

Chapter 5

Creating Reference Geometries

Learning Objectives

After completing this chapter you will be able to:

- *Create reference plane.*
- *Create reference axis.*
- *Create reference coordinate system.*
- *Create model using other Boss/Base options.*
- *Create model using the contour selection technique.*
- *Create a cut feature.*
- *Create multiple disjoint bodies.*

IMPORTANCE OF THE SKETCHING PLANES

In the previous chapter you created the basic models by extruding or revolving the sketches. All of these models were created on a single sketching plane, the Front plane. But most of the mechanical designs consist of various features such as the sketched features, referenced geometries, and placed features. These features are integrated together to complete a model. Most of these features lie on different planes. When you open a new SolidWorks document and enter the sketching environment and create a sketch, the sketch is created on the default plane, which is the **Front** plane. This is because the Front plane is selected by default when you enter the sketching environment. You can also create the base feature on a plane other than the default plane. To create additional features, you need to select an existing plane, or a planar surface, or you have to create a plane that will be used as a sketching plane to create the sketch. Consider the model shown in Figure 5-1, which is created using various features.

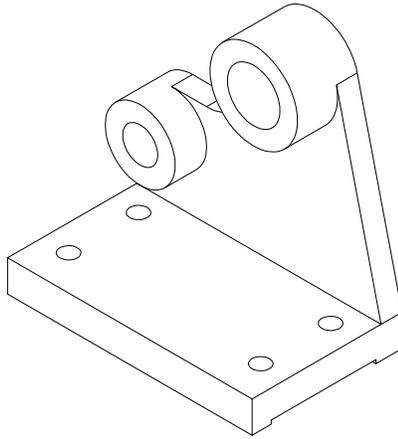


Figure 5-1 A multifeatured model

The base feature of this model is shown in Figure 5-2. The sketch for the base feature is drawn on the Top plane. After creating the base feature you will have to create the other features, which include sketched features, placed features, and referenced features, see Figure 5-3. The boss features, and cut features, are the sketched features that require sketching planes where you can draw the sketch of the features.

It is evident from Figure 5-3 that the features added to the base feature are not created on the same plane on which the sketch for the base feature is created. Therefore, to draw the sketches of other sketched features you will need to define other sketching planes.

REFERENCE GEOMETRY

The reference geometry features are the features that consist of no mass and no volume. These are available only to assist you in the creation of the models. They act as a reference for drawing the sketches for features, defining the sketch plane, assembling the components, references for various placed features and sketched features, and so on. The reference geometry

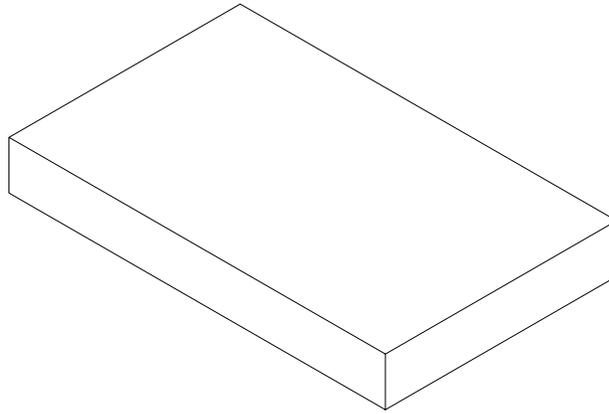


Figure 5-2 Base feature for the model

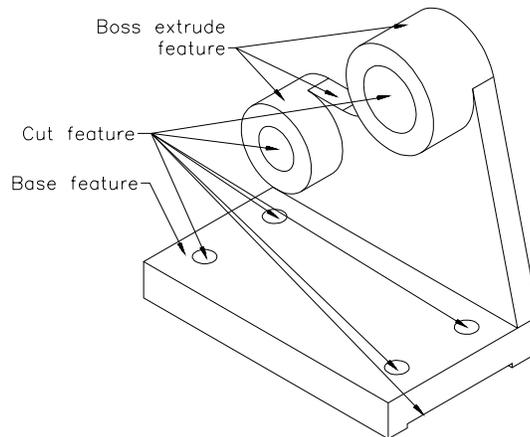


Figure 5-3 Model after adding other features

is widely used in creating complex models; therefore, one must have a good understanding of reference geometry. In SolidWorks reference geometry exists as reference planes or planes, reference axis, and reference coordinate system.

Reference Planes

Generally, all the engineering components or designs are multi- featured models. Also, as discussed earlier, all the features of a model are not created on the same plane on which the base feature is created. Therefore, you have to select one of the default planes or create a new plane that will be used as the sketching plane for the second feature. It is clear from the above discussion that either you can use the default planes as the sketching plane or you can create a plane that can be used as a sketching plane. The default planes and the creation of a new plane are discussed next.

Default Planes

When you create a new SolidWorks part document, SolidWorks provides you with three default planes. These three planes are

1. Front plane
2. Top plane
3. Right plane

The orientation of the component depends on the sketch of the base feature. Therefore, it is recommended that you carefully select the sketching plane for creating the sketch for the base feature. The sketch plane for drawing the sketch of the base feature can be one of the three datum planes provided by default. If you invoke the sketcher environment without selecting any sketching plane, the sketch is created on the Front plane by default. You can select the sketching planes before invoking the sketcher environment from the **FeatureManager Design Tree** available on the left of the graphics screen. The **FeatureManager Design Tree** with three default planes is displayed in Figure 5-4.



Tip. You can display the default planes in the drawing area using the following procedure:

Press and hold the **CTRL** key from the keyboard and one by one select the **Front**, **Top**, and the **Right** planes from the **FeatureManager Design Tree**. Right-click to display the shortcut menu and choose the **Show** option from this shortcut menu to display the planes in the drawing area. Choose the **Isometric** button from the **Standard Views** toolbar. The default planes are transparent and the boundary of the planes is displayed in gray color. Generally, it is not recommended that you display the planes because sometimes may interfere while selecting entities.

To display the shaded planes, choose **Tools > Options** from the menu bar to invoke the **System Options - General** dialog box. Choose the **Display/Selection** option from the left of this dialog box; the name of the dialog box will be displayed as the **System Options - Display/Selection** dialog box. Select the **Display shaded planes** check box from this dialog box and choose the **OK** button.

After displaying the planes in shaded form, invoke the **Rotate View** tool and drag the **Rotate view** cursor to rotate the shaded planes. You will observe that one side of the plane is displayed in green color and the other side of the plane is displayed in red color. The green side of the plane symbolizes the positive side and the red side of the plane symbolizes the negative side. This means that when you create an extrude feature, the depth of extrusion will be assigned to the positive side of the plane by default. When you create a cut feature the depth of the cut feature is assigned to the negative direction by default.

When you work in **Assembly** mode of SolidWorks, you will also find three default assembly planes. The default assembly planes will be discussed in the later chapters.

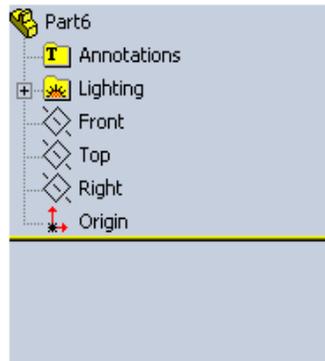


Figure 5-4 FeatureManager Design Tree with default planes



Tip. When you have to create any multifeatured solid model, first try to visualize the number of features in that model and then decide which feature in the model can be considered as the base feature.

Creating New Planes

Toolbar: Reference Geometry > Plane
Menu: Insert > Reference Geometry > Plane



Reference planes or planes are used to draw sketches for the sketched features. These planes are also used to create a placed feature like holes, reference an entity or a feature, and so on. The plane can also be selected to draw the sketch for a sketched feature and these planes are known as sketch planes. You can also select a planar face of a feature that will be used as a sketching plane. Generally, it is recommended that you use the planar faces of the features as the sketching planes. However, sometimes you have to create a sketch at a plane that is at some offset distance from a plane or a planar face. In this case you have to create a new plane at an offset distance from a sketching plane or a planar face.

Consider another case where you have to define a sketching plane tangent to a cylindrical face of a shaft. You have to create a plane tangent to the cylindrical face of the shaft and this plane will be used as a sketching plane. In SolidWorks, there are six methods to create planes. Choose the **Plane** button from the **Reference Geometry** toolbar to invoke the **Plane PropertyManager**. The confirmation corner is also displayed at the top right corner of the drawing area. The **Plane PropertyManager** is displayed in Figure 5-5. Various options available in the **Plane PropertyManager** to create new planes are discussed next.

Creating a Plane Using Through Lines/Points

The **Through Lines/Points** option is used to create a plane that passes through an edge and a point, an axis and a point, or a sketch line and a point. Using this option you can also create a plane that passes through three points. The selected point can be a sketched point or a vertex. To create a plane using this option, invoke the **Plane PropertyManager** and choose the **Through Lines/Points** button and select the required entities from the drawing area. The name of the selected entities will be displayed in the **Reference Entities**

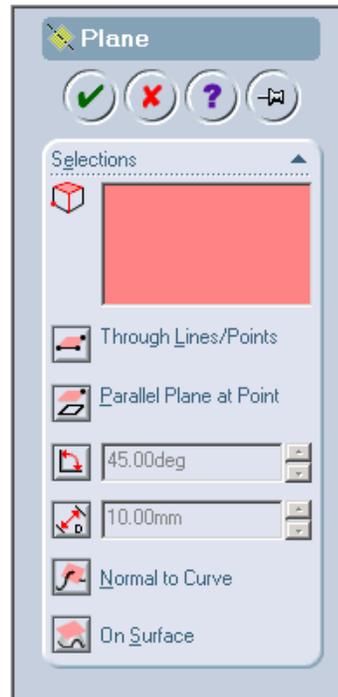


Figure 5-5 Plane PropertyManager

selection list. Choose the **OK** button from the **Plane PropertyManager**. Figure 5-6 shows an edge and a vertex selected to create a plane. The resultant plane is displayed in Figure 5-7. The creation of a new plane by selecting three points is displayed in Figures 5-8 and 5-9.

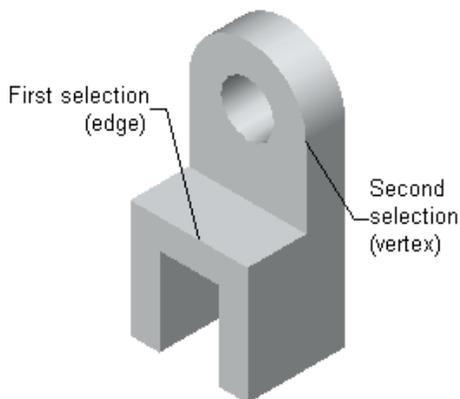


Figure 5-6 Selecting the edge and vertex

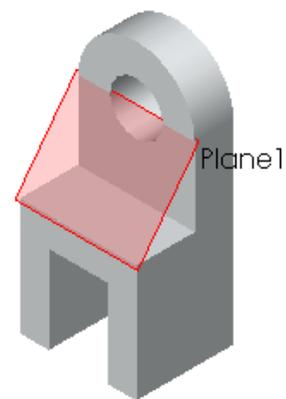


Figure 5-7 Resultant plane

Creating a Plane Parallel to an Existing Plane or Planar Face

The **Parallel Plane** option is used to create a plane that is parallel to another plane or a planar surface and passes through a point. To create a plane using this option, invoke the

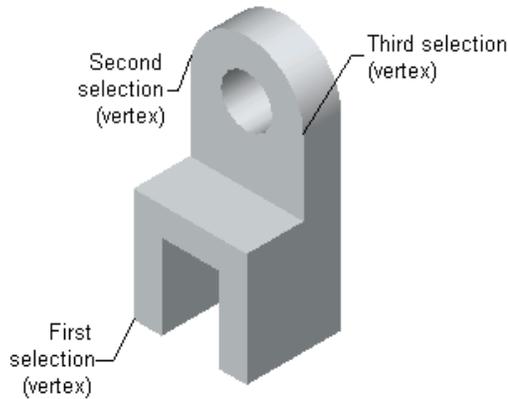


Figure 5-8 Selecting the vertices

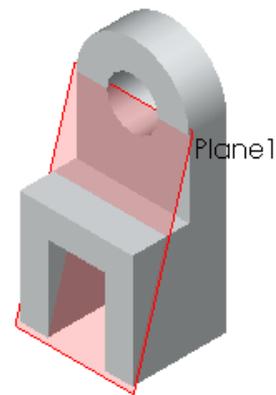


Figure 5-9 Resultant plane

Plane PropertyManager and then choose the **Parallel Plane** button from this **PropertyManager**. Now, select the plane or a planar face to which the newly created plane will be parallel. Then select a sketched point or midpoint of an edge. The newly created plane will pass through this point. Choose the **OK** option. Figure 5-10 shows a planar face and the point selected to create the parallel plane. Figure 5-11 shows the resultant plane.

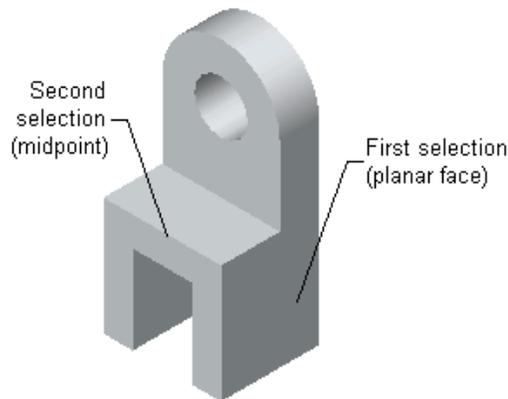


Figure 5-10 Selecting the planar face and edge

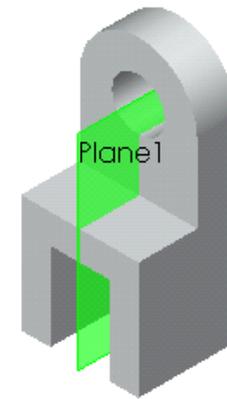


Figure 5-11 Resultant plane

Creating a Plane at an Angle to an Existing Plane or a Planar Face

The **At Angle** option is used to create a plane at an angle to the selected plane or a planar face and passes through an edge, axis, or sketched line. To create a plane at an angle, choose the **At Angle** button from the **Plane PropertyManager**. The **Angle** spinner is invoked. The **Reverse direction** check box and **Number of Plane to Create** spinner appear below the **Distance** spinner in the **Plane PropertyManager** as shown in Figure 5-12. Now, using the left mouse button select an edge, an axis, or a sketched line through which the plane will pass. Next, you have to select a planar face or a plane to define the angle. Now, set the angle value using the **Angle** spinner. You can reverse the direction of plane creation by selecting the **Reverse direction** check box. You can also



Figure 5-12 Plane PropertyManager with At Angle option selected

create multiple planes by increasing the value of the **Number of Planes to Create** spinner. Figure 5-13 shows a planar face and edge selected. Figure 5-14 shows the resultant plane created at an angle of 45° to the selected plane.

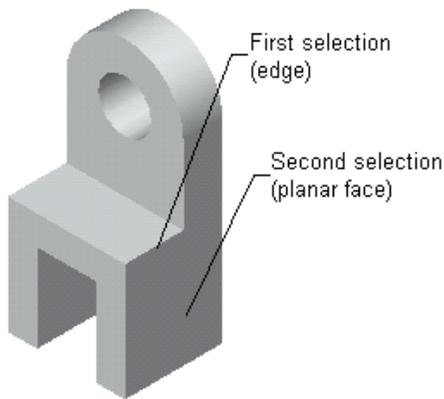


Figure 5-13 Selecting the edge and the planar face

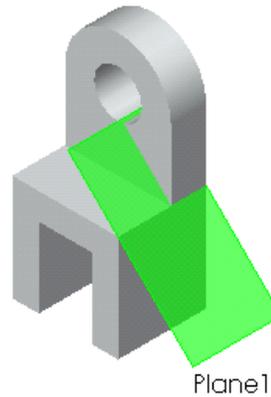


Figure 5-14 Resultant plane

Creating a Plane Using Offset Distance

The **Offset Distance** option is used to create a plane at an offset distance from a selected plane or planar face. To create a plane using this option, choose the **Offset Distance** button from the **Plane PropertyManager**. When you invoke this option, the **Distance** spinner is invoked. Also, the **Reverse direction** check box and the **Number of Planes to Create** spinner are displayed below the **Distance** spinner in the **Plane PropertyManager**. Select a plane or a planar face and set the value of distance in the **Distance** spinner and choose the **OK** button from the **Planar PropertyManager**. You can reverse the direction of plane creation by selecting the **Reverse direction** check box. You can also create multiple planes by increasing the value of the **Number of Planes to Create** spinner. Figure 5-15 shows a plane selected to create parallel plane and Figure 5-16 shows the resultant plane created at the required offset.

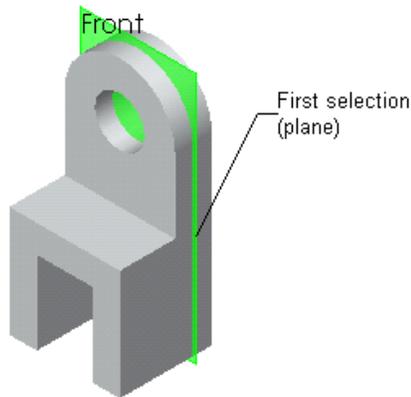


Figure 5-15 Selecting the plane

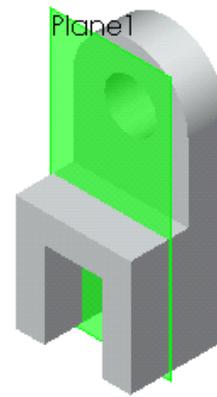


Figure 5-16 Resultant plane



Tip. You can also create the planes by dynamically dragging an existing plane. For creating a plane by dragging, you do not need to invoke the **Plane PropertyManager**. Using the left mouse button select the plane from the **Feature Manager Design Tree** or **Drawing** area. Press and hold down the CTRL key on the keyboard. Now, move the cursor to the selected plane and when the cursor is replaced by the move cursor, press and hold down the left mouse button and drag the cursor. You will notice that the value of distance in the **Distance** spinner of the **Plane PropertyManager** will modify and the preview of the plane is displayed in the drawing area. After dragging the plane to a required location release the left mouse button. Right-click and choose the **OK** option or choose the **OK** button from the **Plane PropertyManager**.

You can also create a plane at an angle by dragging. To create a plane at an angle by dragging, select an edge or an axis and an existing plane. Now, hold the CTRL key and drag the mouse. Enter the angle value in the **Angle** spinner.

Creating a Plane Normal to Curve

This option is used to create a plane normal to a curve. To create a plane normal to a curve, choose the **Normal to Curve** button from the **Plane PropertyManager**. When you choose the **Normal to Curve** button, the **Set origin on curve** check box is displayed. Now, select a curve such as a sketched arc, circle, spline, or circular edge. As soon as you select the curve, the preview of the plane is displayed in the drawing area. Choose the **OK** button from the **Plane PropertyManager** or choose the **OK** icon from the confirmation corner. Figure 5-17 shows a curve to create the plane. The **Set origin on curve** check box is selected to place the origin on curve. By default, this check box is clear. Figure 5-18 shows the resultant plane created normal to the selected curve.



Note

If you select the curve near the first endpoint to create a plane normal to the curve, the plane will be created at the first endpoint of the curve. If you select the curve near the second endpoint, the plane will be created normal to that curve near the second endpoint.

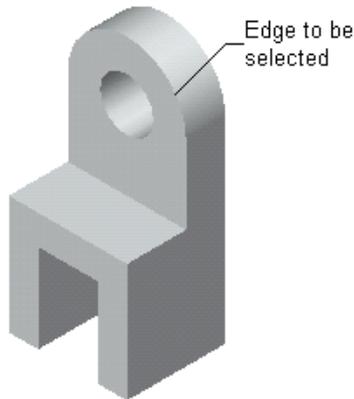


Figure 5-17 Edge to be selected

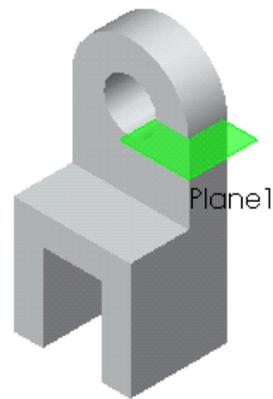


Figure 5-18 Resultant plane

Creating a Plane On Surface

The **On Surface** option is used to create a plane passing through a point on the selected plane or planar surface. To create a plane on surface, choose the **On Surface** button from the **Plane PropertyManager** and select the surface on which you want to create the plane. Next, select the sketched point. The preview of the plane is displayed in the drawing area and you have to right-click to choose the **OK** option. If the sketch is created on a plane at an offset distance from the selected surface, the **Project to nearest location on surface** and **Project onto surface along sketch normal** radio buttons, and the **Other Solutions** button are displayed on the **Plane PropertyManager**. Select any of the radio buttons according to requirement. You can also view the other solutions of the plane creation using the **Other Solutions** button from the **Plane PropertyManager**. Figure 5-19 shows the selection of references for the plane creation and Figure 5-20 shows the resultant plane created.

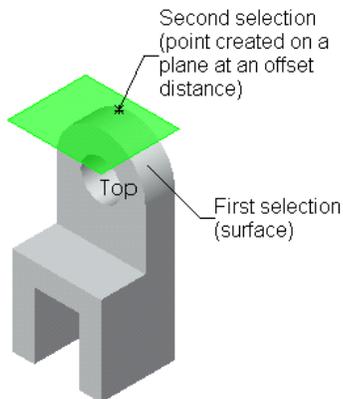


Figure 5-19 References to be selected

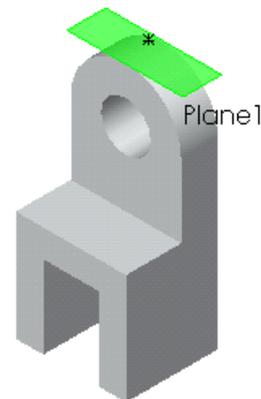


Figure 5-20 Resultant plane

Creating Reference Axis

Toolbar: Reference Geometry > Axis
Menu: Insert > Reference Geometry > Axis



The **Reference Axis** option is used to create a reference axis or construction axis. These axes are the parametric lines passing through a model, feature, or reference entity. A reference axis is used to create reference planes, coordinate systems, circular patterns, and for applying mates in the assembly. These are also used as reference while sketching, or creating features. The reference axes are displayed in the model as well as in the **Feature Manager Design Tree**. When you create a circular feature, a temporary axis is automatically created. You can display the temporary axis by choosing **View > Temporary Axis** from the menu bar. In SolidWorks you have to invoke the **Reference Axis** dialog box to create the reference axis. You can invoke the **Reference Axis** dialog box using the **Axis** button from the **Reference Geometry** toolbar or by choosing **Insert > Reference Geometry > Axis** from the menu bar. The **Reference Axis** dialog box is displayed in Figure 5-21. The various options available in this dialog box are discussed next.

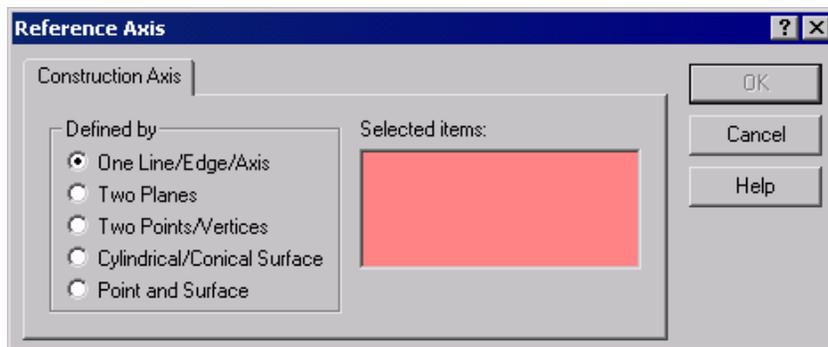


Figure 5-21 The Reference Axis dialog box

Creating a Reference Axis Using One Line/Edge/Axis

The **One Line/Edge/Axis** option available in the **Defined by** area of the **Reference Axis** dialog box is used to create a reference axis by selecting a sketched line or construction line, edge, or temporary axis. To use this option, invoke the **Reference Axis** dialog box; the **One Line/Edge/Axis** radio button is selected by default. Select a sketched line, edge, or a temporary axis. The name of the selected entity is displayed in the **Selected items** display area and the preview of the reference axis is displayed in the drawing area. Now, choose the **OK** button from the **Reference Axis** dialog box. Figure 5-22 shows a construction line selected as a reference for creating the axis. Figure 5-23 shows an axis created using this option.



Tip. If the axis is not displayed in the drawing area, choose **View > Axes** from the menu bar.

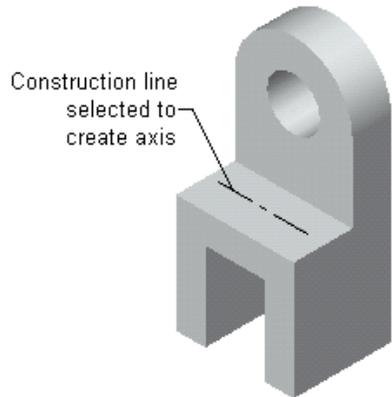


Figure 5-22 Line to be selected



Figure 5-23 Resultant reference axis

Creating a Reference Axis Using Two Planes

Using the **Two Planes** option you can create a reference axis at the intersection of two planes. To create a reference axis using this option, invoke the **Reference Axis** dialog box. Select the **Two Planes** option and then select two planes, two planar faces, or a plane and a planar face. The preview of the axis is displayed in the drawing area. Choose the **OK** button from the **Reference Axis** dialog box. Figure 5-24 shows two planes selected and Figure 5-25 shows the resultant reference axis created using this option.

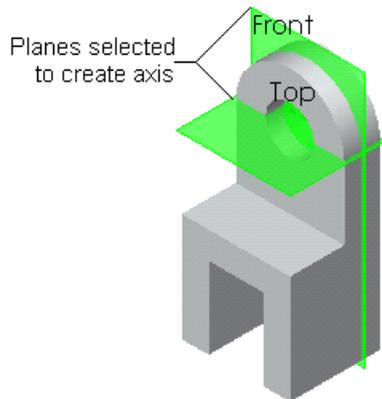


Figure 5-24 Planes to be selected

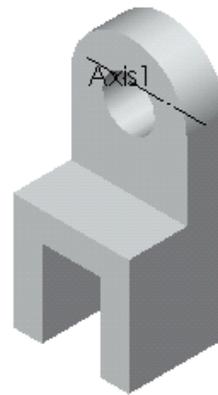


Figure 5-25 Resultant reference axis

Creating a Reference Axis Using Two Points/Vertex

Using the **Two Points/Vertex** you can create a reference axis that passes through two points or two vertices. To create a reference axis using this option, invoke the **Reference Axis** dialog box. Select the **Two Points/Vertex** radio button from this dialog box and then select two points or two vertices using the left mouse button. The preview of the reference axis is displayed in the drawing area. Choose the **OK** button from the **Reference Axis** dialog box. Figure 5-26 shows two vertices to be selected and Figure 5-27 shows the resultant reference axis created using this option.

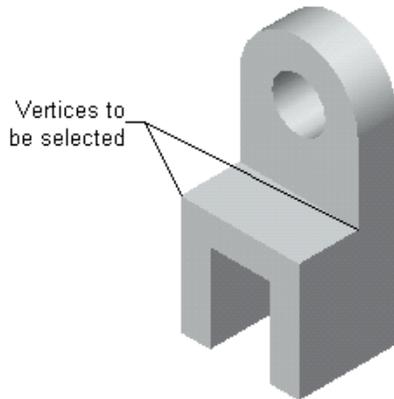


Figure 5-26 Vertices to be selected

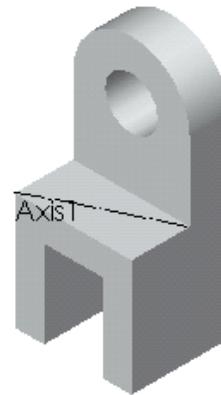


Figure 5-27 Resultant reference axis

Creating a Reference Axis Using Cylindrical/Conical Surface

Using the **Cylindrical/Conical Surface** option you can create a reference axis that passes through the center point of a cylindrical or a conical surface. To create a reference axis using this option, invoke the **Reference Axis** dialog box. Select the **Cylindrical/Conical Surface** radio button from this dialog box. Select the cylindrical or the conical surface using the left mouse button. The preview of the reference axis is displayed in the drawing area. Choose the **OK** button from the **Reference Axis** dialog box. Figure 5-28 shows a cylindrical surface selected and Figure 5-29 shows the resultant reference axis created using this option.

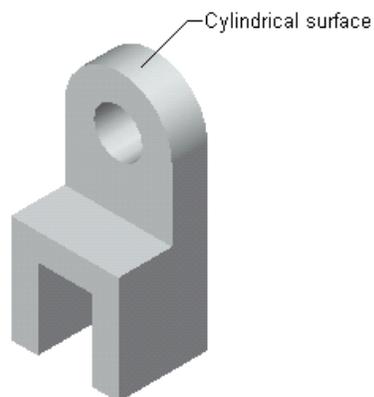


Figure 5-28 Cylindrical surface to be selected

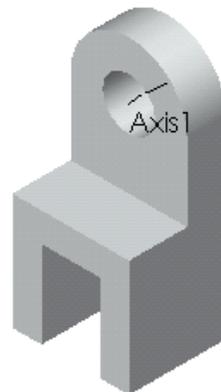


Figure 5-29 Resultant reference axis

Creating a Reference Axis Using Point and Surface

Using the **Point and Surface** option you can create a reference axis that passes through a point and is normal to the selected surface. If the selected surface is a nonplanar surface, the selected point should be created on the surface. To create a reference axis using this option, invoke the **Reference Axis** dialog box. Select the **Point and Surface** radio button from this dialog box. Now, select a point, vertex, or midpoint and then select a surface. The preview of the axis will be displayed in the drawing area. Choose the **OK** button from the **Reference Axis** dialog box. The newly created axis will be normal to the selected surface. Figure 5-30

shows the point and surface selected and Figure 5-31 shows the resultant axis created using this option.

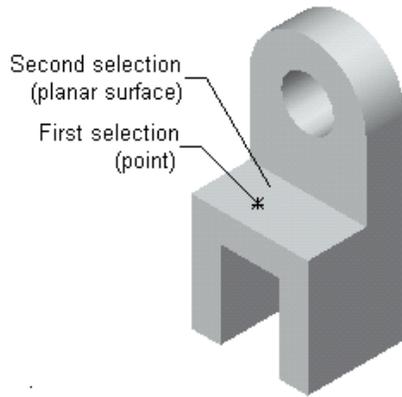


Figure 5-30 Point and surface to be selected

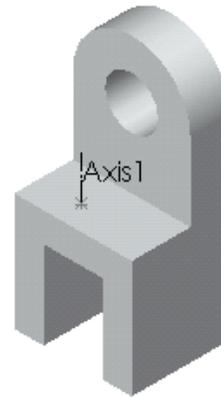


Figure 5-31 Resultant reference axis

Creating Reference Coordinate System

Toolbar: Reference Geometry > Coordinate System
Menu: Insert > Reference Geometry > Coordinate System



In SolidWorks you may need to define some reference coordinate systems other than the default coordinate system for creating features, analyzing the geometry, analyzing the assemblies, and so on. The **Coordinate System** dialog box is used to create the reference coordinate systems. You can invoke this dialog box using the **Reference Axis** button from the **Reference Geometry** tool bar or by choosing **Insert > Reference Geometry > Coordinate System** from the menu bar. The **Coordinate System** dialog box is shown in Figure 5-32.

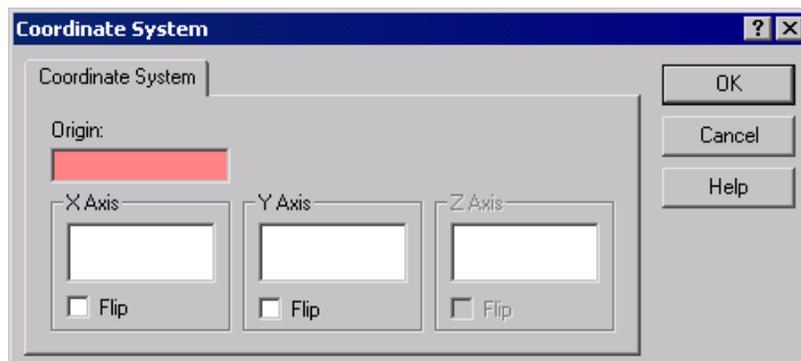
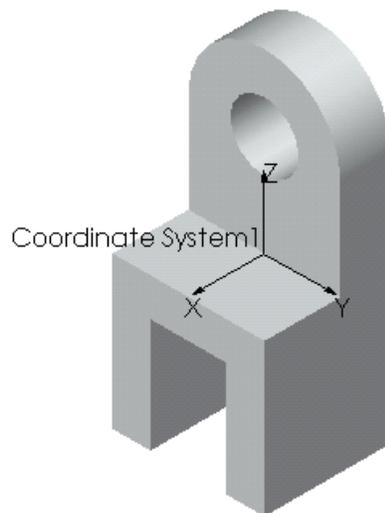


Figure 5-32 The Coordinate System dialog box

As soon as you invoke this dialog box a coordinate system in red color is displayed at the origin of the current document. For creating a new coordinate system you need to select a point that will be selected as the origin for the new coordinate. Therefore, after invoking this

dialog box, select a point, vertex, or endpoint to define the origin of the coordinate system. The name of the selected entity will be displayed below the **Origin** display area. You will also observe that the coordinate system displayed in red color at the origin will be shifted to the newly selected point. If the default orientation of the coordinate system is according to the requirement, choose the **OK** button from the **Coordinate System** dialog box.

You can also select the edges, points, axis, and so on to define the X, Y, and Z directions. To define the X direction, select the **X Axis** display area in the dialog box. It changes to red color. Now, select the edge, axis, vertex, or point to define the direction of the X axis. Using the **Flip** check box available under the **X Axis** display area, you can reverse the X direction. Similarly, you can define Y and Z directions. After defining all the references, choose the **OK** button from the **Coordinate System** dialog box. Figure 5-33 displays a reference coordinate system created using the **Coordinate System** dialog box.



*Figure 5-33 A coordinate system created using the **Coordinate System** dialog*

OTHER BOSS/BASE OPTIONS

Some of the boss/base extrusion options were discussed in previous chapters. In this chapter, the remaining boss/base extrusion option are discussed.

End Condition

The various options available in the **End Condition** drop-down list are discussed next.

Through All

The **Through All** option is available in the **End condition** drop-down list only after you create a base feature. After creating a base feature, select or create a plane and choose the **Sketch** button from the **Sketch** toolbar. The sketching environment is invoked. Create the sketch using the standard sketching tools. Now, choose the **Extruded Boss/Base** button from the **Features** toolbar to invoke the **Extrude PropertyManager**. The confirmation corner is also displayed. The preview of the extruded feature is displayed in temporary graphics in the

drawing area with **Blind** option selected by default in the **End Condition** drop-down list. Select the **Through All** option from the **End Condition** drop-down list and the preview of the extruded feature extends from the sketching plane through all existing geometric entities. You can also reverse the direction of extrusion using the **Reverse Direction** button available on the left of the **End Condition** drop-down list. While creating an extruded feature using the **Through All** option, the sketch extrudes through all the existing geometries. You will observe that the **Merge result** check box is displayed in the **Extrude PropertyManager**. This check box is selected by default. Therefore, the newly created extruded feature will merge with the base feature. If you clear this check box, this extruded feature will not merge with the existing base feature, resulting in the creation of another body. The creation of a new body can be confirmed by observing the **Solid Bodies** folder in the **Feature Manager Design Tree**. The value of the number of disjoint bodies in the model is displayed in the bracket on the right of the **Solid Bodies** folder. You can click the (+) sign on the left of the **Solid Bodies** folder to expand the folder. To collapse the folder back, click the (-) sign. Figure 5-34 shows the expanded **Solid Bodies** folder with two bodies.

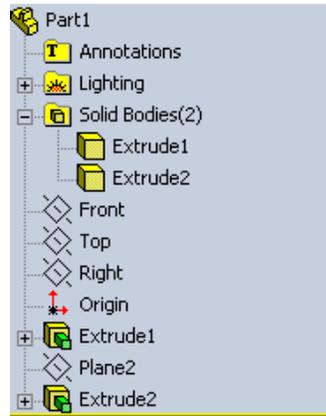


Figure 5-34 The Feature Manager Design Tree displaying expanded **Solid Bodies** folder



Tip. It is recommended that while creating additional features after the base feature, always select the **Merge results** check box in the **Feature PropertyManager**.

The feature created from a multiple disjoint closed contours results in creation of disjoint bodies.

Figure 5-35 displays a sketch created on the sketching plane at an offset distance from the right planar face of the model. Figure 5-36 displays the feature created by extruding the sketch using the **Through All** option.

Up To Next

The **Up To Next** option is used to extrude the sketch from the sketching plane to the next surface that intersects the feature. To create an extruded feature using the **Up To Next** option, you must have a base feature. After creating a base feature, create a sketch by selecting or creating a sketching plane. Invoke the **Extrude PropertyManager**; the preview of the base

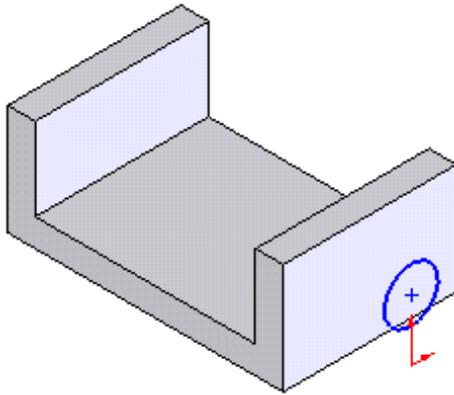


Figure 5-35 A sketch created at an offset distance from the right planar surface

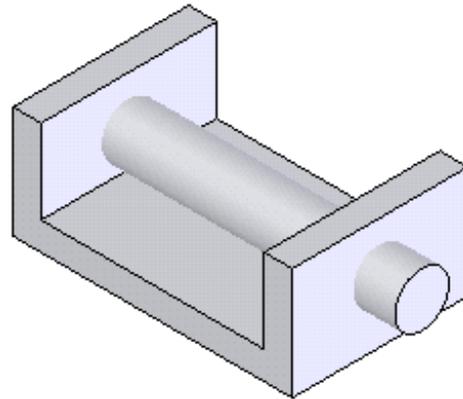


Figure 5-36 Sketch extruded using the **Through All** option

feature is displayed with default options. Select the **Up To Next** option from the **End Condition** drop-down list. You can also reverse the direction of feature creation using the **Reverse Direction** button. The preview of the feature will be modified and the sketch will be displayed as extruded from the sketching plane to the next surface that intersects the feature geometry. Figure 5-37 shows the sketch that will be extruded using the **Up To Next** option and Figure 5-38 shows the resultant feature.

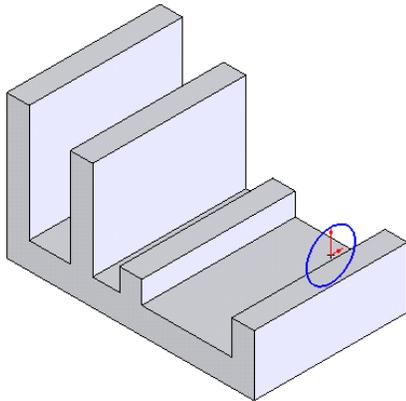


Figure 5-37 A sketch created on the right plane as the sketching plane

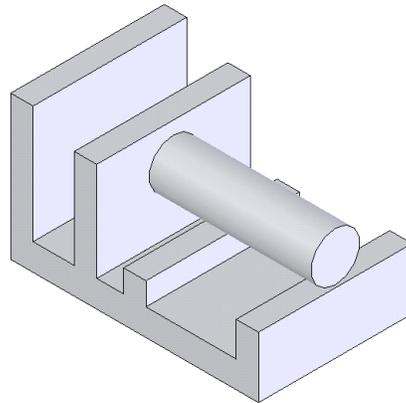


Figure 5-38 Sketch extruded using the **Up To Next** option

Up To Vertex

The **Up To Vertex** option is used to define the termination of the extruded feature at a virtual plane parallel to the sketching plane and passing through the selected vertex. You can also select a point on an edge, or vertices of a sketch. To create an extruded feature using the **Up To Vertex** option create a sketch and invoke the **Extrude PropertyManager**. Select the **Up To Vertex** option from the **Extrude PropertyManager**; the **Vertex** display area is displayed. You can reverse the direction of the extrusion using the **Reverse Direction** button. You are prompted to select a vertex. Using the left mouse button select a vertex; the default preview

of the feature is modified and you can observe that the feature is terminated at the selected vertex. Figure 5-39 shows a sketch created on a plane at an offset distance and Figure 5-40 shows the model in which the sketch is extruded up to the selected vertex.

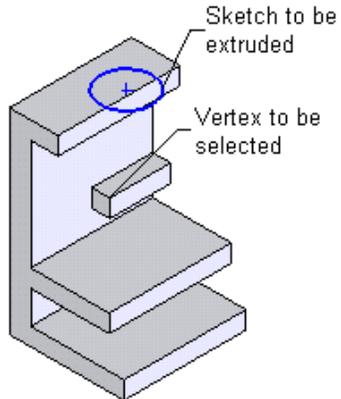


Figure 5-39 Sketch created on a plane created at an offset distance and vertex to be selected

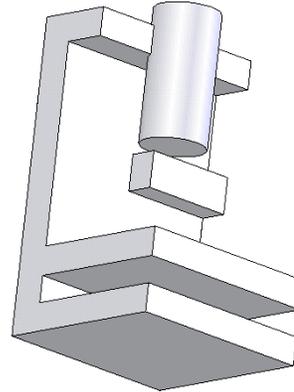


Figure 5-40 Sketch extruded using the *Up To Vertex* option.

Up To Surface

The **Up To Surface** option is used to define the termination of the extruded feature to the selected surface. To create an extruded feature using this option, create a sketch using the normal sketching options and then invoke the **Extrude PropertyManager**. Select the **Up To Surface** option from the **End Condition** drop-down list. The preview of the extruded feature is displayed in temporary graphics. The **Face/Plane** display area is displayed and you are prompted to select a face or a surface. Using the left mouse button select a surface up to which you want to extrude the feature. Figure 5-41 shows the sketch created at an offset distance and the surface to be selected. Figure 5-42 shows the sketch being extruded up to the selected surface.

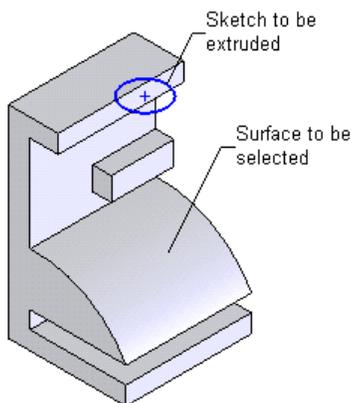


Figure 5-41 Sketch created on a plane created at an offset distance and surface to be selected

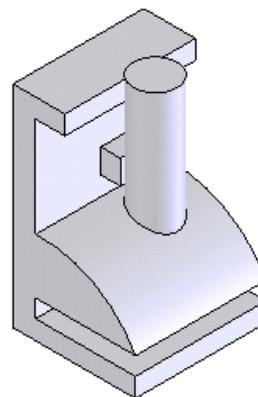
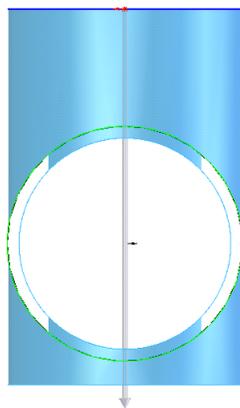


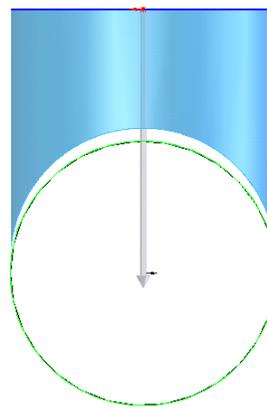
Figure 5-42 Sketch extruded using the *Up To Surface* option.

Offset From Surface

The **Offset From Surface** option is used to define the termination of the extruded feature on a virtual surface created at an offset distance from the selected surface. To create an extruded feature using the **Offset From Surface** option, create a sketch and invoke the **Extrude PropertyManager**. Select the **Offset From Surface** option from the **End Condition** drop-down list. The **Face/Plane** display area is displayed along with the **Offset Distance** spinner. You are prompted to select a face or a surface. Select the surface and set the offset distance in the **Offset Distance** spinner. You can reverse the direction of offset by selecting the **Reverse offset** check box from the **Direction 1** rollout. If the **Translate surface** check box is cleared, the virtual surface created for the termination of the extruded feature will have a concentric relation with the selected surface. Therefore, it reflects the true offset of the selected surface. If the **Translate surface** check box is selected, the center of the virtual surface is at the offset distance from the selected surface. Therefore, a reference surface is created to define the termination of the extruded feature and it does not reflect the true offset of the selected surface. Figure 5-43 shows the front view of the sketch extruded with termination at an offset distance from the selected cylindrical surface with the **Translate surface** check box cleared. Figure 5-44 shows the front view of the extruded feature with the **Translate surface** check box selected.



*Figure 5-43 Sketch extruded using **Offset From Surface** option with **Translate surface** check box cleared*



*Figure 5-44 Sketch extruded using the **Offset From Surface** option with **Translate surface** check box selected*

Up To Body

The **Up To Body** option available in the **End Condition** drop-down list is used to define the termination of the extruded feature to another body. As discussed earlier, if you clear the **Merge results** check box in the **Extrude PropertyManager**, it results in the formation of another body. For creating an extruded feature using the **Up To Body** option, invoke the **Extrude PropertyManager** after creating the sketch and select the **Up To Body** option from the **End Condition** drop-down list. The **Vertex** display area is displayed. Select body to terminate the feature and choose the **OK** button. Figure 5-45 shows the sketch for the extruded feature and a body up to which the sketch will be extruded. Figure 5-46 shows the sketch extruded using the **Up To Body** option.

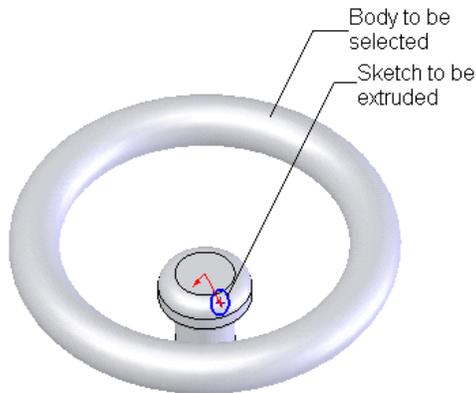


Figure 5-45 Sketch to be extruded and the body to be selected for the extrude feature

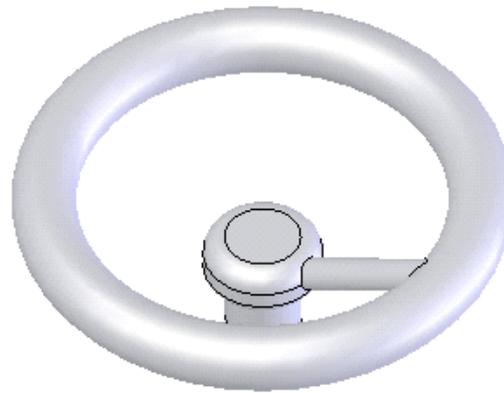


Figure 5-46 Sketch extruded using the *Up To Body* option.

Modeling Using the Contour Selection Method

Modeling using the contour selection method allows you to use the partial sketches for creating the features. Using this method, you can create sketches of the entire model in a single sketching environment and then manipulate the sketches by sharing them between various features. To understand this concept, consider the multi-featured solid model shown in Figure 5-47.

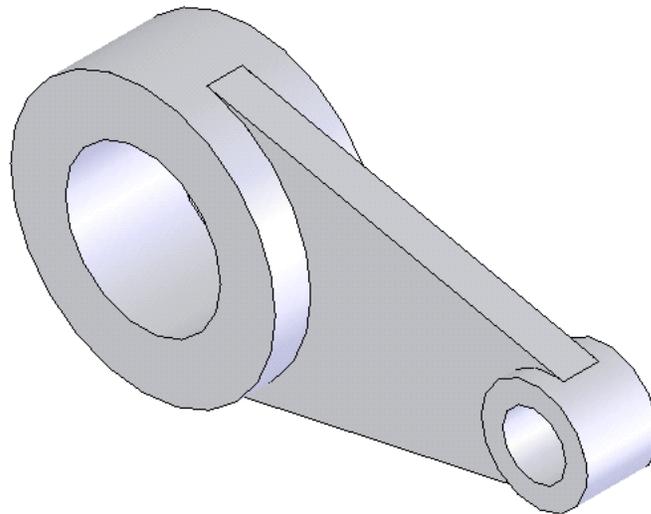


Figure 5-47 Multi-featured solid model

For a multi-featured model as shown above, ideally you first need to create the sketch for the base feature and convert that sketch into the base feature. After that, you have to create the sketch for the second sketched feature and so on. In other words, you have to create the sketch for each sketched feature. But using the contour selection method, you can share the contour created using the sketch for creating features. Figure 5-48 shows the sketch to be created for modeling using the contour selection method.

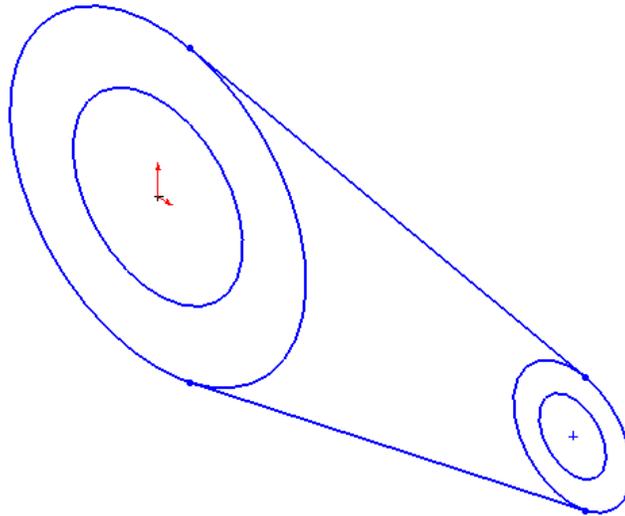


Figure 5-48 Sketch created for creating the model

After creating the entire sketch right-click in the drawing area to invoke the shortcut menu. Choose the **Contour Selection Tool** option from the shortcut menu. The select cursor is replaced by the contour selection cursor and contour selection confirmation corner is displayed. Using the left mouse button select the outer contour of the left larger circle using the contour selection cursor as shown in Figure 5-49.

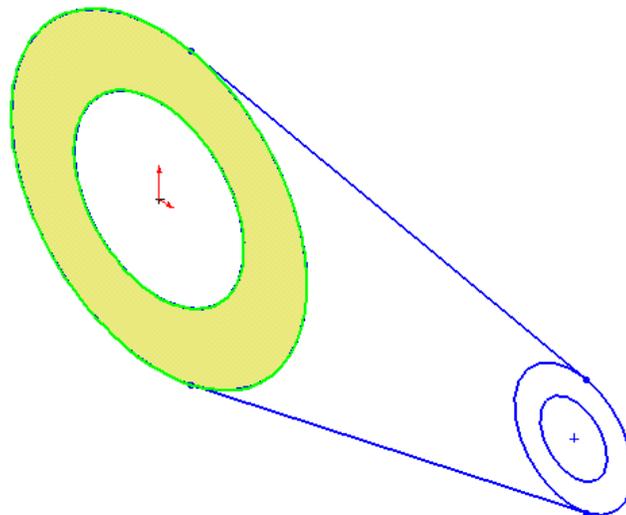


Figure 5-49 Contour selected for creating the extruded feature



Tip. When you move the contour selection cursor in the sketch, the areas where the contour selection is possible are dynamically highlighted in pink color. When you select a specific area using this cursor, the selected contour is displayed in yellow color.

Now, invoke the **Extrude PropertyManager** and extrude the selected contour using the **Mid Plane** option as shown in Figure 5-50.

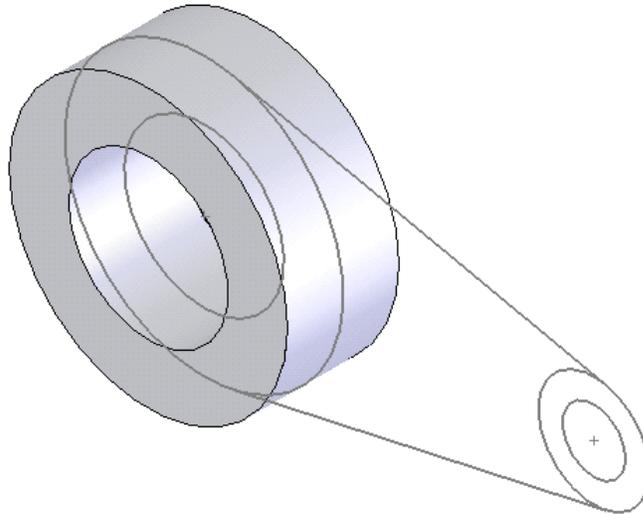


Figure 5-50 Isometric view of the feature created by extruding the selected contour

Now, right-click in the drawing area and choose the **Contour Selection Tool** option from the shortcut menu. Using the contour selection cursor select any entity in the sketch and then select the middle contour of the sketch as shown in Figure 5-51. Invoke the **Extrude PropertyManager** and extrude the selected contour using the mid plane option. Again, invoke the contour selection tool and select an entity in the sketch. Next, select the outer contour of the right circle and extrude the same using the mid plane option.

As the sketches are displayed in the model, you need to hide them. Click the + sign to expand any of the extruded features. Select the sketch icon and right click to invoke the shortcut



Tip. When you select the contour using the contour selection tool and invoke the **Extrude PropertyManager**, you can observe the name of the selected contour in the display area of the **Contour Selection** rollout.

You can select the contours for all the sketched features such as revolve, cut, sweep, loft, and so on.

You can also select the single sketched entity from sketch using the contour selection tool instead of selecting the contour for creating the sketched features.

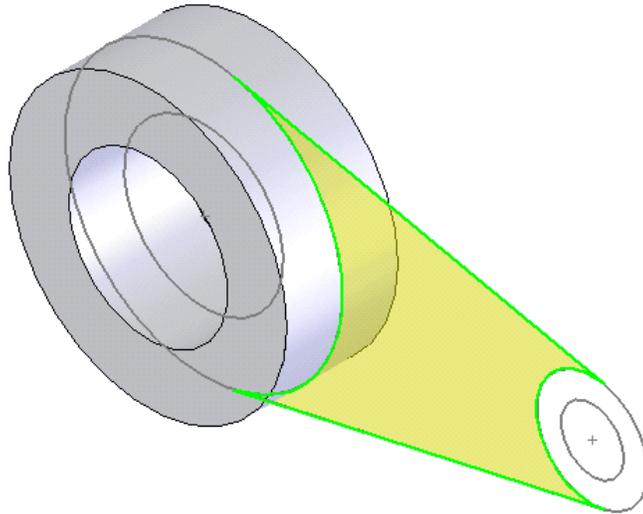


Figure 5-51 Contour selected for creating second feature.

menu. Choose the **Hide Sketch** option. The final model after creating all the features is shown in Figure 5-52.

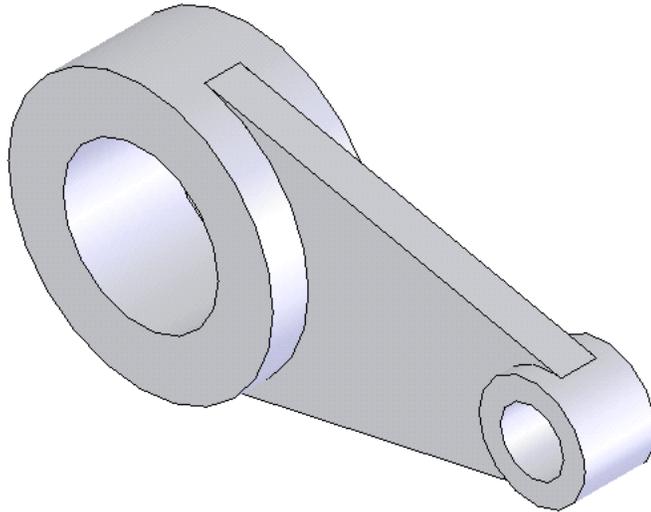


Figure 5-52 Final model.

If you click the + sign to expand the extruded feature in the **FeatureManager Design Tree** you will notice that instead of showing a sketch it will show you a contour selected sketch symbol as shown in Figure 5-53.

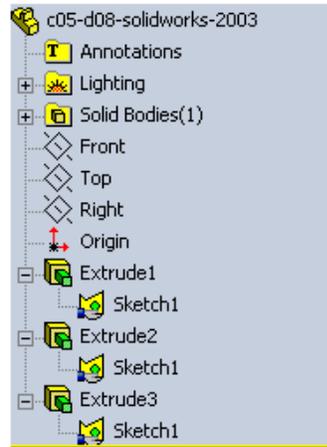


Figure 5-53 The FeatureManager Design Tree

CREATING CUT FEATURES

The cut is a material removal process. You can define a cut feature by extruding a sketch, revolving a sketch, sweeping a section along a path, lofting sections, or using a surface. You will learn more about sweep, loft, and surface in later chapters. The cut feature can be created only if a base feature exists. The cut operation using the extrude and revolve feature is discussed next.

Extruded Cut

Toolbar:	Features > Extruded Cut
Menu:	Insert > Cut > Extrude



To create an extruded cut feature, create a sketch for the cut feature and then choose the **Extruded Cut** button from the **Features** toolbar. You can also choose **Insert > Cut > Extrude** from the menu bar to invoke the **Cut-Extrude PropertyManager**. As soon as you invoke this **PropertyManager**, the preview of the cut feature with default options is displayed in the drawing area. The **Cut-Extrude PropertyManager** is shown in Figure 5-54.

Figure 5-55 shows the preview of the cut feature when you invoke the **Cut-Extrude PropertyManager** after creating a sketch for the cut feature. The material to be removed is

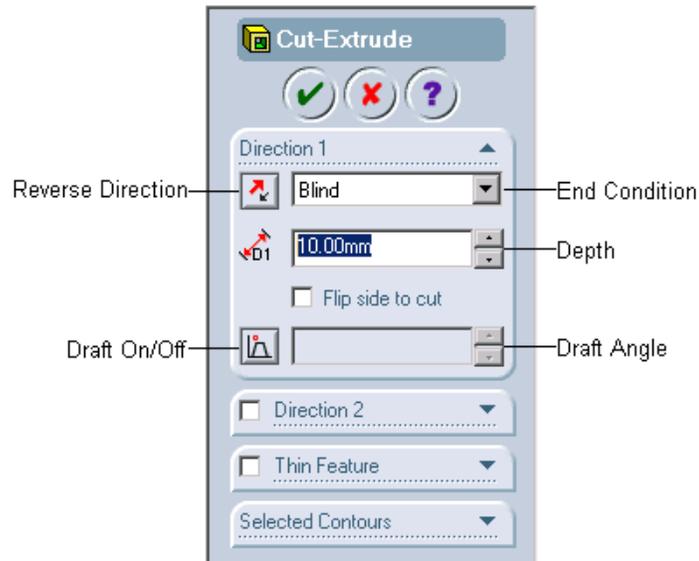


Figure 5-54 The Cut-Extrude PropertyManager

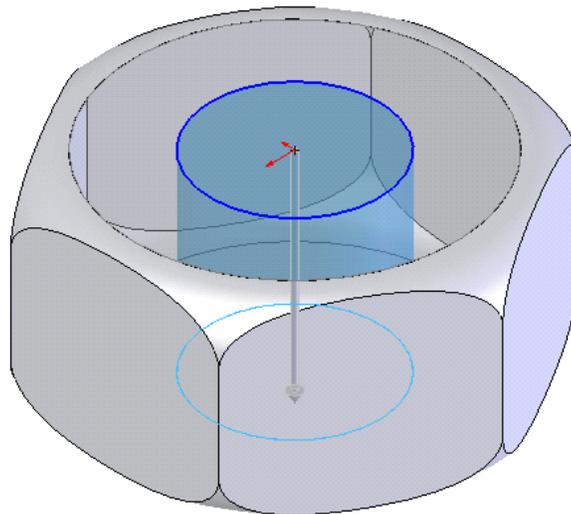


Figure 5-55 The preview of the cut feature

displayed in temporary graphics. Figure 5-56 shows the model after adding the cut feature. The various options available in the **Cut-Extrude PropertyManager** are discussed next.

Direction 1

The **Direction 1** rollout available in the **Extrude-Cut PropertyManager** is used to define the termination of the extrude in the first direction. The various options available in the **Direction 1** rollout are discussed next.

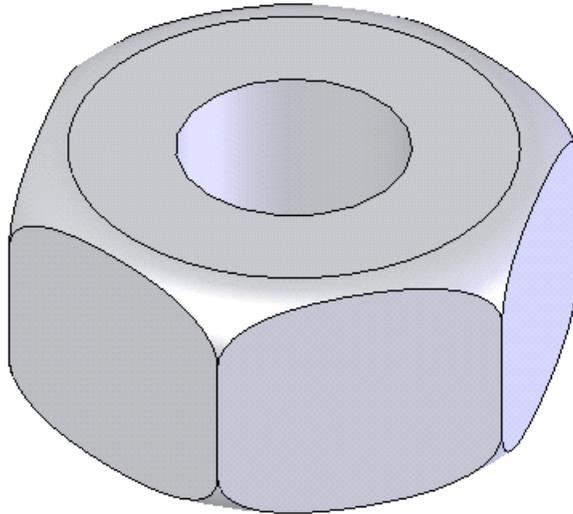


Figure 5-56 Cut feature added to the model

End Condition

The **End Condition** drop-down list available in the **Direction 1** rollout is used to specify the type of termination option available. The feature termination options available in this drop-down list are **Blind**, **Through All**, **Up To Next**, **Up To Vertex**, **Up To Surface**, and **Mid Plane**. These options are the same as those discussed for boss/base options. By default, the **Blind** option is selected in the **End Condition** drop-down list. Therefore, the **Distance** spinner is displayed to specify the depth. If you choose the **Through All** or the **Through Next** options, the spinner will not be displayed. The type of spinner or the display area depends on the option selected from the **End Condition** drop-down list. The **Reverse Direction** button is used to reverse the direction of feature creation. If you choose the **Mid Plane** option from the **End Condition** drop-down list, the **Reverse Direction** button is not available.

Flip Side to Cut

The **Flip side to cut** check box is selected to define the side of material removal. By default, the **Flip side to cut** check box is cleared. Therefore, the material will be removed from inside the profile of the sketch drawn for the cut feature. If you select this check box, the material will be removed from outside the profile of the sketch. Figure 5-57 shows a cut feature with **Flip side to cut** check box cleared and Figure 5-58 shows a cut feature with **Flip side to cut** check box selected.

Draft On/Off

The **Draft On/Off** button available in the **Direction 1** rollout of the **Cut-Extrude PropertyManager** is used to apply the draft angle to the extruded cut feature. The **Draft**



Tip. You can flip the direction of material removal by clicking the arrow available on the sketch while creating the cut feature.

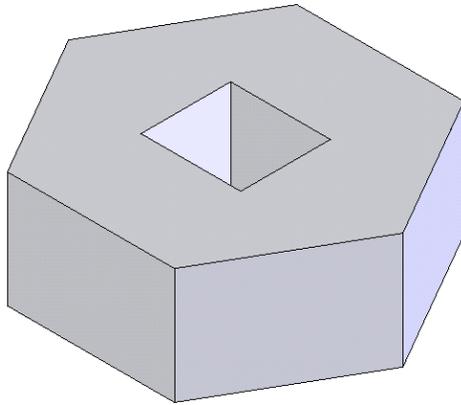


Figure 5-57 Cut feature with **Flip side to cut** check box cleared

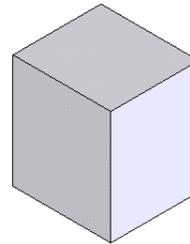


Figure 5-58 Cut feature with **Flip side to cut** check box selected

Angle spinner available on the right of the **Draft On/Off** button is used to set the value of the draft angle. By default, the **Draft outward** check box is cleared. Therefore, the draft is created inwards with respect to the direction of feature creation. If you select this check box, the draft added to the cut feature will be created outwards with respect to the direction of feature creation. Figure 5-59 shows the draft added to the cut feature with the **Draft outward** check box cleared and Figure 5-60 shows the draft added to the cut feature with the **Draft outward** check box selected.

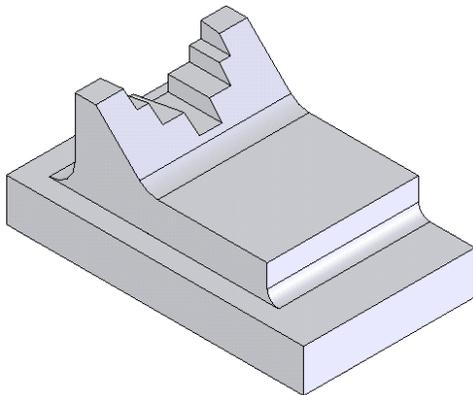


Figure 5-59 Cut feature with **Draft outward** check box cleared

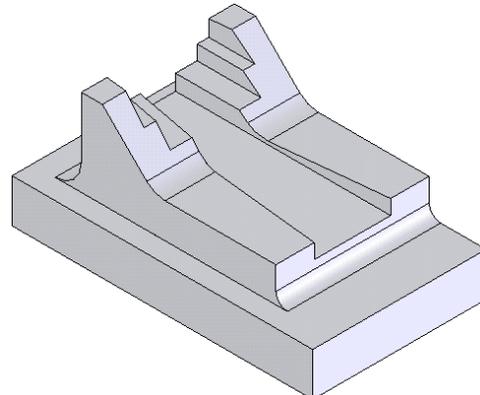


Figure 5-60 Cut feature with **Draft outward** check box selected

The **Direction 2** rollout is used to specify the termination of feature creation in the second direction. The options available in the **Direction 2** rollout are the same as those discussed for the **Direction 1** rollout. The **Selected Contour** rollout is used to add the feature to the selected contours.

Thin Feature

The **Thin Feature** rollout is used to create a thin cut feature. When you create a cut feature, you have to apply the thickness to the sketch in addition to the end condition. This rollout is used to specify the parameters to create the thin feature. To create a thin cut invoke the cut tool after creating the sketch and specify the end conditions in the **Direction 1** and **Direction 2** rollouts. Now, select the check box available in the **Thin Feature** rollout to activate the **Thin Feature** rollout. The **Thin Feature** rollout is shown in Figure 5-61. The options available in this rollout are the same as those discussed for the thin boss feature.



Figure 5-61 The Thin Feature rollout



Tip. The sketch used for the cut feature can be a closed loop or an open sketch. Note that if the sketch is an open sketch, the sketch should completely divide the model in two or more than two parts.

Handling Multiple Bodies in Cut Feature

While creating the cut feature, sometimes because of geometric conditions, feature termination, or end conditions the cut feature results in the creation of multiple bodies. Figure 5-62 shows a sketch created on the top planar surface of the base feature to create a cut feature. Figure 5-63 shows the cut feature created with the end condition as **Through All**. Using this type of sketch and end condition, if you choose **OK** from the **Extrude-Cut PropertyManager** the **Bodies to Keep** dialog is displayed. As multiple bodies are created while applying the cut feature, this dialog box is used to define which body do we want to keep.

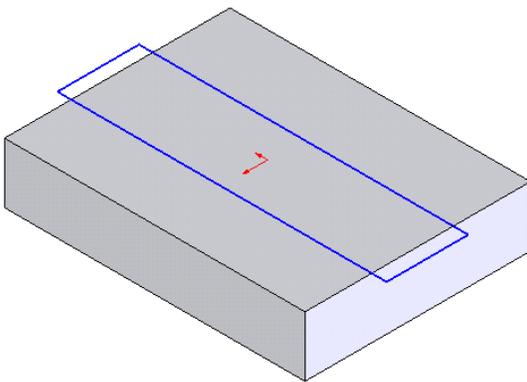


Figure 5-62 Sketch created for the cut feature.

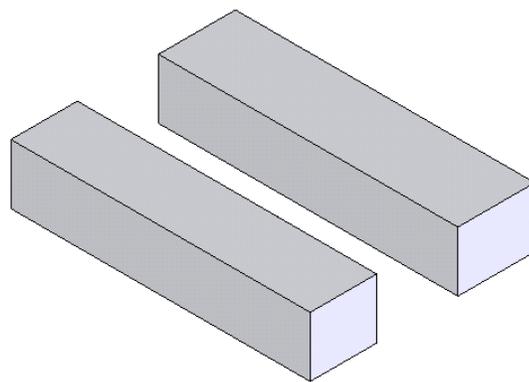


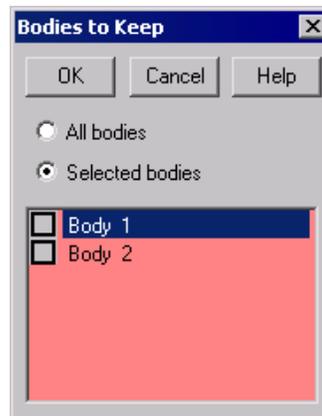
Figure 5-63 Cut feature is applied to the sketch with end condition as **Through All**

The **Bodies to Keep** dialog box is displayed in Figure 5-64. As the body is displayed in temporary graphics, when you invoke this dialog box, the edges of the model are displayed in yellow color.



*Figure 5-64 The **Bodies to Keep** dialog box*

By default, the **All Bodies** radio button is selected. Therefore, if you choose **OK** from this dialog box, all the bodies created after the cut feature will remain in the model. If you want the cut feature to consume any of the body, select the **Selected bodies** radio button from the **Bodies to Keep** dialog box. When you select the **Selected bodies** radio button, a selection and display area will be displayed as shown in Figure 5-65. You can select the check box provided on the left of the name of the body to keep that body. The selected body is displayed in green in temporary graphics. Select the bodies to keep and choose the **OK** button from the **Bodies to Keep** dialog box. Figure 5-66 shows a sketch created for the cut feature using the thin feature. Figure 5-67 shows the cut feature created using the thin option and the **All bodies** option selected from the **Bodies to Keep** dialog box.



*Figure 5-65 The **Bodies to Keep** dialog box with the **Selected bodies** option selected*

IMPORTANCE OF THE FEATURE SCOPE

As discussed earlier, in SolidWorks you can create different disjoint bodies in a single part file. After creating two or more than two disjoint bodies, when you create another feature, a **Feature Scope** rollout is displayed in the **PropertyManager**. This rollout is used to define the bodies that will be affected by the creation of the feature. The feature scope option is used with the following features:

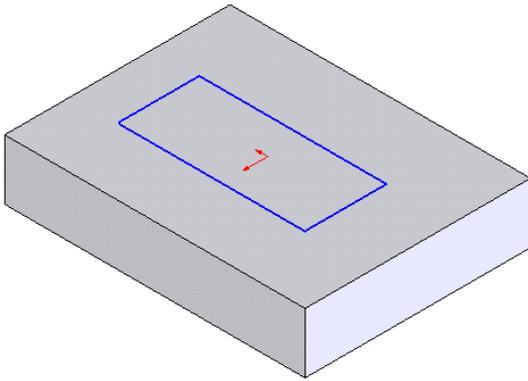


Figure 5-66 Sketch to create a cut feature using the *thin option*

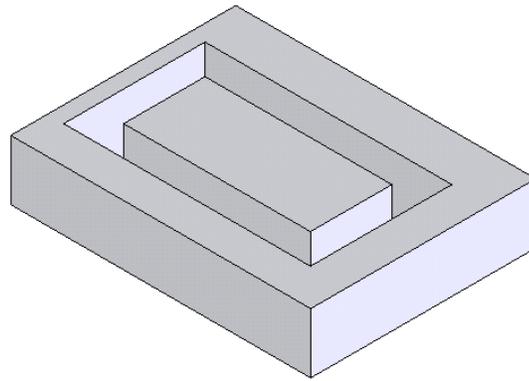


Figure 5-67 A thin cut feature created with the *All bodies* option selected

1. Extrude boss and cut
2. Revolve boss and cut
3. Sweep boss and cut
4. Loft boss and cut
5. Boss cut and thicken
6. Surface cut
7. Cavity

In the **Feature Scope** rollout, the **Selected bodies** radio button and the **Auto-select** check box are selected by default. With the **Auto-select** check box selected, all the disjoint bodies are selected and are affected by the feature creation. If you clear the **Auto-select** check box, a selection display area is invoked. You can select the bodies that you want to be affected. The name of the selected body is displayed in the display area. If you select the **All bodies** radio button then all the bodies available in the part file are selected and will be affected by the creation of the feature.

TUTORIALS

Tutorial 1

In this tutorial you will create the model shown in Figure 5-68. You will create the model by drawing the sketch of the front view of the model and then select the contours to extrude them. As a result, in this tutorial you will learn the procedure of modeling using the contours selection method. The dimensions of the model are shown in Figure 5-69.

(Expected time: 30 min)

It is clear from the figures that the given model is a multifeature model. It consists of various extrude features; therefore, all these features are sketched features. You first need to draw the sketch for each feature and then convert that sketch into the feature. In conventional methods you have to create a separate sketch for each sketched feature. But in this tutorial, you will use the contour selection technique. Using this method you have to create only one

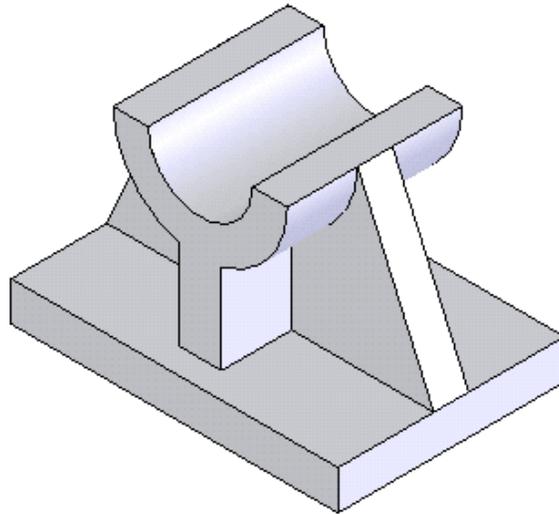


Figure 5-68 Solid model for Tutorial 1

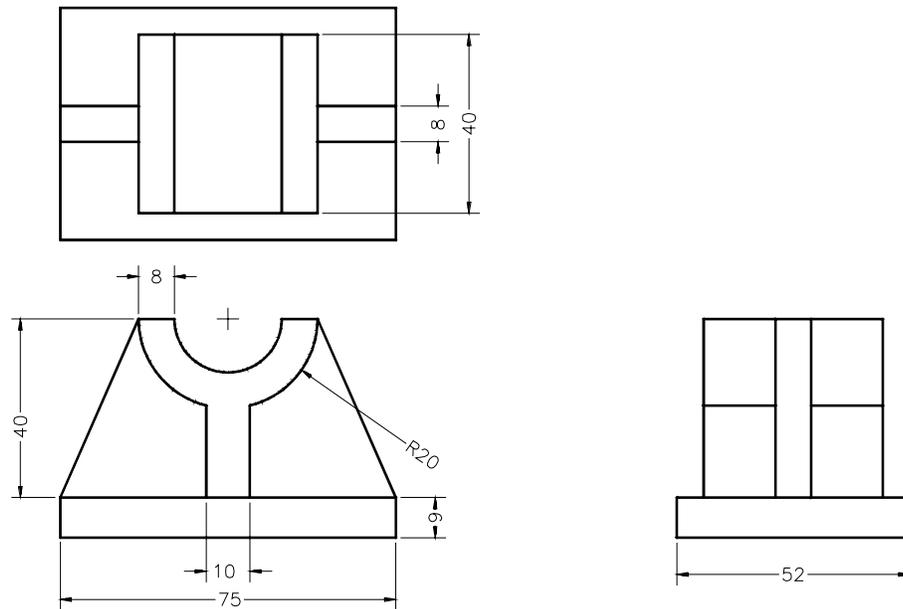


Figure 5-69 Dimensions and views for Tutorial 1

sketch and you will select the contours and share the same sketch for creating all the features. The steps to be followed to complete this tutorial are given next:

- a. Create the sketch on the default plane and apply the required relations and dimensions, refer to Figure 5-70.

- b. Invoke the extrude option and extrude the selected contour, refer to Figures 5-71 and 5-72.
- c. Select the second set of contour and extrude it to the required distance, refer to Figures 5-73 and 5-74.
- d. Select the third set of contour and extrude it to the required distance, refer to Figures 5-75 and 5-76.
- e. Save the file and then close the file.

Creating the Sketch of the Model

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

As discussed earlier, you need to choose the **Sketch** button from the **Sketch** toolbar after opening a new part file to start sketching. You will sketch on the front plane because the front plane is selected by default. In this tutorial you will have to create the sketch on the front plane.

2. Draw the sketch of the front view of the model using the automatic mirroring option to capture the design intent of the model.
3. Add the required relations and dimensions to the sketch as shown in Figure 5-70.

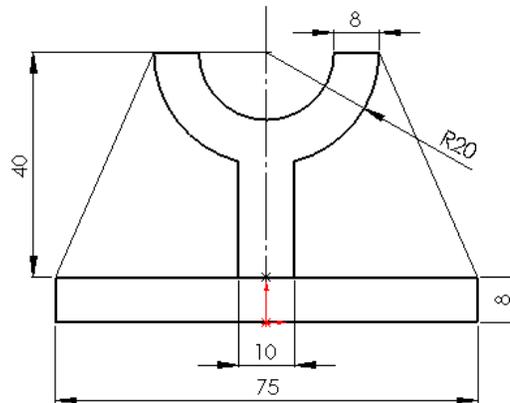


Figure 5-70 Fully defined sketch for creating the model

Selecting and Extruding the Contours of the Sketch

As discussed earlier, in this tutorial you will use the contour selection method to create the model. Therefore, first you need to select one of the contours from the given sketch and extrude it. For a better representation of the sketch, you will also orient the sketch to isometric view.

1. Choose the **Isometric** button from the **Standard Views** toolbar to orient the sketch in the isometric view. 
2. Right-click in the drawing area to invoke the shortcut menu. Now, choose the **Contour**

Select Tool option from the shortcut menu. The select cursor will be replaced by the contour selection cursor and selection confirmation corner is displayed.

3. Move the cursor to the lower rectangle of the sketch. When you move the cursor to the lower rectangle of the sketch, the area of rectangle will be highlighted in pink. This indicates that this rectangle is a closed profile.
4. Now, select the lower rectangle. The selected area is displayed in yellow. Right-click or choose **OK** from the contour selection confirmation corner. You can also right-click immediately after selecting the contour to confirm the selection and choose the **End Select Contours** option from the shortcut menu. Figure 5-71 shows the lower rectangle selected using the contour selection tool.

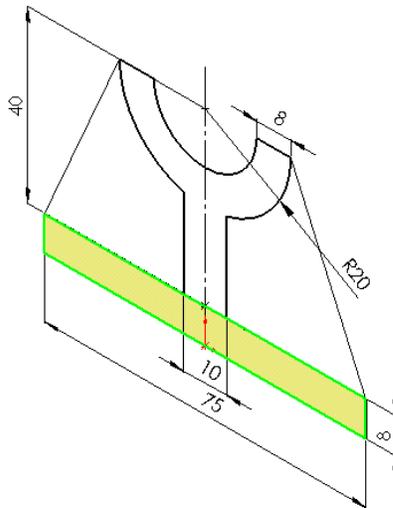


Figure 5-71 Lower rectangle selected as a contour

5. Choose the **Extruded Boss/Base** button from the **Features** toolbar. The **Extrude PropertyManager** is invoked and the preview of the base feature is displayed in the drawing area in temporary graphics. 

The name of the selected contour is displayed in the display area of the **Selected Contours** rollout.

6. Right-click in the drawing area and choose the **Mid Plane** option from the shortcut menu. The preview of the feature is modified dynamically when you choose the **Mid Plane** option.
7. Set the value of the **Depth** spinner to **52** and choose **OK** button from the **Extrude PropertyManager**.

The base feature of the model by extruding the selected contour is shown in Figure 5-72.

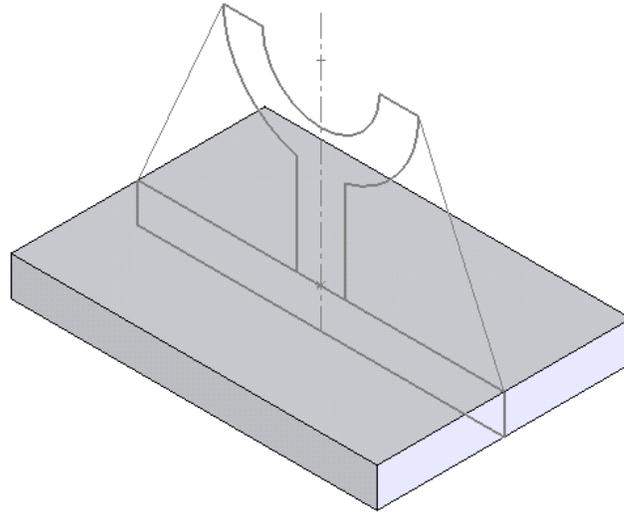


Figure 5-72 Base feature of the model

8. Right-click in the drawing area and choose the **Contour Select Tool** option from the shortcut menu.

The select cursor is replaced by the contour selection cursor.

9. Using the contour selection cursor select any entity of the sketch to invoke the selection mode of the sketch.
10. Using the left mouse button select the middle contour of the sketch. The selected region will be displayed yellow in color. Right-click in the drawing area to exit the contour selection process.

The selected middle contour is shown in Figure 5-73.

11. Choose the **Extrude** button from the **Features** toolbar to invoke the **Extrude PropertyManager**. 
12. Right-click in the drawing area and choose the **Mid Plane** option from the shortcut menu.
13. Set the value of the **Depth** spinner to **40** and choose the **OK** button from the **Extrude PropertyManager**.

The feature created by selecting the middle contour is shown in Figure 5-74.

14. Now, again invoke the **Contour Select Tool** and select a sketch entity. Next, select the right contour of the sketch. Press and hold down the CTRL key and select the left contour of the sketch.

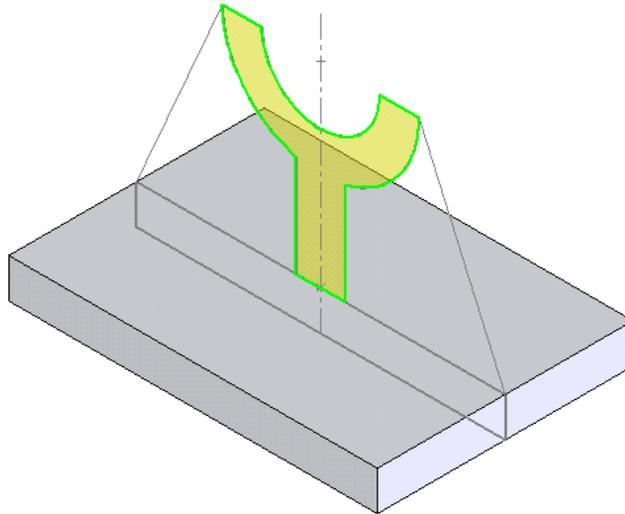


Figure 5-73 Middle contour is selected using the contour selection tool

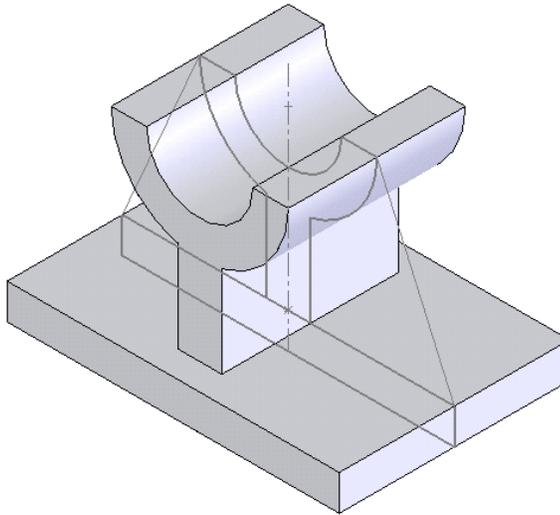


Figure 5-74 Second feature created by extruding the middle contour

The two selected contours are shown in Figure 5-75.

15. Invoke the **Extrude PropertyManager**. Right-click and choose the **Mid Plane** option from the shortcut menu.
16. Set the value of the spinner to **8** and choose the **OK** button from the **Extrude PropertyManager**.

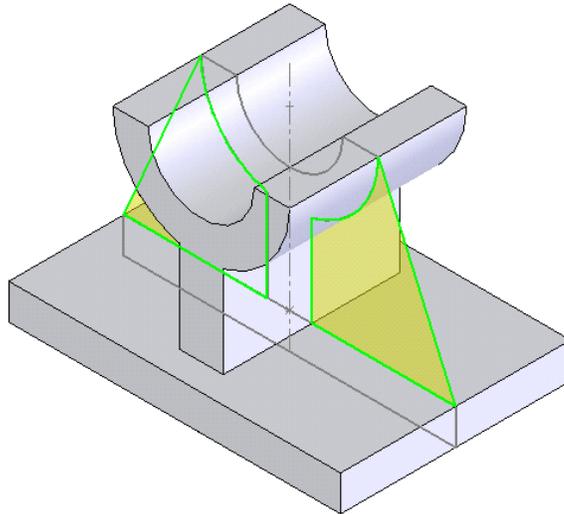


Figure 5-75 The right and the left contours selected using the contour selection tool

The model is completed. But, the sketch is also displayed in the model. Therefore, you need to hide the sketch.

17. Move the cursor to any of the sketched entity and when the entity turns red in color, select the entity. The selected sketched entity will be displayed in green. Now, right-click and choose the **Hide Sketch** option from the shortcut menu.

The final model with hidden sketch is shown in Figure 5-76. The **FeatureManager Design Tree** of the model is shown in Figure 5-77.

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:



\My Documents\SolidWorks\c05\c05-tut01.SLDPRT.

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

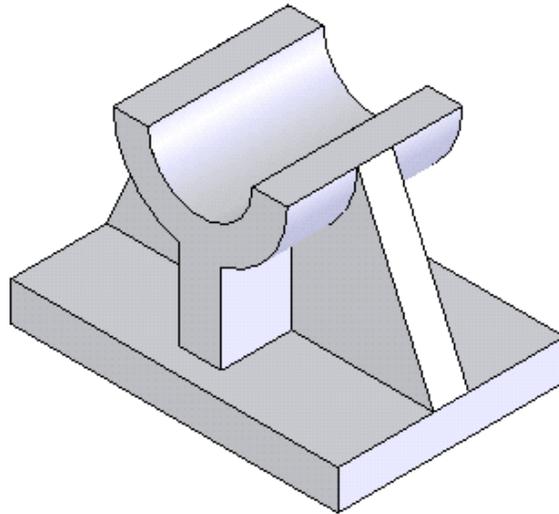


Figure 5-76 Final solid model



Figure 5-77 The FeatureManager Design Tree

Tutorial 2

In this tutorial you will create a model shown in Figure 5-78. You will use a combination of the conventional model method and the contour selection modeling methods to create this model. The dimensions of the model are given in Figure 5-79. **(Expected time: 30 min.)**

The steps that will be used to complete the model are discussed next:

- Create the sketch of the front view of the model, refer to Figure 5-80.
- Extrude the selected contours, refer to Figures 5-81 through 5-82.
- Add a cut feature to the model by creating the sketch on the left planar surface, refer to Figures 5-84 and 5-85.
- Create four holes using the cut feature on the top face of the base feature, refer to Figures 5-86 and 5-87.

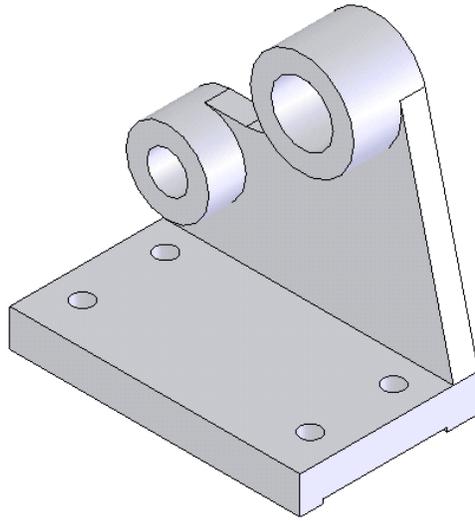


Figure 5-78 Solid model for Tutorial 2

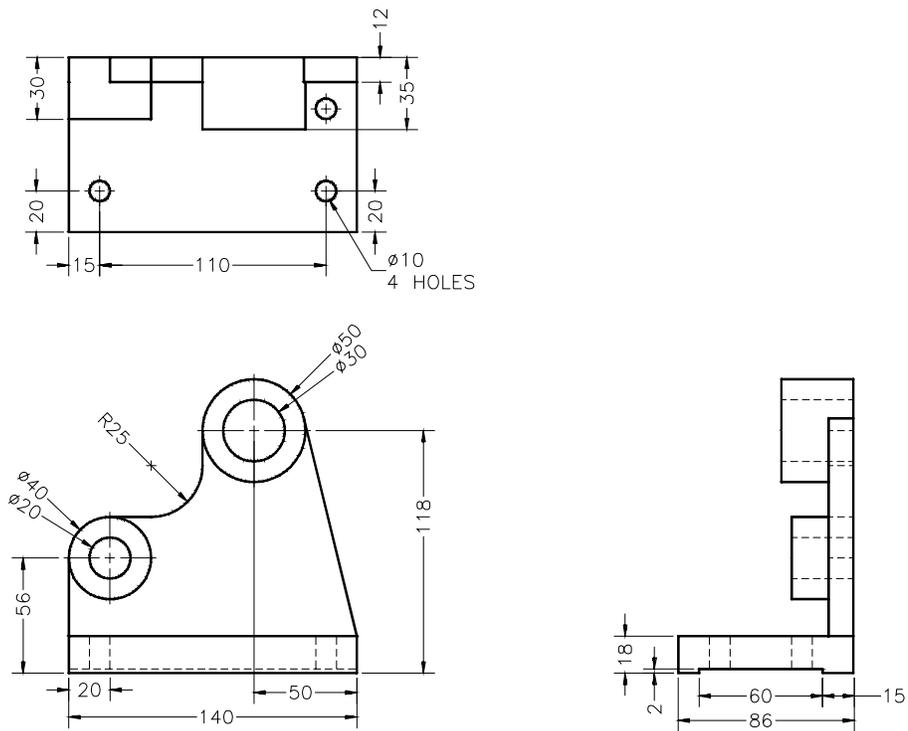


Figure 5-79 Drawing views of the solid model

Creating the Sketch for Contour Selection Modeling

You need to create the sketch by referring the front view of the model. The contours will be selected from this sketch and the selected contour will be extruded.

1. Draw the sketch of the front view using the standard sketching tools.
2. Apply the required relations and dimensions to fully define the sketch. The fully defined sketch is shown in Figure 5-80.

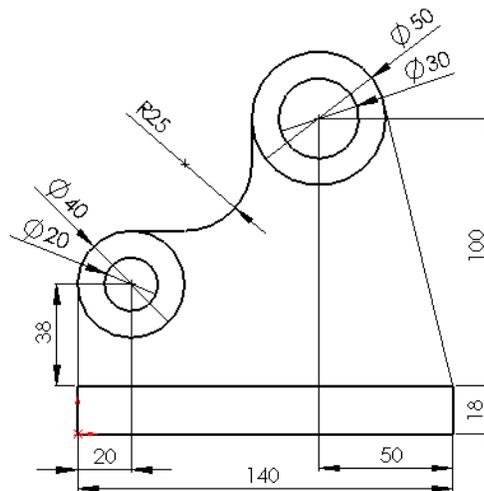


Figure 5-80 Fully defined sketch

Orient the view to isometric view because it will help you in the selection of contours.

3. Choose the **Isometric** button from the **Standard Views** toolbar to orient the view to isometric view. 
4. Right-click and choose the **Contour Select Tool** option from the shortcut menu. The select cursor is replaced by the contour selection cursor.
5. Using the contour selection cursor select the area enclosed by the lower rectangle as shown in Figure 5-81.
6. Choose the **Extruded Boss/Base** button from the **Features** toolbar to invoke the **Extrude PropertyManager**. 
7. Set the value of the **Depth** spinner to **86** and choose the **OK** button from the **Extrude PropertyManager**.

The base feature created after extruding the selected contour is shown in Figure 5-82.

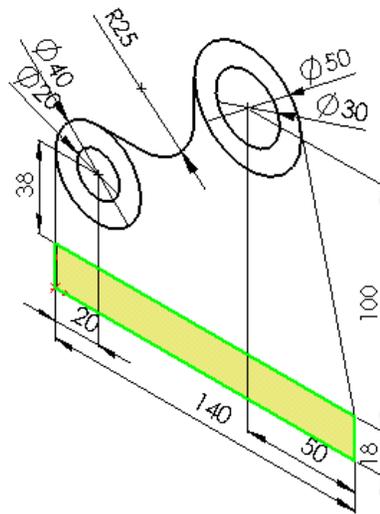


Figure 5-81 Lower rectangle selected as contour

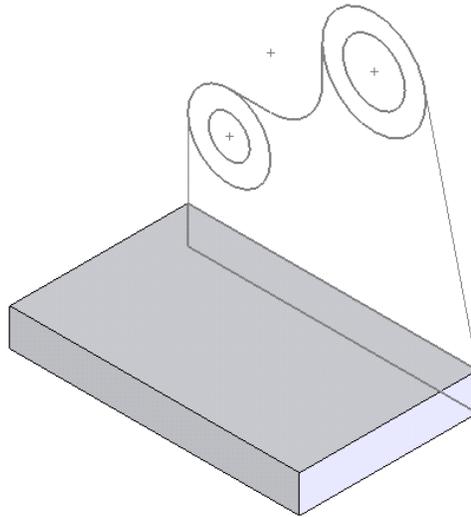


Figure 5-82 Base feature created after extruding the selected contour

8. Using the contour selection tool and the extrude tool create the other features by extruding the selected contours and then hide the sketch. The model created after extruding all the contours is shown in Figure 5-83.

Creating the Recess at the Base of the Model

After creating the extruded features of the model, you have to create the recess provided at the base of the model. The recess will be created by extruding a sketch using the cut option created at the right planar face of the model.

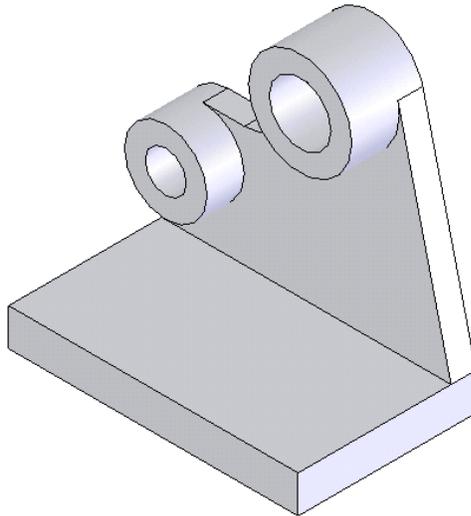


Figure 5-83 Model created after extruding the selected contour

1. Select the right planar face of the base feature as the sketching plane. The selected face will be displayed in green color.
2. Right-click in the drawing area and choose the **Insert Sketch** option from the shortcut menu to invoke the sketching environment.

Now you need to orient the view such the selected face is normal to your eye view.

3. Choose the **Normal To** button from the **Standard Views** toolbar to orient the selected face normal to the eye view. 
4. Using the standard sketching tools create the sketch for the recess and apply the required relations and dimensions to the sketch. The fully defined sketch for the cut feature is shown in Figure 5-84.
5. Choose the **Extruded Cut** button from the **Features** toolbar to invoke the **Cut-Extrude PropertyManager**. 

The preview of the cut feature will be displayed in the drawing area in temporary graphics.

6. Right-click in the drawing area and choose the **Through All** option from the shortcut menu.
7. Choose the **OK** button from the **Cut-Extrude PropertyManager** to complete the feature creation.

The model after creating the cut feature is shown in Figure 5-85.

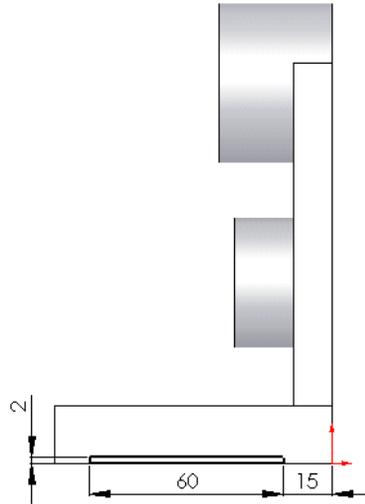


Figure 5-84 Sketch for the cut feature

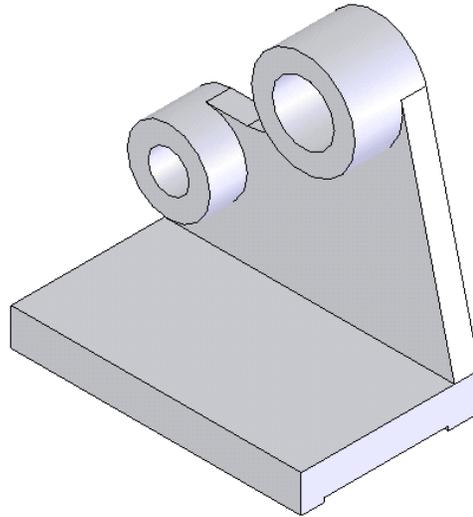


Figure 5-85 Cut feature added to the model

Creating the Holes

After creating the recess provided at the bottom of the base, you need to create the holes on the base of the model. You will create the sketch of the hole feature on the top planar face of the base feature of the model. For creating the sketch of the holes you will first draw a circle and then using the **Linear Sketch Step and Repeat** option create the remaining circles. After that use the cut option to complete the hole feature.

1. Select the top planar face of the base feature as the sketching plane. Right-click and choose the **Insert Sketch** option from the shortcut menu.

2. Click anywhere in the drawing area to exit the selection mode.
3. Choose the **Normal To** button from the **Standard Views** toolbar to orient the view normal to the eye view. 
4. Using the standard sketching tools create a circle and then using the **Linear Sketch Step and Repeat** create a pattern of the remaining circles.
5. Apply the required relations and dimensions to fully define the sketch. You may need to apply horizontal relation between the centers of the top two circles to fully define it.

The fully defined sketch is shown in Figure 5-86.

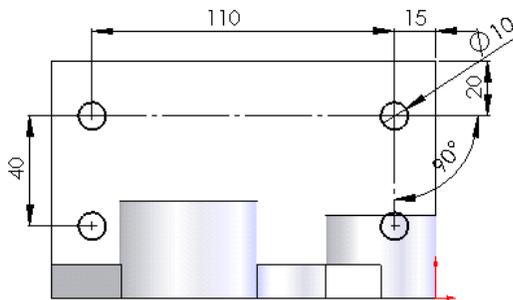


Figure 5-86 Holes sketched for the cut feature

6. Choose the **Extruded Cut** button from the **Features** toolbar to invoke the **Cut-Extrude PropertyManager**. 
7. Right-click in the drawing area and choose the **Through All** option from the shortcut menu.
8. Choose the **OK** button from the **Cut-Extrude PropertyManager**.

The final model is shown in Figure 5-87 and the **FeatureManager Design Tree** of the model is shown in Figure 5-88.

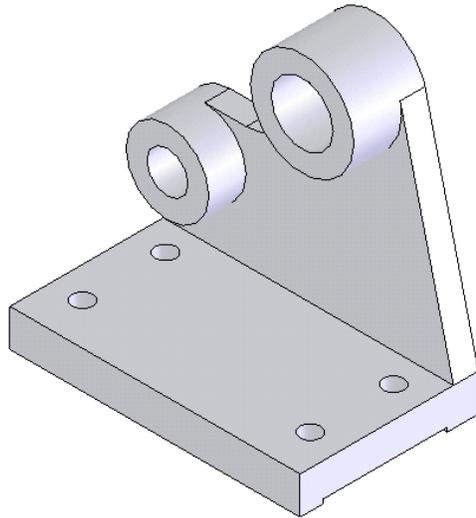


Figure 5-87 Final solid model

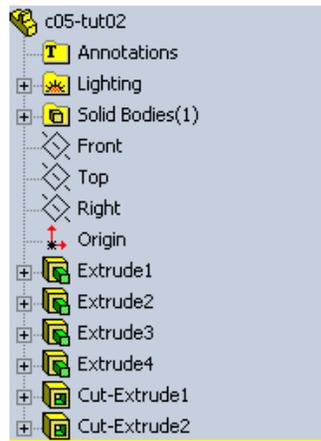


Figure 5-88 The FeatureManager Design

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:



My Documents\SolidWorks\c05\c05-tut02.SLDPRT.

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

Tutorial 3

In this tutorial you will create the model shown in Figure 5-89. The dimensions of the model are given in Figure 5-90. **(Expected Time: 30 min)**

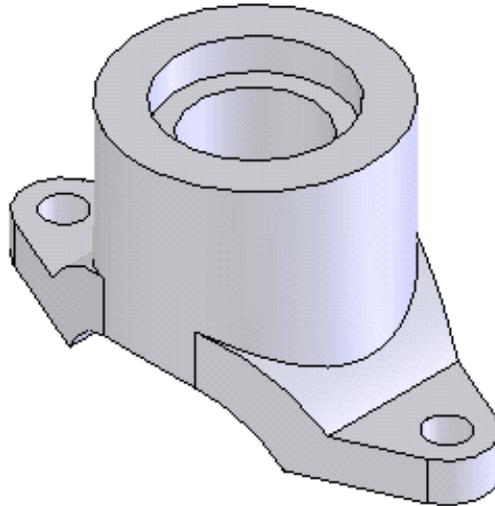


Figure 5-89 Solid model for Tutorial 3

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

- Create the base feature by extruding the sketch created on the front plane, refer to Figures 5-91 and 5-92.
- Extrude the sketch created on the top plane to create a cut feature, refer to Figures 5-93 and 5-94.
- Create a plane at an offset distance of 150 from the top plane, refer to Figure 5-95.
- Create the sketch on the newly created plane and extrude it to a selected surface, refer to Figures 5-96 and 5-97.
- Create a contour bore using the cut revolve option, refer Figures 5-98 and 5-99.
- Create the holes using the cut feature, refer Figure 5-100 and 5-101.

Creating the Base Feature

It is evident from the model that the base of the model comprises complex geometry. Therefore, you need to create the base feature and then apply the cut feature to the base of the model to get the desired shape. You will create the base feature on the **Front** plane as the sketching plane. After creating the sketch, you will extrude the sketch using the mid plane option to complete feature creation.

- Open a new SolidWorks document in the part mode and invoke the sketching environment to draw the sketch for the base feature.
- Using the standard sketching tools create the sketch of the base feature and then apply the required relations and dimensions to the sketch as shown in Figure 5-91.

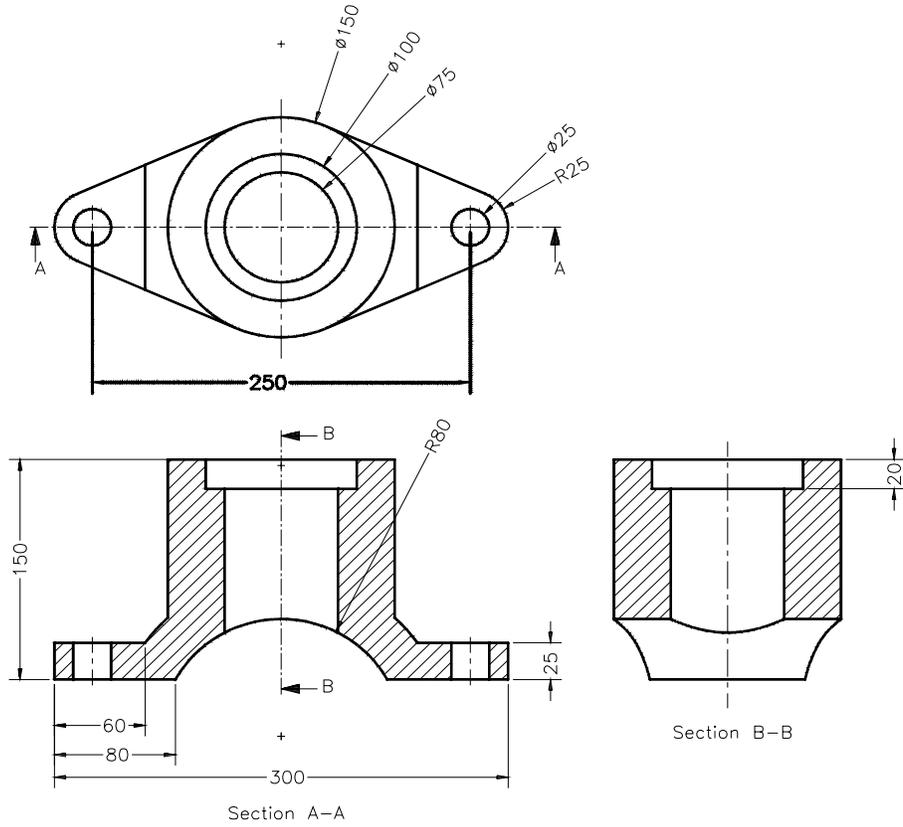


Figure 5-90 Top view, front section view, and right section view of the model

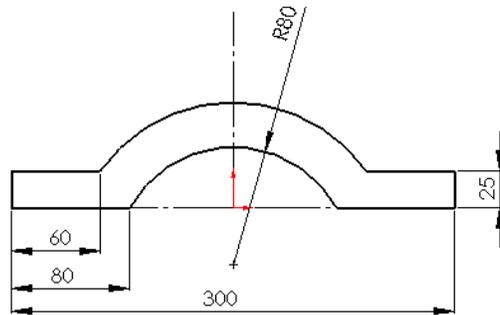


Figure 5-91 Sketch of the base feature

3. Choose the **Extruded Boss/Base** button from the **Features** toolbar to invoke the **Extrude PropertyManager**. 
4. Right-click in the drawing area and choose the **Mid Plane** option from the shortcut menu.
5. Set the value of the **Depth** spinner to **150** and choose the **OK** button from the **Extrude PropertyManager**.

The isometric view of the base feature of the model is shown in Figure 5-92.

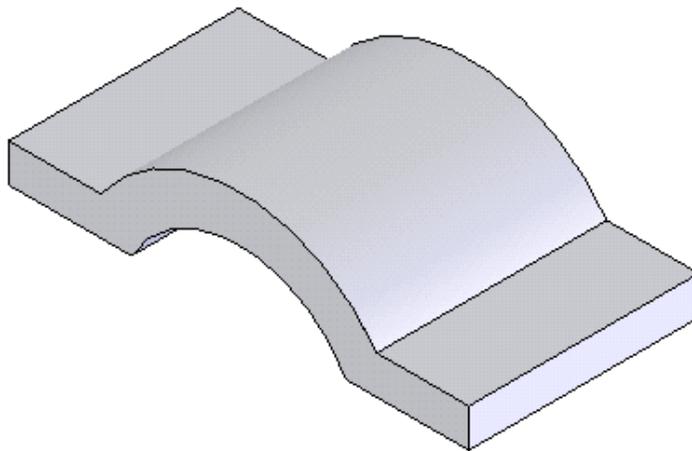


Figure 5-92 Base feature of the solid model

Creating the Cut Feature

Now, you need to create a sketch on the top plane and then you will create a cut feature using that sketch. After creating this feature you will be able to get the base of the model.

1. Select the **Top** plane from the **FeatureManager Design Tree** and invoke the sketching environment.
2. Create the sketch for the cut feature using the standard sketching tools and then apply the required relations and dimensions to the sketch as shown in Figure 5-93.



Note

To change the radial dimension to the diametrical dimension, select the dimension and invoke the shortcut menu. Choose the **Properties** option from the shortcut menu to display the **Dimension Properties** dialog box. Select the **Diameter dimension** check box from this dialog box. Choose the **OK** button from the **Dimension Properties** dialog box.

To change the diameter dimension to the radial dimension, invoke the **Dimension Properties** dialog box and clear the **Diameter dimension** check box and choose the **OK** button from the dialog box.

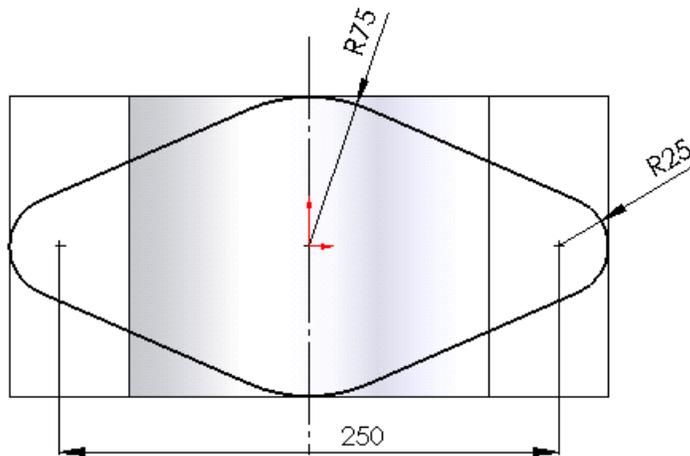


Figure 5-93 Sketch for the cut feature

3. Choose the **Extruded Cut** button from the **Features** toolbar to invoke the **Cut-Extrude PropertyManager**. 

Since the preview of the cut feature is not displayed in the current view, therefore, you need to orient the model in the isometric view to preview the cut feature.

4. Choose the **Isometric** button from the **View** toolbar to orient the model in the isometric view. 

It is evident from the preview of the cut feature that the direction of feature creation is opposite to the required direction. Therefore, you need to change the direction of feature creation.

5. Choose the **Reverse Direction** button provided at the left of the **End Condition** drop-down list in the **Cut-Extrude PropertyManager**.

You will also observe that direction of material removal is not in the required direction. Therefore, you will have to flip the direction of material removal.

6. Select the **Flip side to cut** check box. You will observe that the direction of material removal is also changed in the preview.
7. Right-click in the drawing area and choose the **Through All** option from the shortcut menu and choose the **OK** button from the **Cut-Extrude PropertyManager**.

The model after creating the base feature is shown in Figure 5-94.

Creating a Plane at an Offset Distance

After creating the base of the model you will create a plane at an offset distance from the top plane. This newly created plane will be used as a sketching plane for the next feature.

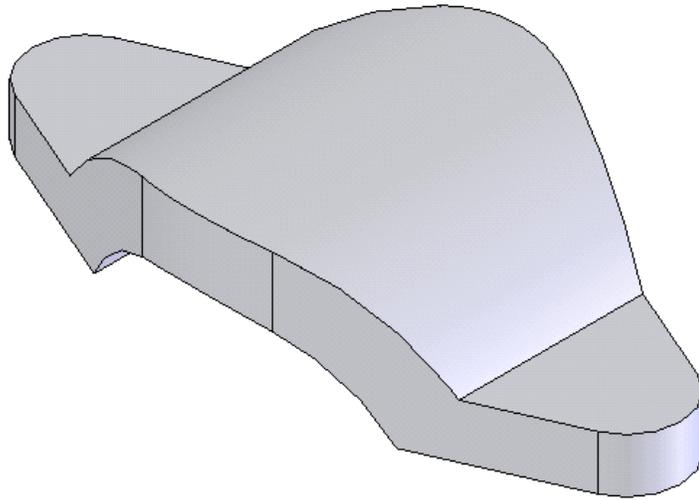


Figure 5-94 Cut feature added to the base feature

1. Choose the **Plane** button from the **Reference Geometry** toolbar to invoke the **Plane PropertyManager**. The **Plane PropertyManager** is displayed on the left of the drawing area. 
2. Choose the **Offset Distance** button from the **Plane PropertyManager**. The **Distance** spinner, **Reverse Direction** check box, and **Number of Planes to Create** spinner are displayed in the **Plane PropertyManager**.

Since the planes are not displayed in the drawing area, therefore, you will invoke the **FeatureManager Design Tree** flyout to select the **Top** plane from the same.

3. Move the cursor to the top of the **Plane PropertyManager** where the name of the **PropertyManager** is displayed. The tool tip will show you the name of that area as **Show FeatureManager**.
4. Use the left mouse button at that location to invoke the **FeatureManager Design Tree** flyout.
5. Select the **Top** plane from the **FeatureManager Design Tree**.

When you choose the **Top** plane, the preview of the newly created plane at an offset distance using a default value is displayed in the drawing area.

6. Set the value of the **Distance** spinner to **150** and choose the **OK** button from the **Plane PropertyManager**.

The newly created plane is shown in Figure 5-95.

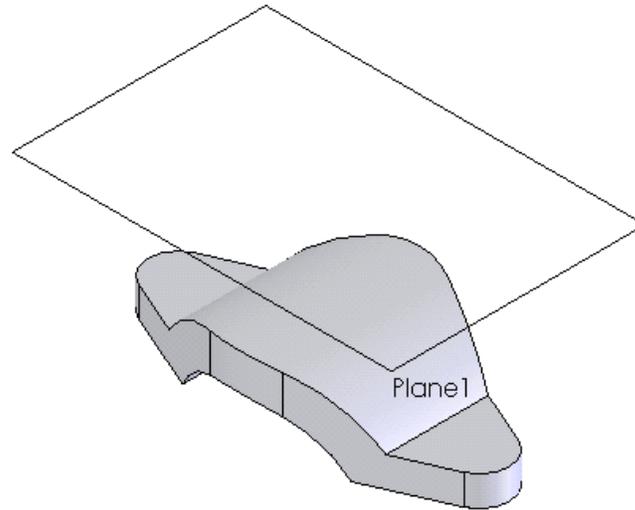


Figure 5-95 Plane created at an offset distance from the top plane

Creating the Extruded Feature

After creating the plane at an offset distance from the top plane, you will create the next feature, which is an extruded feature. The sketch of the next feature is created on the newly created plane and the sketch is extruded up to a selected surface.

Since the newly created plane is selected, therefore, you do not need to select the plane.

1. Invoke the sketching environment and click anywhere in the drawing area to remove the plane from the selection set.
2. Choose the **Normal To** button from the **Standard Views** toolbar to orient the selected plane normal to the eye view. 

The model will be oriented such that the sketch plane is oriented normal to the eye view.

4. Draw the sketch of the circle and apply the required relations to the sketch as shown in Figure 5-96.
5. Choose the **Extruded Boss/Base** button from the **Features** toolbar. The preview of the feature is displayed in the drawing area. 

You need to orient the model in the isometric view because the preview of feature creation is not clear in the current view.

6. Orient the model in the isometric view.

You will observe from the preview that the direction of feature creation is opposite to the required direction. Therefore, you need to change the direction of feature creation.

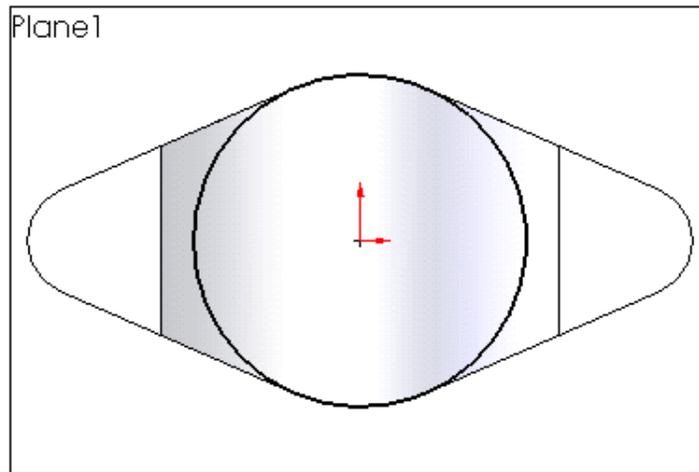


Figure 5-96 Sketch created on the newly created plane

7. Choose the **Reverse Direction** button provided at the left of the **End Condition** drop-down list to reverse the direction of feature creation.

You will observe that the preview of the feature is also changed when you reverse the direction of feature creation.

8. Right-click in the drawing area and choose the **Up To Surface** button from the shortcut menu. You are prompted to select a face or a surface to complete the specification of end condition in **Direction 1**.

The **Face/Plane** display area is displayed under the **End Condition** drop-down list.

9. Select the upper curved surface of the model using the left mouse button. You will observe that the preview shows the feature extruded upto the selected surface.
10. Choose the **OK** button from the **Extrude PropertyManager** or right-click to choose the **OK** option.

Since the plane is displayed in the drawing area, you need to turn its display off.

11. Select **Plane 1** from the **FeatureManager Design Tree** or from the drawing area and invoke the shortcut menu. Choose the **Hide** option from the shortcut menu.

The model created after creating the extruded feature is shown in Figure 5-97.

Creating the Counterbore Hole

Next, you have to create the counterbore hole by revolving a sketch created on the front plane using the cut option.

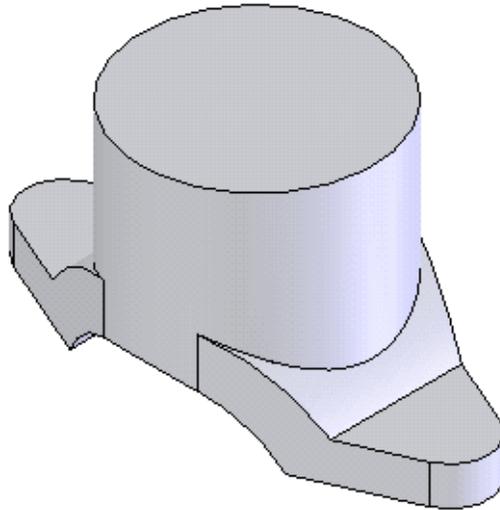


Figure 5-97 Sketch extruded upto a selected surface

1. Invoke the sketching environment by selecting the **Front** plane as the sketching plane and orient the front plane normal to the eye view.
2. Create the sketch of the counterbore hole using the standard sketching tools. As discussed earlier the sketch will be revolved along a centerline using the cut option. Therefore, after creating the sketch and applying the required relations you will dimension the sketch using the diametrical dimensioning along the centerline.

The sketch after applying the required relations and dimensions is shown in Figure 5-98.

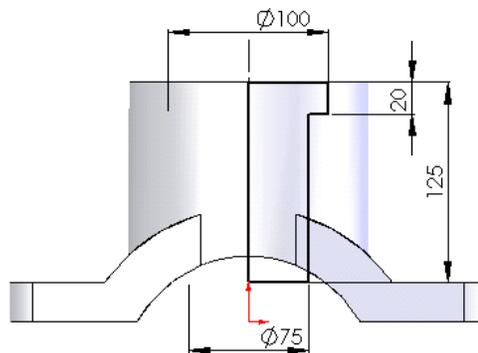


Figure 5-98 Fully defined sketch for the contour bore

3. Choose the **Revolved Cut** button from the **Features** toolbar to display the **Cut-Revolve PropertyManager**. The preview of the cut feature is displayed in the drawing area in temporary graphics. The value of the angle in the **Angle** spinner is set to **360** by default. Therefore, you do not need to set the value of the **Angle** spinner. 
4. Choose the **OK** button from the **Cut-Revolve PropertyManager**.

Figure 5-99 shows the model after creating the revolve cut feature.

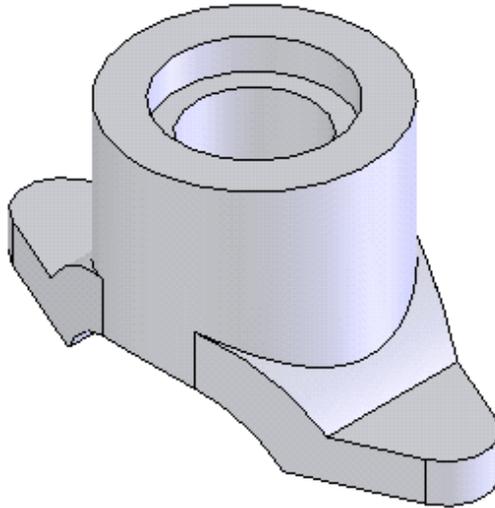


Figure 5-99 Cut feature added to the model

Creating the Holes

After creating all the features, you will create the holes using the cut feature to complete the model. The sketch for the cut feature will be created using the top planar surface of the base feature as the sketching plane.

1. Select the top planar surface of the base feature and invoke the sketching environment. Orient the model so that the selected face of the model is oriented normal to the eye view.
2. Create the sketch using the standard sketching tools and apply the required relations and dimensions as shown in Figure 5-100.
3. Choose the **Extruded Cut** button from the **Features** toolbar to invoke the **Cut-Extrude PropertyManager**. 
4. Right-click and choose the **Through All** option from the shortcut menu and choose the **OK** button from the **Cut-Extrude PropertyManager** or invoke the shortcut menu and choose the **Through All** option from the shortcut menu.

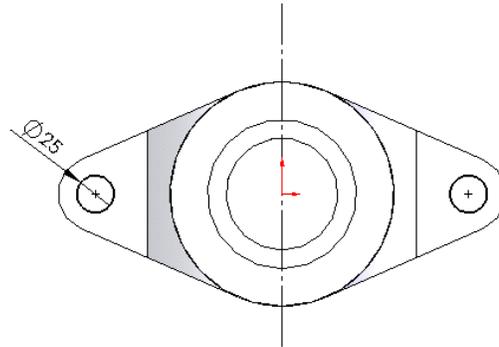


Figure 5-100 Fully defined sketch for cut feature

Orient the model in the isometric view. The final model is shown in Figure 5-101. The **FeatureManager Design Tree** of the model is shown in Figure 5-102.

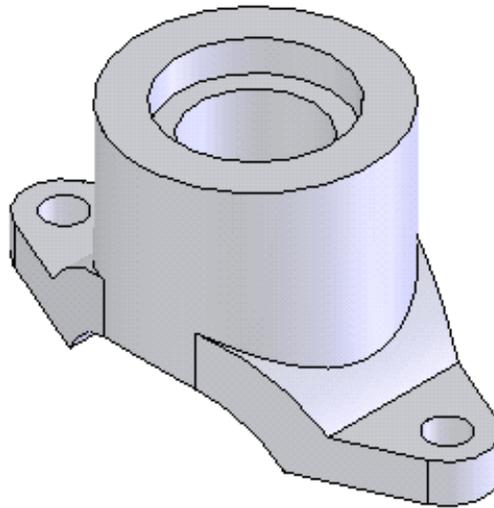


Figure 5-101 Final model

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:





Figure 5-102 FeatureManager Design Tree of the model

\\My Documents\SolidWorks\c05\c05-tut03.SLDPRT.

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

SELF-EVALUATION TEST

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

1. When you create a sketch for the first time in the sketching environment, the sketch is created on the default plane, which is **Front** plane. (T/F)
2. When you create a new SolidWorks part document, SolidWorks provides you with two default planes. (T/F)
3. You can choose the **Plane** button from the **Features** toolbar to invoke the **Plane PropertyManager**. (T/F)
4. You cannot create a plane at an offset distance by dynamically dragging. (T/F)
5. When you create a circular feature a temporary axis is automatically created. (T/F)
6. The _____ option is used to extrude a sketch from the sketching plane to the next surface that intersects the feature.
7. The _____ option available in the **End Condition** drop-down list is used to define the termination of the extruded feature upto another body.
8. The _____ check box is used to merge the newly created body with the parent body.

9. Using the _____ option you can create a reference axis that passes through the center point of a cylindrical or a conical surface.
10. Sometimes multiple bodies are created while applying the cut feature; therefore, the _____ dialog box is displayed that allows you to define which body do we want to keep.

REVIEW QUESTIONS

Answer the following questions:

1. If the _____ check box is cleared, then the virtual surface created for the termination of the extruded feature will have a concentric relation with the selected surface.
2. The _____ option is selected from the shortcut menu to select the contours.
3. The _____ option is available in the **End condition** drop-down list only after you create the base feature.
4. The _____ check box is used to define the side of material removal.
5. The _____ check box is used to create an outward draft in a cut feature.
6. Which check box is selected while creating any other feature in a single body modeling?
 - (a) **Combine results**
 - (b) **Fix bodies**
 - (c) **Merge results**
 - (d) **Union results**
7. Which button is used to add a draft angle to a cut feature?
 - (a) **Add Draft**
 - (b) **Create Draft**
 - (c) **Draft On/Off**
 - (c) None of these
8. Which **PropertyManager** is invoked to create a cut feature by extruding a sketch?
 - (a) **Extruded Cut**
 - (b) **Cut-Extrude**
 - (c) **Extrude-Cut**
 - (d) **Cut**
9. The option used to define the termination of feature creation at an offset distance to a selected surface is
 - (a) **Distance To Surface**
 - (b) **Normal From Surface**
 - (c) **Distance From Surface**
 - (d) **Offset From Surface**
10. The option used to define the termination of feature creation to the selected surface is
 - (a) **To Surface**
 - (b) **Selected Surface**
 - (c) **Up To Surface**
 - (d) None of these

EXERCISES

Exercise 1

Create the model shown in Figure 5-103. The dimensions of the model are given in Figure 5-104. **(Expected time: 30 min)**

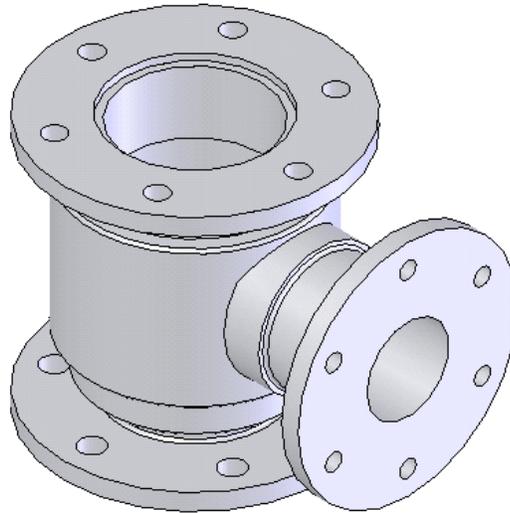


Figure 5-103 Model for Exercise 1

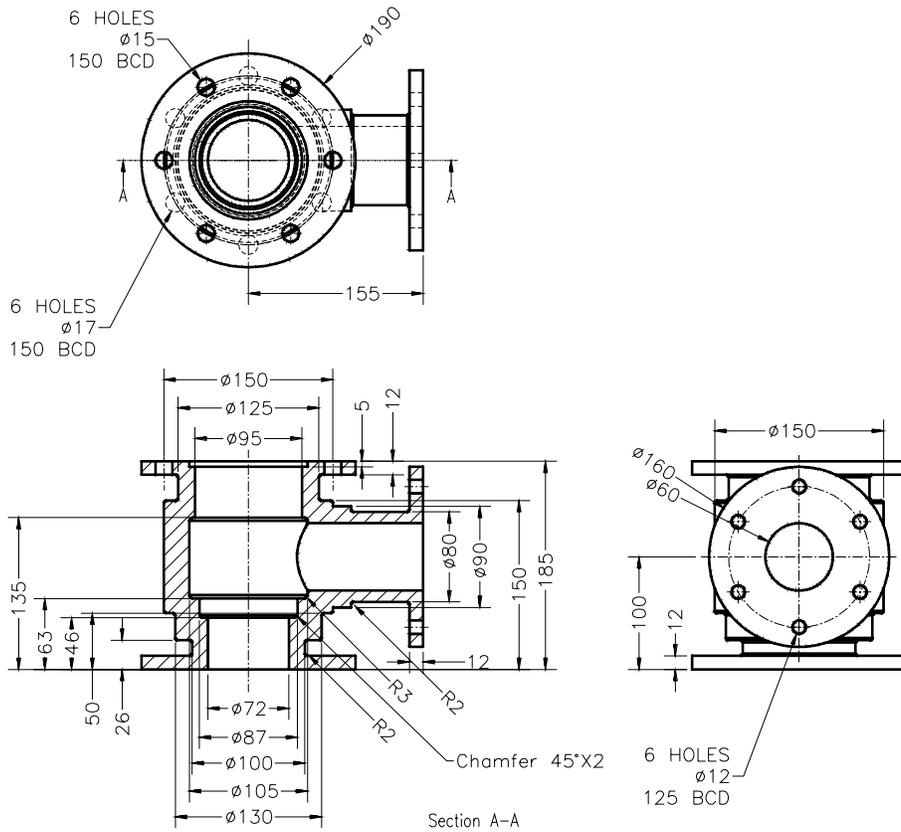


Figure 5-104 Dimensions of the model for Exercise 1

Exercise 2

Create the model shown in Figure 5-105. The dimensions of the model are given in Figure 5-106. **(Expected time: 30 min)**

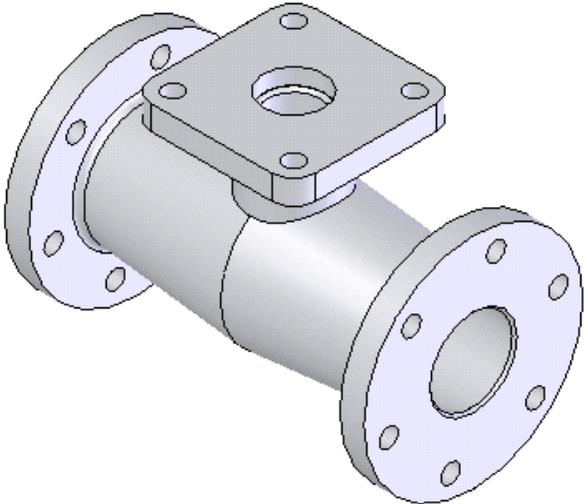


Figure 5-105 Model for Exercise 2

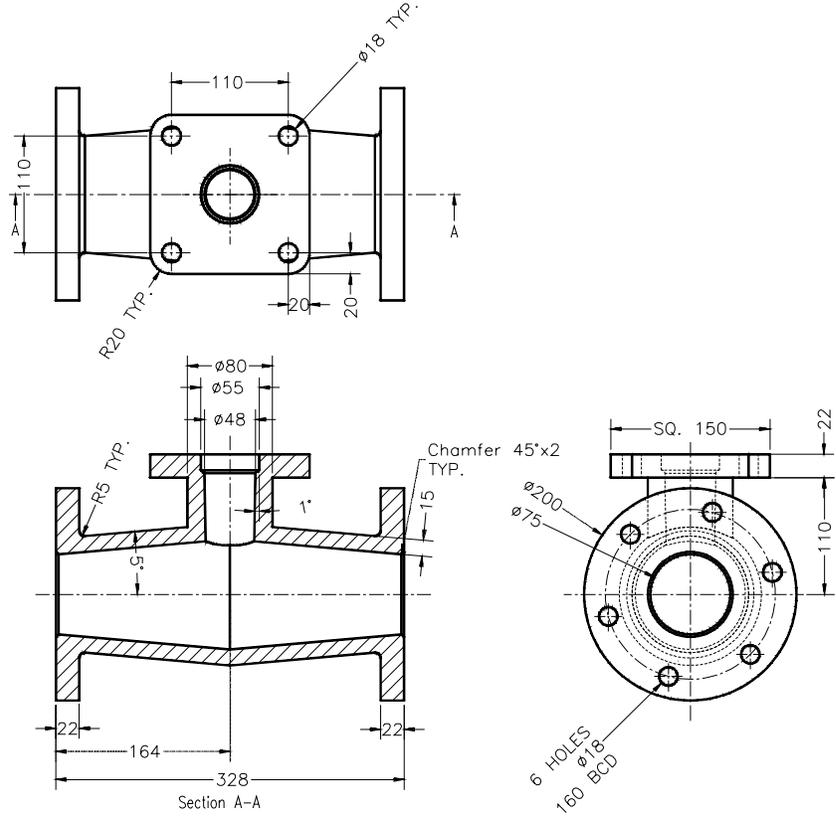


Figure 5-106 Dimensions for Tutorial 2

Answers to Self-Evaluation Test

1. T, 2. F, 3. T, 4. F, 5. T, 6. Up To Next, 7. Up To Body, 8. Merge results, 9. Cylindrical/Conical Surface, 10. Bodies to Keep