

3. Exit the sketching environment. Figure 9-124 shows the profile and path of the sweep feature.

Creating the Sweep Feature

You will create a sweep feature to complete the creation of spring.

1. Invoke the **Sweep PropertyManager** and you are prompted to select the sweep profile.

2. Select the sweep profile from the drawing area.

   You are prompted to select the sweep path.

3. Select the path and choose the **OK** button from the **Sweep PropertyManager**.

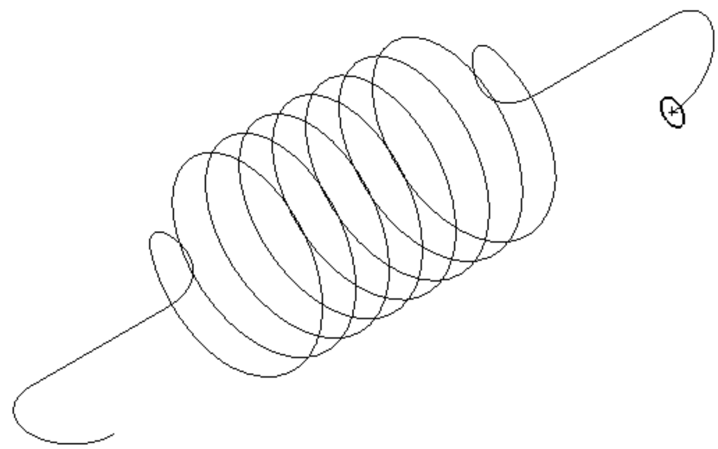


Figure 9-124 *Profile and path of the sweep feature*

The spring created after sweeping the profile along the path is shown in Figure 9-125. The **FeatureManager Design Tree** of the spring is shown in Figure 9-126.

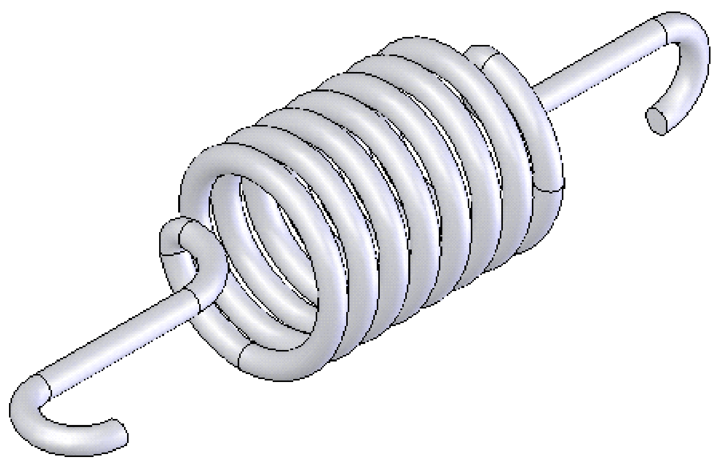


Figure 9-125 *Final model*

Saving the Model

Next, you need to save the model.

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below and close the file

   \My Documents\SolidWorks\c09\c09-tut02.SLDPRT.

Figure 9-126 FeatureManager Design Tree

SELF-EVALUATION TEST

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

1. You need a profile and a path to create a sweep feature. (T/F)

2. The loft feature need at least two sections to create the feature. (T/F)

3. You cannot sweep a closed profile along a closed path. (T/F)

4. You cannot create a thin sweep feature. (T/F)

5. You can also create a loft feature using open sections. (T/F)

6. The __________ **PropertyManager** is used to create a loft feature.

7. The __________ option is used to create a curve by joining the continuous chain of existing sketches, edges, or curves.

8. You have to apply __________ relation between the sketch and the guide curve while sweeping a profile along a path using guide curves.

9. __________ option is used to create a curve by defining coordinates.

10. __________ **PropertyManager** is invoked to create a cut sweep feature.

REVIEW QUESTIONS

Answer the following questions:

1. Using the ___________ rollout you can define the tangency at the start and end in the sweep feature.

2. The __________ rollout is used to create a thin loft feature.

3. You need to invoke __________ dialog box to create a spiral curve.

4. The __________ **PropertyManager** is invoked to create a curve projected on a surface.

5. The __________ dialog box is used to specify the coordinates to create the curve.

6. Which button is selected to invoke the 3D sketching environment?

   (a) **2D Sketch** (b) **3D Sketching Environment**
   (c) **3D Sketch** (d) **Sketch**

7. Which rollout available in the **Sweep PropertyManager** is used to define the tangency?

   (a) **Start/End Tangency** (b) **Tangency**
   (c) **Options** (d) None of these

8. Which button available in the **Features PropertyManager** is used to invoke the **Draft PropertyManager**?

   (a) **Draft** (b) **Taper Angle**
   (c) **Draft Feature** (c) **Draft Angle**

9. In which rollout you can define the pull direction while creating a draft feature using the parting line option?

   (a) **Pull Direction** (b) **Direction of Pull**
   (c) **Reference Direction** (d) **Options**

10. Which button available in the **Guide Curves** rollout is used to display the sections while creating the sweep feature with guide curves.

    (a) **Preview Sections** (b) **Show Sections**
    (c) **Sections** (d) **Preview**

EXERCISES

Exercise 1

Create the model shown in Figure 9-127. The dimensions of the model are shown in Figure 9-128. **(Expected time: 1 hr)**

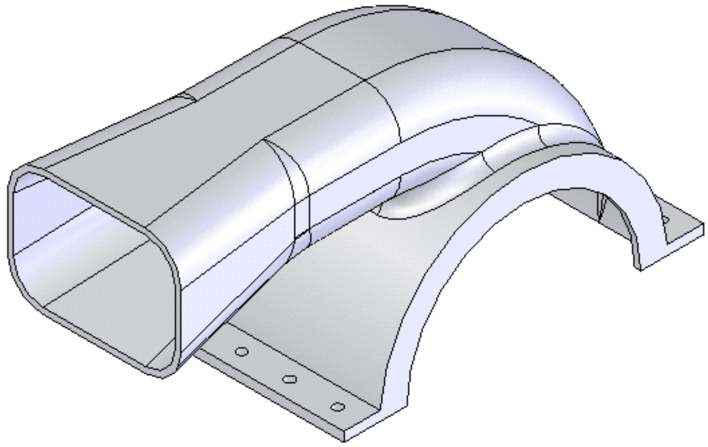


Figure 9-127 *Model for Exercise 1*

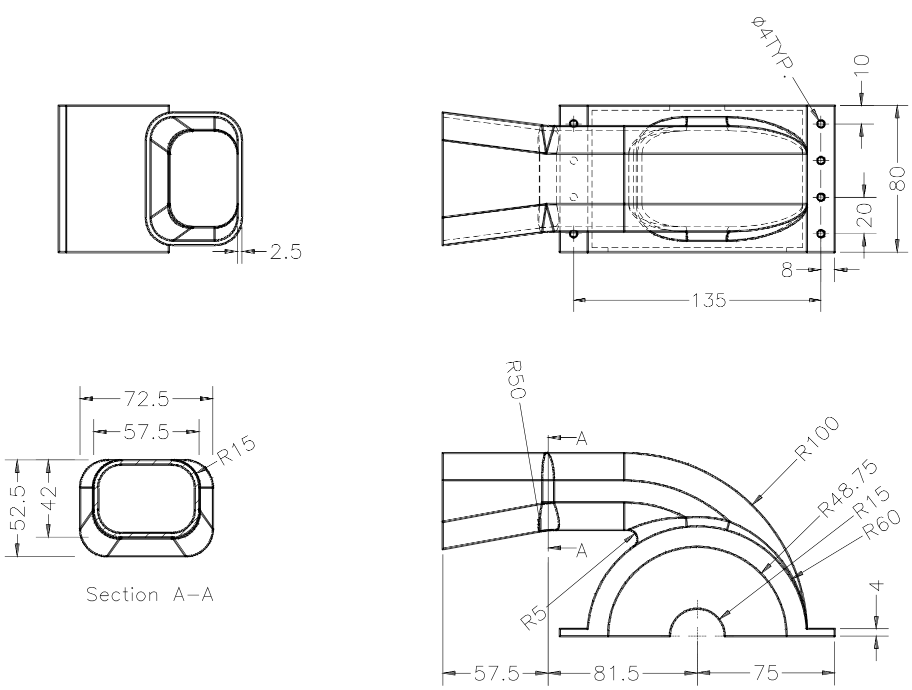
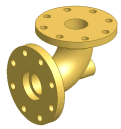


Figure 9-128 *Views and Dimensions of the model for Exercise 1*



Tip. *This model is divided in three major parts. The first part is the base and is created by extruding the sketch to a distance of 80mm using the* ***Mid Plane*** *option.*

The second part of this model is the right portion of the discharge venturi. This portion is created using the sweep feature. The path of the sweep feature will be created on the right plane. You need to create a plane at the left endpoint of the path and will be normal to the path.

The third part of this model is the left portion of the discharge venturi. This will be created using the loft feature. You need to create the first section of the loft feature on the planar face of the sweep feature created earlier. The second section will be created on a plane at an offset distance from the planar face of the sweep feature created earlier. Create a loft feature using the two section created earlier. The other features needed to complete the model are fillets, hole, circular pattern and so on.

Exercise 2

Create the model shown in Figure 9-129. The dimensions of the model are shown in Figure 9-130. **(Expected time: 1 hr)**

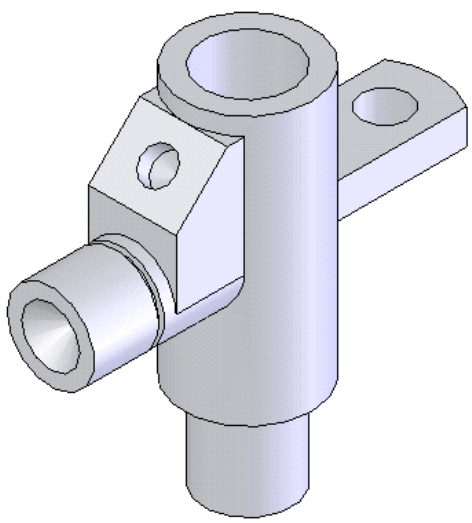


Figure 9-129 *Model for Exercise 2*

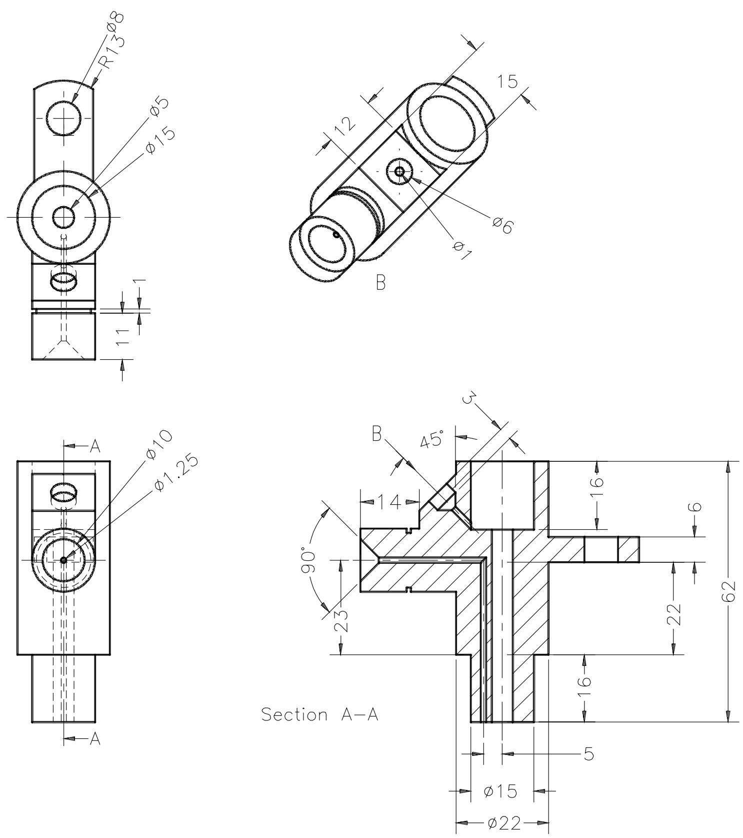


Figure 9-130 *Views and dimensions of the model for Exercise 2*

Answer to Self-Evaluation Test

1. T, **2.** T, **3.** F, **4.** F, **5.** T, **6. Loft**, **7. Composite Curve**, **8. Pierce**, **9. Curve Through Free Points**, **10. Cut-Sweep**