Chapter 5

Advanced Dimensioning Techniques and Base Feature Options



Learning Objectives

After completing this chapter, you will be able to:

- *Fully define a sketch.*
- *Dimension the true length of an arc.*
- *Create solid base extruded features.*
- *Create thin base extruded features.*
- *Create solid base revolved features.*
- *Dynamically rotate the view to display the model in all directions.*
- *Apply materials to models.*
- *Change the appearance of models.*

In this chapter, you will learn about the advanced dimensioning techniques that are used to dimension the sketches. In SolidWorks, you can apply all possible relations and dimensions to a sketch by using a single tool. Also, you will learn about the tools that are used to convert a sketch into a base feature of a model in the **Part** environment.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 5-1. The sketch of the model is shown in Figure 5-2. First, you will draw the sketch of the model and make it fully defined by applying the required relations and dimensions in the sketching environment. Next, you will convert the sketch into a model by extruding it in two directions. The parameters for extruding the sketch are given next.

Direction 1
Depth = 10 mm
Draft angle = 35 degrees

Direction 2
Depth = 15 mm
Draft angle = 0 degree

After creating the model, you will turn on the option to display shadows and also apply Alloy Steel (SS) material to the model. Additionally, you will determine the mass properties of the model. **(Expected time: 30 min)**

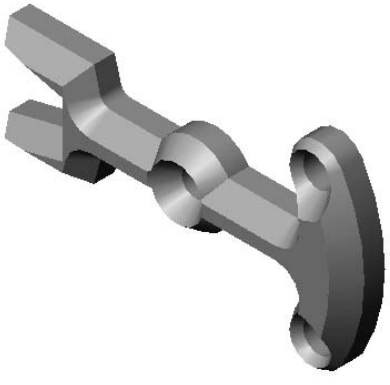


Figure 5-1 *Solid Model for Tutorial 1*

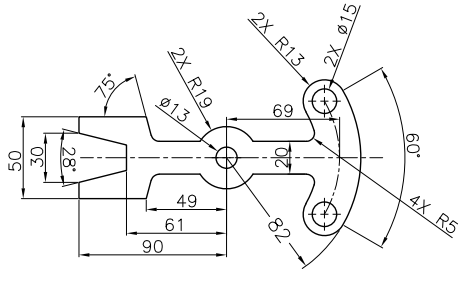


Figure 5-2 *Sketch of the model*

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The following steps are required to complete this tutorial:

a. Create the sketch of the model using the sketching tools, refer to Figures 5-3 through 5-14.
b. Apply dimensions to the sketch and make it fully defined, refer to Figure 5-15.
c. Invoke the **Extruded Boss/Base** tool and convert the sketch into a model, refer to Figures 5-30 through 5-33.
d. Display the shadow of the model, refer to Figure 5-35.
e. Assign materials to the model, refer to Figure 5-36.
f. Determine the mass properties of the model, refer to Figure 5-37.

Drawing the Sketch

1. Start SolidWorks and then invoke a new Part document. Next, choose the **Sketch** button from the **Sketch CommandManager** and then select the **Front Plane** as the sketching plane to invoke the sketching environment.

   As evident from Figure 5-2, the sketch of the model is symmetric about its centerline, therefore, you need to draw it with the help of a mirror line.

2. Increase the display of the drawing area by using the **Zoom In/Out** tool and invoke the **Centerline** tool from the **Line** flyout of the **Sketch CommandManager**.



3. Move the line cursor to a location whose coordinates are close to -102 mm, 0 mm, and 0 mm and specify the start point of the centerline at this point.

4. Move the line cursor horizontally toward the right and specify the endpoint of the centerline when the length of the line shows a value close to 204.

5. Exit the **Centerline** tool.

6. Choose the **Zoom to Fit** button from the **View (Heads-Up)** toolbar to fit the sketch into the drawing area.



7. Choose the **Centerpoint Arc Slot** button from the **Slot** flyout; the arrow cursor is replaced by the arc cursor and the **Slot PropertyManager** is displayed.



8. Move the arc cursor close to the origin. Click to specify the center point of the slot, when the cursor snaps to the origin. Next, move the cursor toward the right and below the horizontal axis and click when the arc cursor displays a value close to 69, refer to Figure 5-3.

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9. Move the arc cursor in a counterclockwise direction. Click to specify the endpoint of the arc slot when the value of the angle above the arc cursor is close to 60 degrees; a reference slot is attached to the cursor. Specify a point in the drawing area where the value of the width of the slot is close to 26 mm, refer to Figure 5-4.

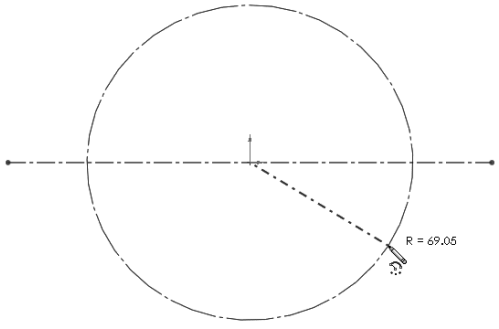


Figure 5-3 *Specifying the start point of the slot*

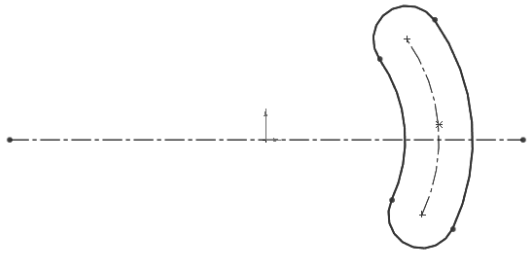


Figure 5-4 *The sketch after drawing the slot*

10. Choose the **Add Relation** button from the **Display/Delete Relations** flyout; the **Add Relations PropertyManager** is invoked.



11. Right-click in the drawing area and then choose the **Clear Selections** option from the shortcut menu to clear selections from the selection set. Select the centerline and the coordinate point of the slot, as shown in Figure 5-5; the **Coincident** button is highlighted in bold in the **Add Relations** rollout of the **Add Relation PropertyManager**. This indicates that the coincident relation is the most appropriate relation for the selected entities.

12. Choose the **Coincident** button from the **Add Relations PropertyManager**.

13. Choose the **OK** button from the **Add Relations PropertyManager** or choose **OK** from the confirmation corner. The sketch after applying the coincident relation is shown in Figure 5-6.

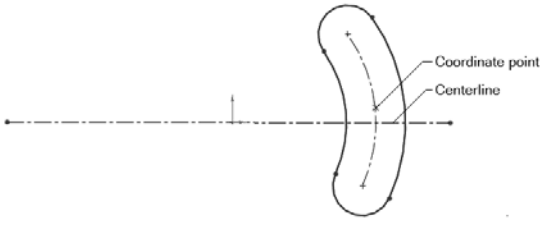


Figure 5-5 *The centerline and the coordinate point to be selected*

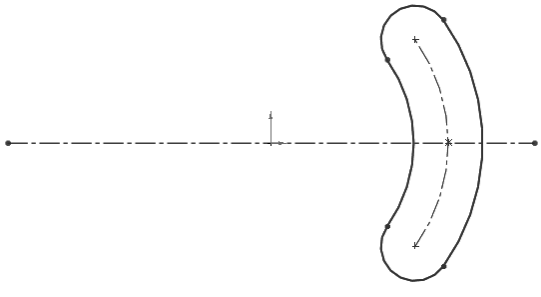


Figure 5-6 *The sketch after applying the coincident relation*

Next, you need to make the centerline as the mirror line and draw the sketch on the upper side of the mirror line. On doing so, the same sketch is reflected on the other side of the mirror line.

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14. Choose **Tools > Sketch Tools > Dynamic Mirror** from the SolidWorks menus and select the centerline; the centerline is converted into a mirror line.

15. Invoke the **Line** tool from the **Sketch CommandManager**. Next, move the cursor to the point whose coordinates are close to -61, 0, and 0 and then click to specify the start point of the line.

16. Move the cursor vertically upward and draw a line of length close to 7.

17. Move the cursor toward the left at an angle of 166-degree with the horizontal axis. Refer to the **Parameters** rollout of the PropertyManager for the angle measurement. Next, click to specify the endpoint of the line where the length of the line is close to 30; the mirrored entities are created on the other side.

18. Move the cursor vertically upward and specify the endpoint of the vertical line where the length of the line above the line cursor displays a value close to 10.

19. Move the cursor horizontally toward right and specify the endpoint where the length of the line above the line cursor displays a value close to 42.ss

20. Move the cursor downward at an angle of 285-degree, refer to the **Parameters** rollout of the PropertyManager for angle measurement. Next, specify the endpoint of the line where the length is close to 15. The sketch after drawing the inclined line is shown in Figure 5-7.

21. Move the cursor horizontally toward the right and specify the endpoint when the line cursor snaps to the left arc of the slot, refer to Figure 5-8. Next, exit the **Line** tool.

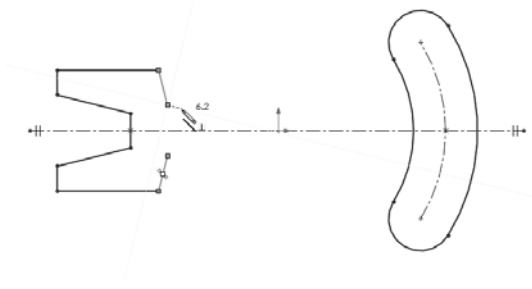


Figure 5-7 *Sketch after drawing the inclined line*

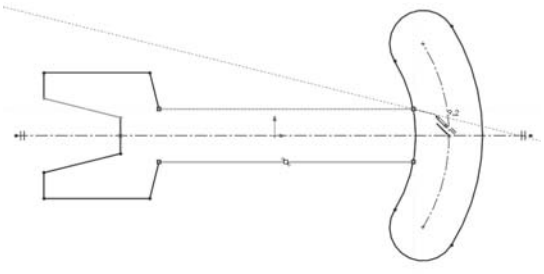


Figure 5-8 *Sketch after creating the horizontal lines*

22. Choose the **Circle** button from the **Sketch CommandManager**.

23. Move the cursor to the endpoint of the slot and specify the center point of the circle when the center point is highlighted. Next, move the cursor horizontally toward the right and click the left mouse button when the radius of the circle above the circle cursor is close to 7.5. Figure 5-9 shows the sketch after drawing circles.

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24. Right-click in the drawing area and then choose **Recent Commands > Dynamic Mirror Entities** to exit the **Dynamic Mirror** tool.

25. Invoke the **Circle** tool again from the **Sketch CommandManager**.

26. Move the cursor to the origin and click to specify the center point of the circle when cursor snaps to the origin. Next, move the cursor horizontally toward the right and click the left mouse button when the radius of the circle above the circle cursor shows a value close to 6.5.

27. Similarly, create one more circle of radius 19 approximately, with its center point at the origin. The sketch after drawing the required slot, circles, and lines is shown in Figure 5-10.

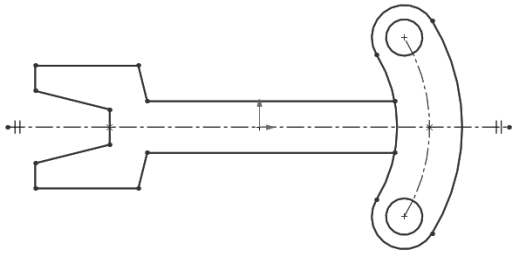


Figure 5-9 *Sketch after drawing two circles*

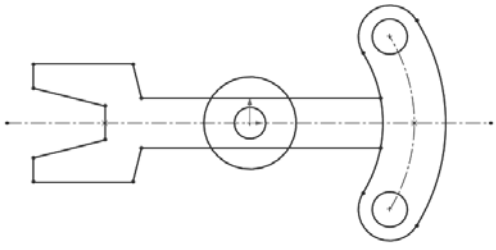


Figure 5-10 *Sketch after drawing slots, circles, and lines*

Trimming Unwanted Entities

After drawing the sketch, you need to trim some of the unwanted sketched entities using the **Trim Entities** tool.

1. Choose the **Trim Entities** button from the **Sketch CommandManager** to invoke the **Trim PropertyManager**.



2. Choose the **Trim to closest** button from the **Options** rollout of the PropertyManager, if it is not chosen by default; the select cursor is replaced by the trim cursor.

3. Select the entities to be trimmed, refer to Figure 5-11; the entities are dynamically trimmed. Next, exit the tool.



Note

While trimming the unwanted sketches of the slot, the ***SolidWorks 2012*** *message box is displayed with the message* ***This trim operation will destroy the slot entity. Do you want to continue?****. Choose the* ***OK*** *button to continue the trimming operation.*

As the trim operation destroys the slot entities, you need to apply the tangent and symmetric relations to them.

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4. Apply the tangent and symmetric relations to the slot entities by using the **Add Relations PropertyManager**, refer to Figure 5-12. Also, make sure that the center points of both the circles that having diameter 15 mm merge with the center points of the arcs of the slot.

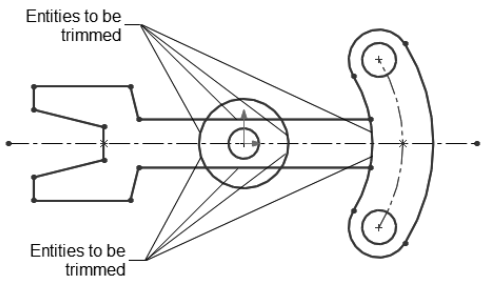


Figure 5-11 *The entities to be trimmed*

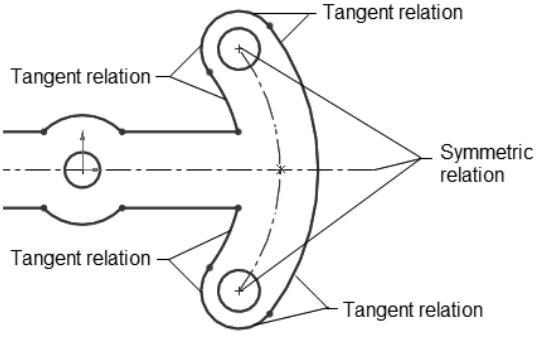


Figure 5-12 *Relations to be applied to the slot entities*

Filleting Sketched Entities

Next, you need to fillet the sketched entities. Fillets are generally added to avoid the stress concentration at sharp corners.

1. Choose the **Sketch Fillet** button from the **Sketch CommandManager**; the **Sketch Fillet PropertyManager** is displayed. Next, set the value **5** in the **Radius** spinner of the PropertyManager.

2. Select the set of entities one by one to apply fillet, refer to Figure 5-13. Next, exit the **Sketch Fillet** tool. Figure 5-14 shows the sketch after applying fillets of radius 5 mm.

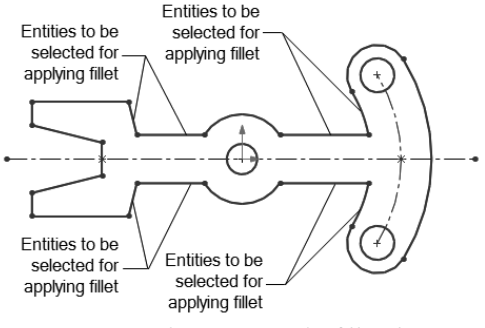


Figure 5-13 *The entities to be filleted*

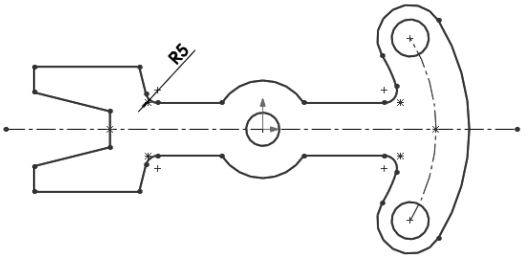


Figure 5-14 *Sketch after filleting the sketched entities*

Adding Dimensions to the Sketch

Next, you need to apply dimensions to the sketch and fully define it.

1. Choose the **Smart Dimension** button from the **Sketch CommandManager**; the arrow cursor is replaced by the dimension cursor.



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2. Select the construction arc of the slot that is displayed as a construction entity; a radius dimension is attached to the cursor. Move the cursor away from the sketch toward the right and place the dimension; the **Modify** dialog box is displayed.

3. Enter **69** in the **Modify** dialog box and press ENTER, refer to Figure 5-15.

4. Select the upper arc of the slot; a radius dimension is attached to the cursor. Move the cursor away from the sketch toward the right and place the dimension.

5. Enter **13** in the **Modify** dialog box and press ENTER, refer to Figure 5-15.

6. Select the origin and the start point of the construction arc of the slot; a dimension is attached to the cursor. Next, select the endpoint of the construction arc; an angular dimension is attached to the cursor. Place the angular dimension outside the sketch.



Tip. *In SolidWorks, you can also apply angular dimensions to an arc by using the* ***Smart Dimension*** *tool. To do so, you need to select two endpoints and the center point of the arc.*

7. Enter **60** as the value of the angular dimension in the **Modify** dialog box and press ENTER, refer to Figure 5-15.

8. Select the upper right circle; a diameter dimension is attached to the cursor. Place the dimension outside the sketch.

9. Enter **15** as the value of the diameter dimension in the **Modify** dialog box and press ENTER, refer to Figure 5-15.

10. Select the upper right horizontal line and the lower right horizontal line that coincides with the trimmed circle; a vertical dimension is attached to the cursor. Move the cursor vertically upward and click to place the dimension.

11. Enter **20** in the **Modify** dialog box and press ENTER, refer to Figure 5-15

12. Select the circle at the origin; a diameter dimension is attached to the cursor. Move the cursor upward and place the dimension outside the sketch.

13. Enter **13** as the value of the diameter dimension in the **Modify** dialog box and press ENTER.

14. Select the outer trimmed circle and place the radius dimension outside the sketch.

15. Enter **19** as the value of the radial dimension in the **Modify** dialog box and press ENTER.

16. Select the upper right inclined line; a dimension is attached to the cursor. Next, select the upper left horizontal line; an angular dimension is attached to the cursor. Place the dimension above the upper left horizontal line, refer to Figure 5-15.

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17. Enter **75** as the value of the angular dimension in the **Modify** dialog box and press ENTER.

18. Select the origin and the lower endpoint of the lower right inclined line. Move the cursor vertically downward and place the dimension. Enter **49** in the **Modify** dialog box and press ENTER, refer to Figure 5-15.

19. Select the origin and the middle left vertical line, refer to Figure 5-15. Move the cursor vertically downward and place the dimension below the previously placed dimension.

20. Enter **61** in the **Modify** dialog box and then press ENTER.

21. Select the origin and the lower endpoint of the outer left vertical line, refer to Figure 5-15. Move the cursor vertically downward and place the dimension below the last placed dimension.

22. Enter **90** in the **Modify** dialog box and then press ENTER.

23. Select the upper left and lower left inclined lines, refer to Figure 5-15; an angular dimension is attached to the cursor. Move the cursor horizontally toward the left and place the dimension.

24. Enter **28** as the value of the angular dimension in the **Modify** dialog box and then press ENTER.

25. Select the upper left and the lower horizontal lines; a linear dimension is attached to the cursor. Move the cursor horizontally toward the left and place the dimension.

26. Enter **50** in the **Modify** dialog box and then press ENTER.

27. Select the lower endpoint of the upper left vertical line and the upper endpoint of the lower left vertical line; a linear dimension is attached to the cursor. Move the cursor horizontally toward the left and click to place the dimension.

28. Enter **30** in the **Modify** dialog box and then press ENTER.

29. Add the remaining dimensions to the sketch and make it a fully defined sketch. Figure 5-15 shows the fully defined sketch.



Tip. *In SolidWorks, you can also fully defined a sketch by applying the required relations and dimensions to it automatically using the* ***Fully Define Sketch*** *tool. To fully define a sketch, draw the sketch using the standard sketching tools. Next, choose the* ***Fully Define Sketch*** *button from the* ***Display/Delete Relations*** *flyout in the* ***Sketch CommandManager****; the* ***Fully Define Sketch PropertyManager*** *will be displayed. You can also right-click and choose the* ***Fully Define Sketch*** *option from the shortcut menu to display this PropertyManager. Using this PropertyManager, you can make all the entities of a sketch or selected entities of a sketch fully defined by selecting the respective option from the PropertyManager.*

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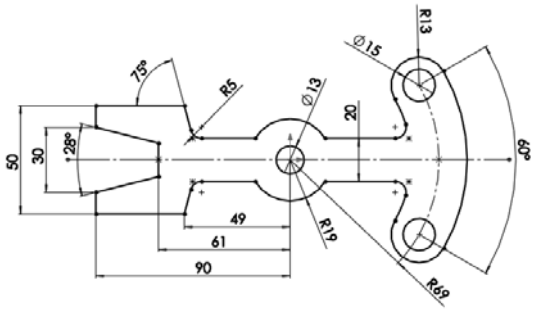


Figure 5-15 *Sketch after applying all relations and dimensions*

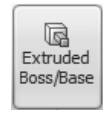
Extruding the Sketch

After creating the sketch, you need to convert it into a base feature. To do so, you need to invoke the **Extruded Boss/Base** tool and extrude the sketch using the parameters given in the tutorial description.

1. Choose the **Features** tab from the **CommandManager** to display the **Features CommandManager** tools.

The **Features CommandManager** provides all the modeling tools that are used in feature-based solid modeling.

2. Choose the **Extruded Boss/Base** button from the **Features CommandManager**; the sketch is automatically oriented to the trimetric view and the **Boss-Extrude PropertyManager** is displayed, as shown in Figure 5-16.



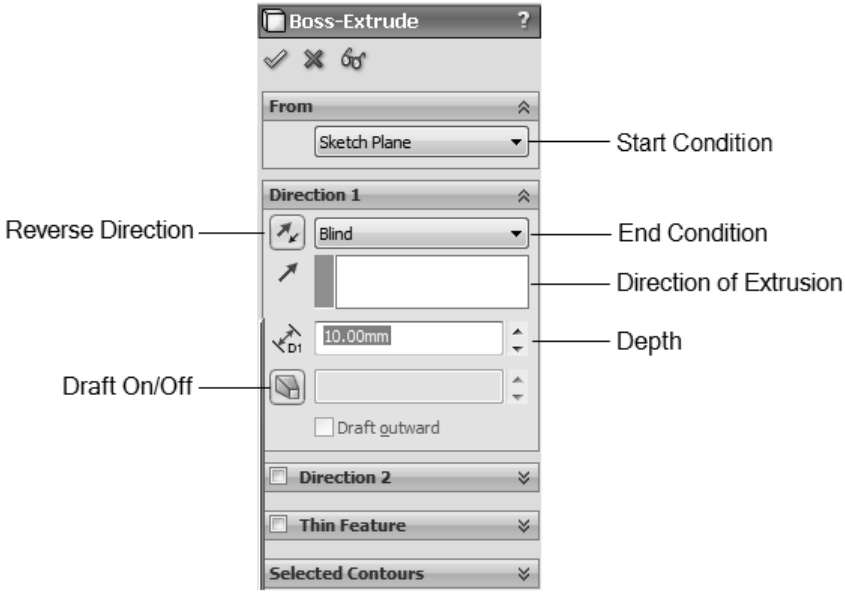


Figure 5-16 *The* ***Boss-Extrude PropertyManager***

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Also, you will notice that the preview of the base feature is displayed in the temporary shaded graphics with the default values, refer to Figure 5-17. Additionally, an arrow will appear in front of the sketch. Note that the closed loops that are available inside the outer loop of the sketch are automatically subtracted from the outer loop while extruding it, refer to Figure 5-17.

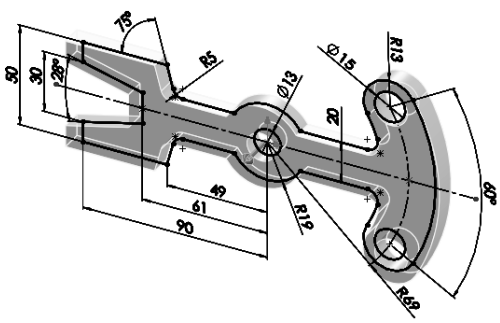


Figure 5-17 *Preview of the feature being extruded*

The **Extruded Boss/Base** tool is used to add material to the area defined by a sketch.

3. Make sure that the **Sketch Plane** option is selected in the **Start Condition** drop-down list of the **From** rollout in the **Boss-Extrude PropertyManager**.

In the **Boss-Extrude PropertyManager**, the **Sketch Plane** option is selected by default in the **Start Condition** drop-down list of the **From** rollout. Therefore, the resulting extrude feature will start from the sketching plane on which the sketch is drawn. This option is mostly used while creating the extrude feature, refer to Figure 5-17. In SolidWorks, you can also select the **Surface/Face/Plane**, **Vertex**, and **Offset** options from the **Start Condition** drop-down list of the **From** rollout. The **Surface/Face/Plane** option is used to start the extrude feature from a selected surface, face, or a plane, instead of the plane on which the sketch is drawn. Figure 5-18 shows the sketch to be extruded and the face to be selected as the face to start extrusion. Figure 5-19 shows the resulting extrude feature created on the selected face up to a specified depth.

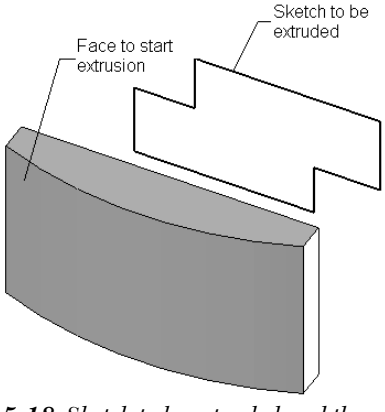


Figure 5-18 *Sketch to be extruded and the reference face selected*

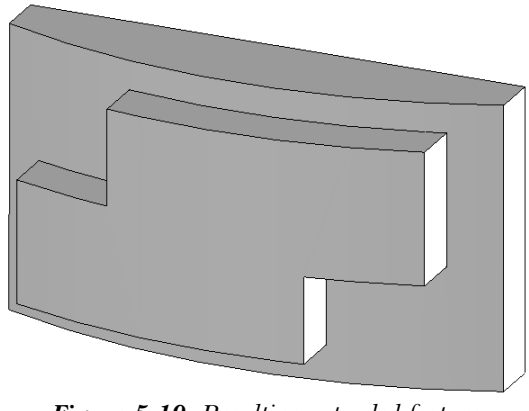


Figure 5-19 *Resulting extruded feature*

The **Vertex** option of the **Start Condition** drop-down list is used to specify a vertex as a reference for starting the extrude feature. Figure 5-20 shows the sketch to be extruded and the vertex to be selected as a reference to start the extrude feature. Figure 5-21 shows the resulting extruded feature created on the selected vertex to the defined depth.

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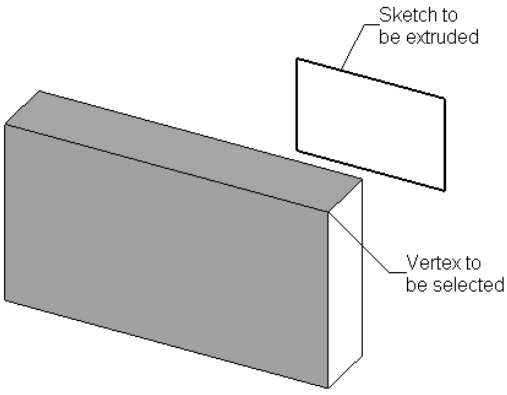


Figure 5-20 *Sketch to be extruded and a reference vertex to be selected*

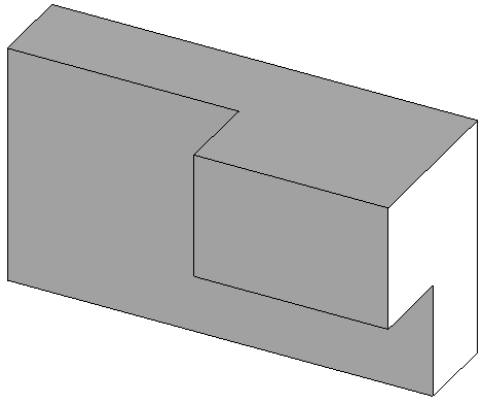


Figure 5-21 *Resulting extruded feature*

The **Offset** option of the **Start Condition** drop-down list is used to start the extrude feature at an offset distance from the plane on which the sketch is drawn. Figure 5-22 shows the preview of a sketch drawn on the front plane and being extruded using the **Offset** option. Figure 5-23 shows the resulting extruded feature created at an offset distance from the sketching plane.

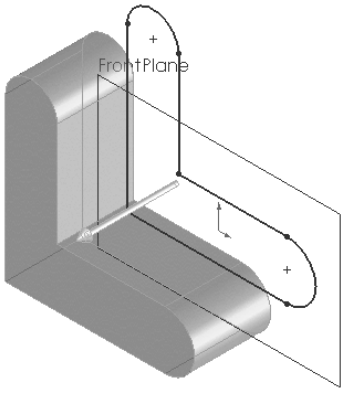


Figure 5-22 *Sketch being extruded using the* ***Offset*** *option*

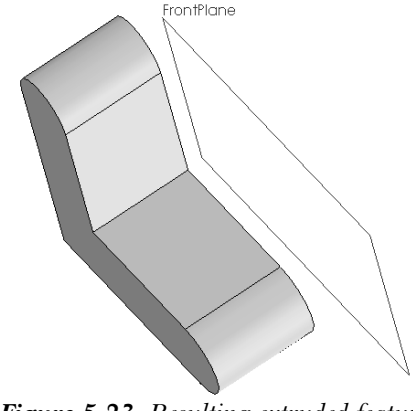


Figure 5-23 *Resulting extruded feature*

4. Make sure that the **Blind** option is selected in the **End Condition** drop-down list of the **Direction 1** rollout of the **Boss-Extrude PropertyManager**.

The **End Condition** drop-down list of the **Direction 1** rollout in the PropertyManager is used to define the termination of extrude features in one direction from the sketching plane. Note that the **Blind** option is selected by default in this drop-down list and is used to define the termination of the extrude features by specifying the depth of extrusion. The depth of extrusion is specified in the **Depth** spinner of the PropertyManager. The other options of this drop-down list are discussed next.

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Mid Plane

The **Mid Plane** option of the **End Condition** drop-down list is used to create a feature by extruding the sketch equally in both the directions of the plane on which the sketch is drawn.

Up To Vertex

The **Up To Vertex** option of the **End Condition** drop-down list is used to define the termination of a feature at a virtual plane that is parallel to the sketching plane and passes through the selected vertex. The vertex can be a point on an edge, a sketched point, or a reference point. Figure 5-24 shows a sketch drawn on a plane at an offset distance and Figure 5-25 shows the model in which the sketch is extruded up to the selected vertex.

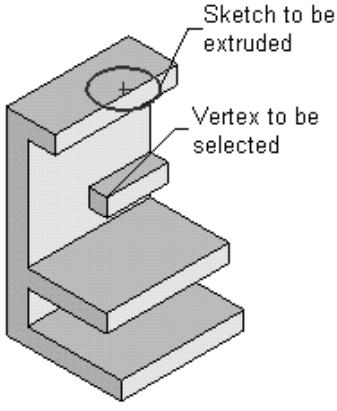


Figure 5-24 *Sketch drawn on a plane created at an offset distance and the vertex to be selected*

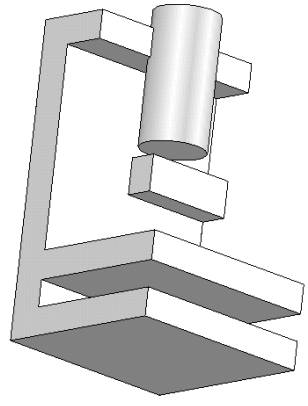


Figure 5-25 *Sketch extruded using the* ***Up To Vertex*** *option*

Up To Surface

The **Up To Surface** option of the **End Condition** drop-down list is used to define the termination of a feature up to a selected surface or face. Figure 5-26 shows the sketch drawn at an offset distance and the surface to be selected. Figure 5-27 shows the resulting feature extruded up to the selected surface.

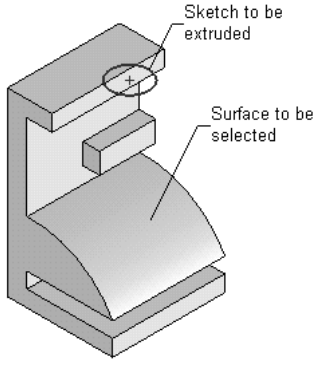


Figure 5-26 *Sketch drawn on a plane created at an offset distance and the surface to be selected*

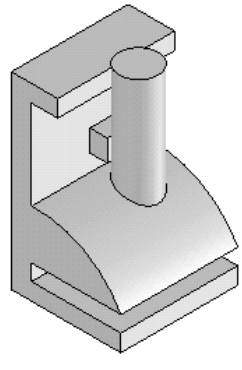


Figure 5-27 *Sketch extruded using the* ***Up To Surface*** *option*

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Offset From Surface

The **Offset From Surface** option is used to define the termination of a feature on a virtual surface that is created at an offset distance from the selected surface.

Up To Body

The **Up To Body** option is used to define the termination of the extruded feature to another body. Figure 5-28 shows the sketch for the extruded feature and a body up to which the sketch will be extruded. Figure 5-29 shows the resulting feature.

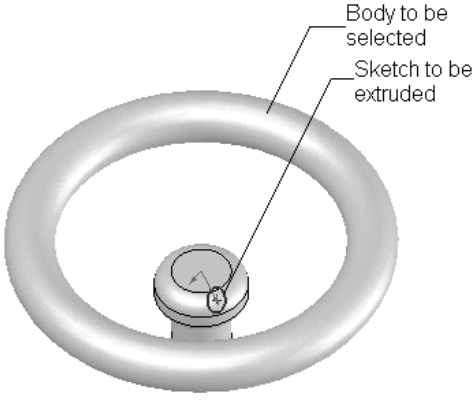


Figure 5-28 *Sketch to be extruded and the body to be selected for the extrude feature*



Figure 5-29 *Sketch extruded using the* ***Up To Body*** *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. This option will be available in the **End Condition** drop-down list only after you create a base feature.

Through All

The **Through All** option is used to extrude the sketch from the sketching plane to all the existing geometric entities. This option will be available in the **End Condition** drop-down list only after you create a base feature.

5. Set the value of the **Depth** spinner to 10 mm.



Tip. *In SolidWorks, you can also extrude a sketch to a blind depth by dragging the feature dynamically using the mouse. To do so, move the mouse to the arrow displayed in the preview; the move cursor will be displayed and the color of the arrow will also be changed. Left-click once on the arrow; a scale will be displayed. Now, move the cursor; the value of the depth of extrusion will change dynamically on this scale as you move the cursor. Left-click again to specify the termination of the extruded feature.*

6. Choose the **Draft On/Off** button from the **Direction 1** rollout of PropertyManager to enable the **Draft Angle** spinner.



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The **Draft On/Off** button is used to specify a draft angle while extruding a sketch. Applying a draft angle results in tapering of the resulting feature. By default, the feature will be tapered inward. However, if you want to taper the feature outward, you need to select the **Draft outward** check box that is displayed below the **Draft Angle** spinner.

7. Set the value of the **Draft Angle** spinner to 35 mm; the preview of the feature is modified, as shown in Figure 5-30.

   These are the settings for direction 1. Next, you need to specify the settings for direction 2.

8. Select the **Direction 2** check box to invoke the **Direction 2** rollout. As soon as you invoke the **Direction 2** rollout, another arrow is displayed in the preview of the feature. Also, the material is added along the second direction with the default depth value, refer to Figure 5-31.

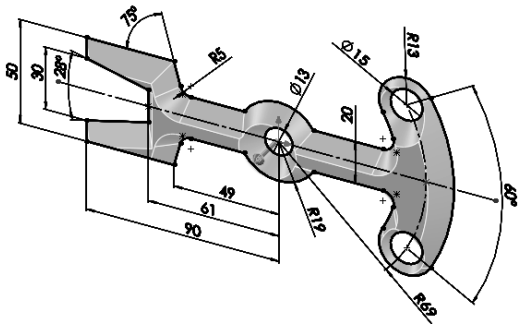


Figure 5-30 *Preview of the extrude feature after applying the draft angle*

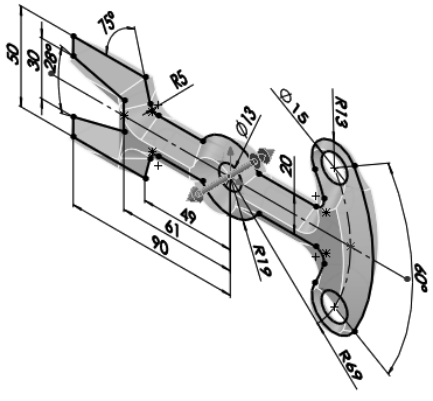


Figure 5-31 *Preview of the extrude feature after invoking the* ***Direction 2*** *rollout*

The **Direction 2** rollout is used to extrude a sketch with different values in the second direction of the sketching plane. You can expand this rollout by selecting the check box available in its title bar. This check box will not be enabled, if the **Mid Plane** option is selected in the **End Condition** drop-down list of the **Direction 1** rollout. The options available in the **Direction 2** rollout are similar to the options in the **Direction 1** rollout.

9. Choose the **Draft On/Off** button in the **Direction 2** rollout to turn it off, if it is on. This is because you do not require the draft angle in the second direction.

10. Set **15** in the **Depth** spinner of the **Direction 2** rollout as the depth in the second direction.

11. Choose the **OK** button to create the feature or choose **OK** from the confirmation corner. The feature is created and its name is displayed in the **FeatureManager Design Tree**.

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The **FeatureManager Design Tree** that is available on the left side of the drawing area is one of the most important components of SolidWorks screen. It contains information about the default planes, materials, lights, and all other features that are added to a model. When you add features to a model using the modeling tools, the added features are also displayed in the **FeatureManager Design Tree**. It stores and displays all features in hierarchical manner.

It is recommended that you change the view to isometric after creating the feature to view it properly.

12. Choose the **View Orientation** button from the **View (Heads-Up)** toolbar; the **View Orientation** flyout is displayed, refer to Figure 5-32. Next, choose the **Isometric** button from it. The isometric view of the resulting solid model is shown in Figure 5-33.

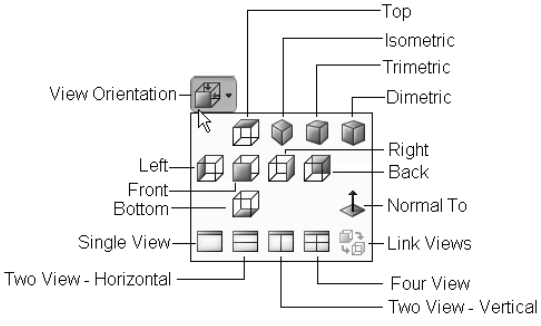


Figure 5-32 *The **View Orientation** flyout*

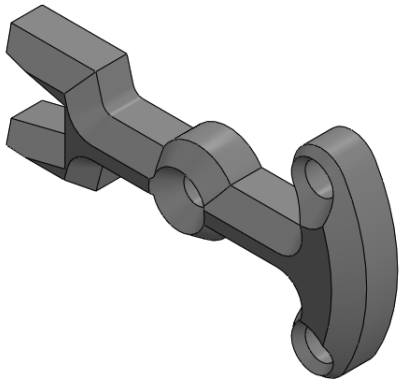


Figure 5-33 *Resulting extruded feature*

The tools available in the **View Orientation** flyout are used to change the view orientation as per the predefined standard views. This flyout also provide tools to display a model in two or four viewports.

Displaying the Shadow

As mentioned in the tutorial description, you need to display the shadow of the model, if it is not displayed by default. You can turn on the display of the shadow using the **View (Heads-Up)** toolbar.

1. Choose **View Settings** from the **View (Heads-Up)** toolbar; the **View Setting** flyout is displayed, refer to Figure 5-34. Next, choose the **Shadows In Shaded Mode** button from this flyout to display the model with shadow, as shown in Figure 5-35.

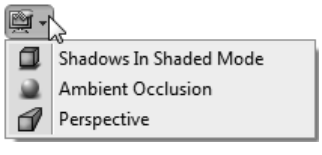


Figure 5-34 *The **View Setting** flyout*

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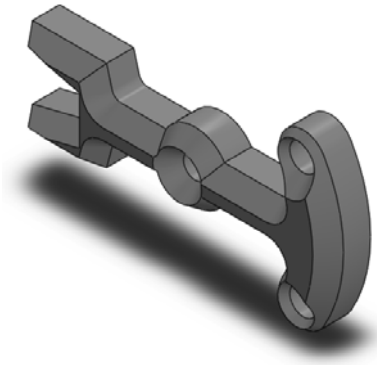


Figure 5-35 *Model with the display of shadow turned on*

Assigning Materials to the Model

As mentioned in the tutorial description, you need to apply Alloy Steel (SS) material to the model. When you apply material to a model, the physical properties such as Density, Young's modulus, and so on of the selected material are assigned to the model. As a result, when you calculate the mass properties of the model, they will be based on the physical properties of the material applied.

1. Choose **Edit > Appearance > Material** from the SolidWorks menus; the **Material** dialog box is displayed. Alternatively, right-click on the **Material <not specified>** option in the **FeatureManager Design Tree** and select **Edit Material** from the shortcut menu to invoke the **Material** dialog box.

The **Material** dialog box displays the list of all materials available in the material library of SolidWorks. A number of material families are available in the left area of the **Material** dialog box. When you click on the (+) sign located on the left of a material family, it displays a list of all materials under that family.

2. Select **Alloy Steel (SS)** from the list of materials available in the **Steel** family; all the properties of the Alloy Steel (SS) material are displayed on the right side of the dialog box.

3. Choose the **Apply** button from the **Material** dialog box and then choose the **Close** button to exit. The model, after assigning the material, is shown in Figure 5-36.

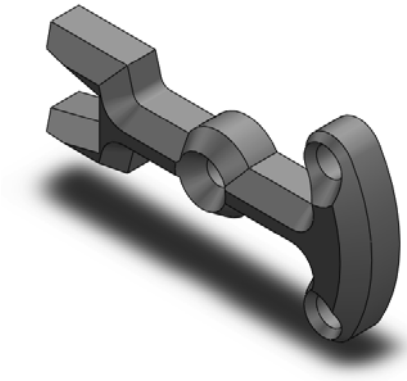


Figure 5-36 *Model after assigining the Alloy Steel (SS) material*

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Determining the Mass Properties of the Model

After assigning the material to the model, you need to calculate the mass properties of the model.

1. Choose the **Mass Properties** tool from the **Evaluate CommandManager**; the **Mass Properties** dialog box with the mass properties of the current model is displayed, as shown in Figure 5-37.



The **Mass Properties** tool is used to determine the mass properties of the part or assembly that is available in the current session. Note that this tool will not be enabled, if there is no solid model available in the current session. The mass properties include density, mass, volume, surface area, center of mass, principal axes of inertia and principal moments of inertia, and moments of inertia.

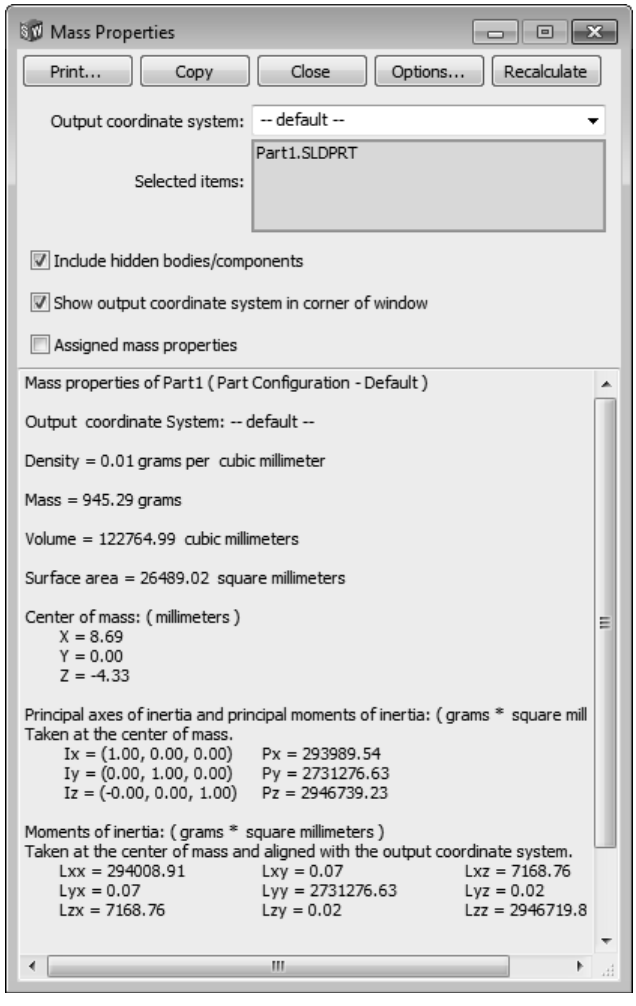


Figure 5-37 *The* ***Mass Properties*** *dialog box*

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Saving the Model

1. Choose the **Save** button from the Menu Bar to invoke the **Save As** dialog box. Next, browse to *\Documents\SolidWorks Tutorials* and create a new folder with the name *c05*.

2. Enter **c05_tut01** as the name of the model in the **File name** edit box of the **Save As** dialog box and then choose the **Save** button. The model is saved at the location *\Documents\SolidWorks Tutorials\c05*.

3. Choose **File > Close** from the SolidWorks menus to close the document.

Tutorial 2

In this tutorial, you will open the sketch drawn in Tutorial 2 of Chapter 2. Then you will apply the required relations and dimensions to the sketch to make it fully defined. You will also determine the section properties of the sketch and convert it into a revolve feature by revolving the sketch through an angle of 270 degrees, as shown in Figure 5-38.

(Expected time: 45 min)

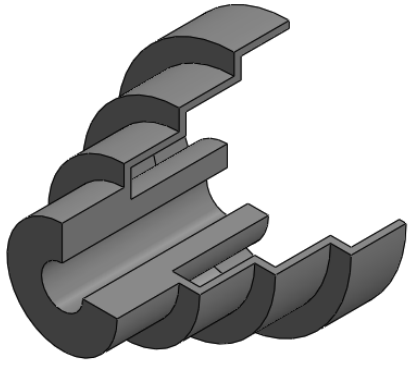


Figure 5-38 *Model for Tutorial 2*

The following steps are required to complete this tutorial:

a. Open Tutorial 2 of Chapter 2, refer to Figure 5-41.
b. Save this document in the *c05* folder with a new name.
c. Apply required relations and dimensions to the sketch, refer to Figures 5-40 through 5-42.
d. Determine the section properties of the sketch, refer to Figure 5-43.
e. Invoke the **Revolved Boss/Base** tool and revolve the sketch through an angle of 270 degrees, refer to Figure 5-44.
f. Change the current view to isometric view and then save the document.
g. Save the model.

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Opening Tutorial 2 of Chapter 2

As the required document is saved in the *c02* folder, first you need to open it in SolidWorks 2012.

1. Choose the **Open** button from the SolidWorks menus; the **Open** dialog box is displayed.



The **Open** tool is used to open an existing SolidWorks part, assembly, or drawing document. You can also import files from other applications saved with standard file formats by using this tool.

2. Browse to the *SolidWorks Tutorials* folder and then double-click on the *c02* folder.

3. Select the **c02_tut02.sldprt** document and then choose the **Open** button from the dialog box to open the selected part in the current session of SolidWorks.

   As the sketch was saved in the sketching environment in Chapter 2, it opens in the sketching environment.

Saving the Document in c05 Folder

When you open a document of some other chapter, it is recommended that you first save the document with some other name in the folder of the current chapter (document) before modifying it. This ensure that the original document of the other chapter is not affected if any modifications are made in the current document.

1. Choose the **File > Save As** from the SolidWorks menus; the **Save As** dialog box is displayed.

2. Browse to *\SolidWorks Tutorials\c05* folder and then enter **c05_tut02** as the new name of the document in the **File name** edit box and then choose the **Save** button to save the document.

   The document is saved with the new name and is now displayed in the drawing area, as shown in Figure 5-39.

Applying Relations and Dimensions

As mentioned earlier, some of the relations are automatically applied to a sketch while drawing.

1. Apply the horizontal relation to the horizontal lines and vertical relation to the vertical lines of the sketch, if they are not applied automatically while drawing the sketch. You can apply relations by using the **Add Relations PropertyManager** or the pop-up toolbar that will be displayed after selecting the entity.

2. Apply the equal relation to those entities of the sketch that are of same length, refer to Figure 5-40.

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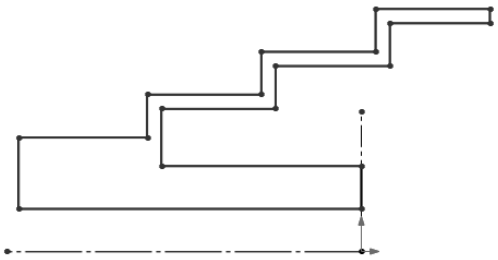


Figure 5-39 Sketch displayed in the drawing area

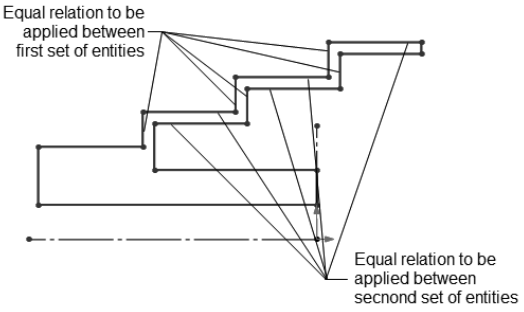


Figure 5-40 Equal relations to be applied

3. Invoke the **Smart Dimension** tool from the **Sketch CommandManager** and apply the linear diameter dimensions to the sketch, as shown in Figure 5-41.

   Next, you will apply the horizontal ordinate dimensions to the sketch by using the **Horizontal Ordinate Dimension** tool. You can also use the **Smart Dimension** tool for applying remaining dimensions to the sketch to make it fully defined.

4. Choose the **Horizontal Ordinate Dimension** button from the **Smart Dimension** flyout in the **Sketch CommandManager**.

The **Horizontal Ordinate Dimension** tool is used to dimension the horizontal distance of the selected entities from the specified datum.

Note

*When you apply an ordinate dimension, the **Modify** dialog box will not be displayed to modify the dimension value. In this case, to modify dimensions, first you need to place all the ordinate dimensions, exit the tool, and then double-click on the dimensions and modify the values.*

5. Select the upper right vertical line of the sketch as the datum entity from where the remaining entities are to be measured; the dimension value 0 is attached with the cursor.

6. Move the cursor vertically upward to a small distance and place the dimension by clicking the left mouse button, refer to Figure 5-42.

7. Select the other vertical entities of the sketch to apply the ordinate dimensions, refer to Figure 5-42. The fully defined sketch after applying the required relations and dimension is shown in Figure 5-42.

Determining the Section Properties of the Sketch

As mentioned in the tutorial description, you need to determine the section properties of the sketch.

1. Click on the **Evaluate** tab available in the **CommandManager** to invoke the **Evaluate CommandManager**.

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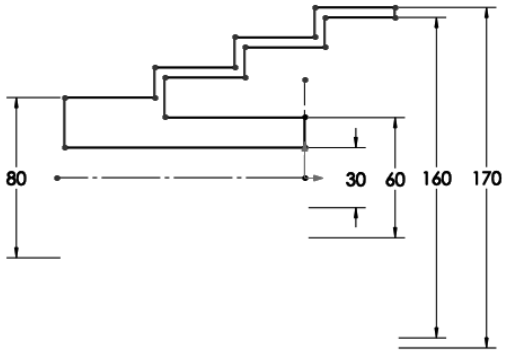


Figure 5-41 Sketch after applying the linear diameter dimensions

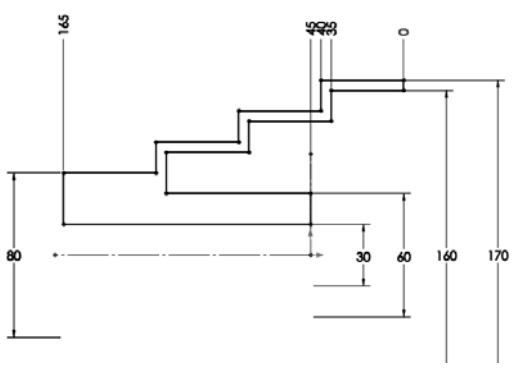


Figure 5-42 Sketch after applying the horizontal ordinate dimensions

2. Choose the **Section Properties** button from the **Evaluate CommandManager**; the **Section Properties** dialog box is displayed, as shown in Figure 5-43. Also, a 3D triad is placed at the centroid of the sketch.

The **Section Properties** dialog box displays the section properties of the sketch such as the area, centroid relative to sketch origin, centroid relative to the part origin, moment of inertia, polar moment of inertia, angle between principle axes and sketch axes, and principle moment of inertia.

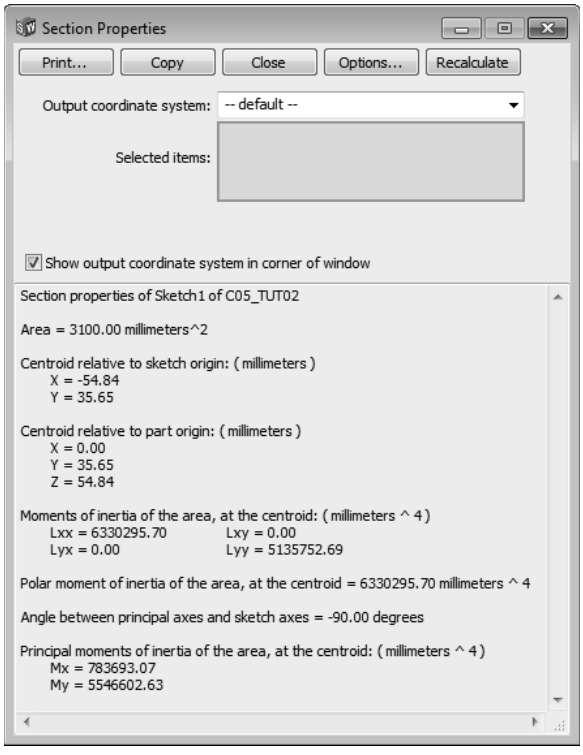


*Figure 5-43 The **Section Properties** dialog box*

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The **Section Properties** tool enables you to determine the section properties of the sketch in the sketching environment. You can also determine the section properties of the selected planar face in the **Part** mode or in the **Assembly** mode by using this tool.

Note

*The section properties of closed sketches with non-intersecting closed loops can only be determined by using the **Section Properties** tool.*

Revolving the Sketch

Now, you need to convert the sketch into a revolve feature by revolving it through an angle of 270 degrees around the horizontal centerline.

1. Choose the **Features** tab from the **CommandManager** to display the **Features CommandManager**.

2. Choose the **Revolved Boss/Base** button from the **Features CommandManager**; the sketch is automatically oriented in the trimetric view and the **Revolve PropertyManager** is displayed. As the sketch has two centerlines, SolidWorks cannot determine which one of them has to be used as an axis of revolution. As a result, you are prompted to select the axis of revolution.



3. Select the horizontal centerline, which has been used to create linear diameter dimensions, as the axis of revolution; the preview of a complete revolved feature in temporary shaded graphics is displayed in the drawing area. If the preview of the model is not displayed properly in the current view, you need to fit the view of the model into the drawing area.

4. Choose the **Zoom to Fit** button from the **View (Heads-Up)** toolbar or press the F key on the keyboard to fit the preview of the model into the drawing area.



5. In the **Direction 1** rollout, select the **Blind** option in the **Revolve Type** drop-down list, if it is not selected by default. Set the value of the **Direction 1 Angle** spinner to **270** and click anywhere on the screen; the preview of the revolved model is modified accordingly.

Note

In SolidWorks, you can specify the start and end conditions for a revolve feature similar to that of an extrude feature.

Note that if the horizontal centerline is drawn from left to right, then the direction of revolution has to be reversed to get the required model. You can reverse the direction of revolution by choosing the **Reverse Direction** button that is available at the left side of the **Revolve Type** drop-down list in the PropertyManager.

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Tip. *In SolidWorks, the right-hand thumb rule is followed for determining the direction of revolution. This rule states that if the thumb of the right hand points in the direction of the axis of revolution, the direction of the curled fingers will determine the default direction of revolution.*

6. Choose the **OK** button from the **Revolve PropertyManager**; the revolved feature is created.

7. Choose the **View Orientation** button from the **View (Heads-Up)** toolbar; a flyout is displayed. Choose the **Isometric** option from the flyout to orient the model to isometric view. The isometric view of the final revolved model is shown in Figure 5-44.

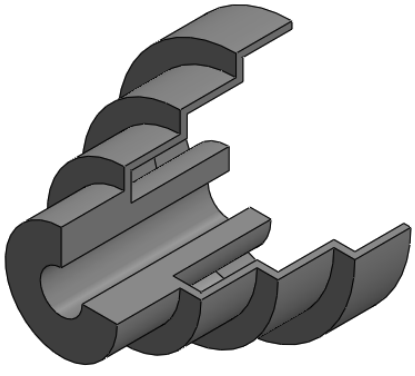


Figure 5-44 *Final model for Tutorial 2*

Saving the Model

As the name of the document was specified at the beginning, now you need to choose the **Save** button to save the document.

1. Choose the **Save** button from the Menu Bar to save the model. If the **SolidWorks** warning box is displayed, choose **Yes** from it to rebuild the model before saving. The model is saved at the location *\Documents\SolidWorks Tutorials\c05*

2. Choose **File > Close** from the SolidWorks menus to close the document.

Tutorial 3

In this tutorial, you will create the model shown in Figure 5-45. The sketch of the model is shown in Figure 5-46. First you will draw the outer loop of the sketch of the model and make it fully defined by applying the required relations and dimensions in the sketching environment. Next, you will convert that outer loop of the sketch into a thin extruded model by adding thickness of 3 mm in the inward direction. The depth of extrusion is 20 mm. After creating the model, add a color of your choice to it. **(Expected time: 45 min)**

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The following steps are required to complete this tutorial:

a. Create the sketch of the model. Apply the required relations and dimensions to make the sketch fully defined, refer to Figure 5-47.
b. Invoke the **Extruded Boss/Base** tool and convert the sketch into a thin extrude feature, refer to Figure 5-48.
c. Apply color to the model, refer to Figure 5-50.
d. Save the model.

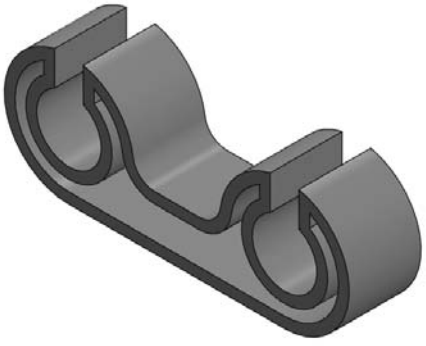


Figure 5-45 *Solid Model for Tutorial 3*

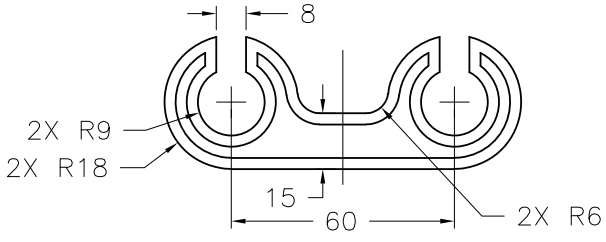


Figure 5-46 *Sketch of the model*

Drawing the Sketch

As mentioned in the tutorial description, you will first draw the outer loop of the sketch and then convert it into a thin extrude feature.

1. Start SolidWorks and then invoke a new Part document. Next, choose the **Sketch** button from the **Sketch CommandManager** and then select the **Front Plane** as the sketching plane to invoke the sketching environment.

2. Draw the outer loop of the sketch by using the sketching tools and apply the required relations and dimensions to it, refer to Figure 5-47.

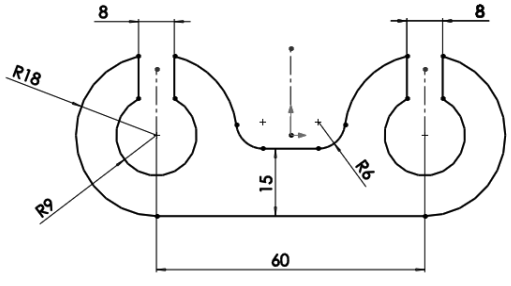


Figure 5-47 *Sketch of the outer loop*

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Creating Thin Extrude Feature

Now, you will convert the sketch into a thin extruded feature by adding thickness of 3 mm in the inward direction and depth of 20 mm.

1. Choose the **Features** tab from the **CommandManager** to display the **Features CommandManager**.

2. Choose the **Extruded Boss/Base** button from the **Features CommandManager**; the sketch is automatically oriented in the trimetric view and the **Boss-Extrude PropertyManager** is displayed. Also, the preview of the extrude feature is displayed in the drawing area with the default values.



3. Select the **Mid Plane** option from the **End Condition** drop-down list; the preview of the extrude feature is modified and the material is added to both sides of the sketching plane.

The **Mid Plane** option is used to create a feature by extruding the sketch equally in both the directions of the plane on which the sketch is drawn. If the total depth of the extruded feature is 30 mm, it will be extruded 15 mm toward the front and 15 mm toward the back of the sketching plane.

4. Set the value of the **Depth** spinner to 20 mm.

As it is evident from Figure 5-45 that the model is a thin extrude feature, therefore you need to invoke the **Thin Feature** rollout of the PropertyManager to add the thickness inward direction in the sketch.

5. Select the check box available on the title bar of the **Thin Feature** rollout; the **Thin Feature** rollout is invoked and the preview of the extrude feature is changed to thin extrude feature using the default values.

The thin extruded features can be created using a closed or an open sketch. If the sketch is a closed sketch, the thickness will be specified inside or outside the sketch to create a cavity inside the feature. However, if the sketch is an open sketch, the thickness will be specified below or above the sketch. Also, if the sketch to be extruded is an open sketch, the **Thin Feature** rollout will be invoked automatically on invoking the **Extrude PropertyManager**.

6. Make sure that the **One-Direction** option is selected in the **Type** drop-down list of the **Thin Feature** rollout.

The options provided in the **Type** drop-down list are used to specify the method used for defining the thickness of the thin feature. The **One-Direction** option of this drop-down list is used to add the thickness to one side of the sketch. The **Mid-Plane** option is used to add the thickness equally on both sides of the sketch and the **Two-Direction** option is used to create a thin feature by adding different thicknesses on both sides of a sketch.

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7. Set the value of thickness in the **Thickness** spinner to 3; the preview of the feature is modified and the thickness of 3 mm is added in the outward direction.

   Now, you need to reverse the direction of thickness toward the inward direction.

8. Choose the **Reverse Direction** button available on the left of the **Type** drop-down list to reverse the direction of thickness.



9. Choose the **OK** button from the **Boss-Extrude PropertyManager**; the thin extruded feature is created, refer to Figure 5-48. Next, change the orientation of the model to an isometric view by choosing the **Isometric** button from the **View Orientation** flyout. The isometric view of the model after adding thickness and extrusion depth is shown in Figure 5-48.

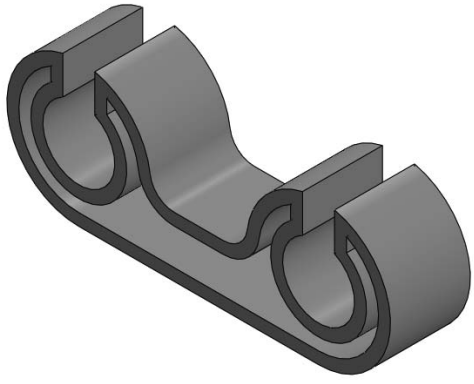


Figure 5-48 *Model after adding thickness and extrusion depth*

Adding Color to the Model

As mentioned in the tutorial description that after completing the model, you need to add color of your choice to it.

1. Choose **Edit > Appearance > Appearance** from the SolidWorks menus; the **color PropertyManager** is displayed on the left side of the drawing area, as shown in Figure 5-49. Ensure that the **Select Part** button is chosen and the name of the part is displayed in the **Selected Entities** area of the **Selected Geometry** rollout in the PropertyManager.

   Note that, as soon as the **color PropertyManager** is invoked, the **Appearance, Scenes, and Decals** task pane is also displayed on the right-side of the drawing area.

In the **color PropertyManager**, the **Selected Geometry** rollout has five buttons on its left, namely **Select Part**, **Select Faces**, **Select Surfaces**, **Select Bodies**, and **Select Features**. These buttons are used as filters for making a selection to assign the color to a model. For example, if you want to assign a color to the face of the model, clear all existing selections from the **Selected Entities** area and then choose the **Select Faces** button from the **Selected Geometry** rollout. This allows you to select only faces of the model.

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2. Select a color from the **Color** rollout of the PropertyManager; the selected color is displayed in the **Color** display area of the **Color** rollout. You can also set the required color using the **Pick to Color** display area.

   Alternatively, you can use the **Red Component of Color**, **Green Component of Color**, and **Blue Component of Color** spinners to set the color. The color selected in this rollout will be applied to the selected part, features, faces, or bodies.

3. Choose the **OK** button from the PropertyManager; the selected color is applied to the model. Figure 5-50 shows the final model.



Figure 5-49 *The* ***color PropertyManager***

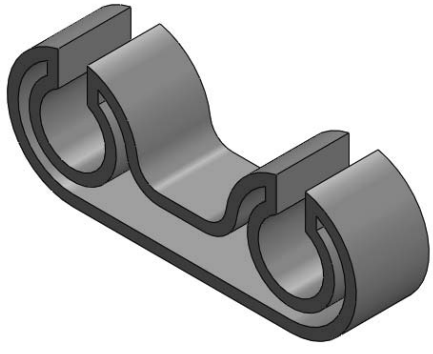


Figure 5-50 *The final model for Tutorial 3*

Saving the Model

Now, you need to save the model in the *c05* folder.

1. Save the model in the *c05* folder with the name *c05_tut03*. The model is saved at the location *\Documents\SolidWorks Tutorials\c05*

2. Choose **File > Close** from the SolidWorks menus to close the document.

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SELF-EVALUATION TEST

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

1. In SolidWorks, a sketch is revolved using the **Boss-Extrude PropertyManager**. (T/F)

2. You can also specify the depth of extrusion dynamically in the preview of the extruded feature. (T/F)

3. In SolidWorks, the right-hand thumb rule is followed for determining the direction of revolution. (T/F)

4. You can invoke the drawing display tools such as **Zoom to Fit** even if the preview of a model is displayed on the screen. (T/F)

5. The _____ tool enables you to determine the mass properties of the part or assembly that is available in the current session.

6. The _____ button of the **Boss-Extrude PropertyManager** is used to taper the feature.

7. Which of the following tools is used to apply the horizontal ordinate dimensions?

   (a) **Smart Dimension** (b) **Horizontal Ordinate Dimension**
   (c) **Horizontal Dimension** (d) **Hor Ordinate Dimension**

8. The _____ check box is used to create a feature with different values in both directions of the sketching plane.

9. The thin extruded feature can be created using a _____ or an _____ sketch.

10. The _____ tool is used to reverse the direction of material.

REVIEW QUESTIONS

Answer the following questions:

1. When you invoke the **Extruded Boss/Base** tool or the **Revolved Boss/Base** tool, the view is automatically changed to a _____.

2. The thin extruded features are created by invoking the _____ rollout of the PropertyManager.

3. The **Section Properties** tool is available in the _____ **CommandManager**.

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4. The _____ option is used to define the termination of a feature on a virtual surface created at an offset distance from the selected surface.

5. In SolidWorks, you can change the appearance of a model by assigning color to it. (T/F)

6. In SolidWorks, you can display the drawing area in multiple viewports. (T/F)

7. If you clear the **Draft outward** check box of the **Boss-Extrude PropertyManager**; the feature will be tapered outward. (T/F)

8. Which of the following tools is used to determine the section properties of the sketch?

(a) **Properties** (b) **Sec Properties**
(c) **Section Properties** (d) None

9. In which of the following features can you convert an open sketch?

(a) Thin feature (b) Solid feature
(c) Both (d) None of these

10. Which of the following tools is used to make a sketch fully defined in SolidWorks?

(a) **Fully Define Sketch** (b) **Smart Dimension**
(c) Both (d) None of these

EXERCISES

Exercise 1

Create the model shown in Figure 5-51. The dimensions of the model are shown in Figure 5-52. The extrusion depth of the model is 35 mm. **(Expected time: 30 min)**

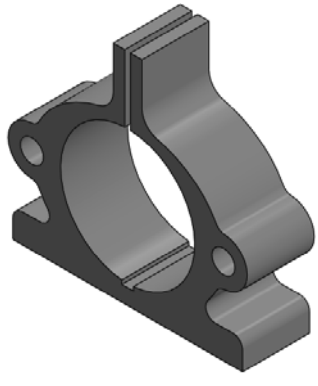


Figure 5-51 *Model for Exercise 1*

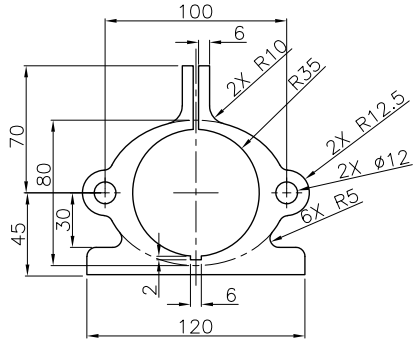


Figure 5-52 *Dimensions of the model for Exercise 1*

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Exercise 2

Create the model shown in Figure 5-53. The dimensions of the model are shown in Figure 5-54. The angle of revolution about the axis is 180 degrees. **(Expected time: 25 min)**

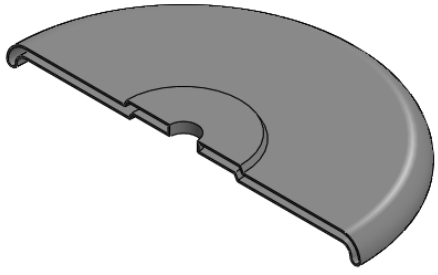


Figure 5-53 *Model for Exercise 2*

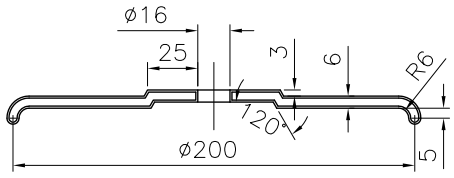


Figure 5-54 *Dimensions of the model for Exercise 2*

Exercise 3

Create the model shown in Figure 5-55. The dimensions of the model are shown in Figure 5-56. After creating the model, apply the copper material to it. Also, determine the mass properties of the model. **(Expected time: 35 min)**

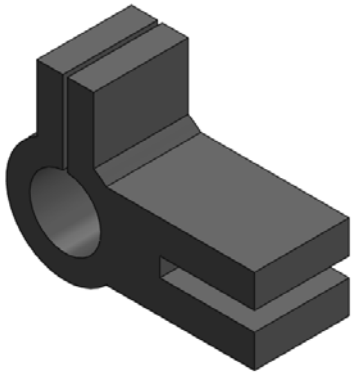


Figure 5-55 *Model for Exercise 3*

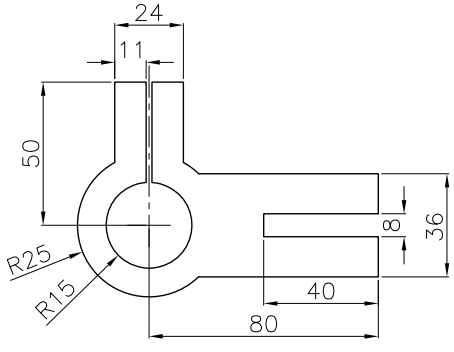


Figure 5-56 *Dimensions of the model for Exercise 3*

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Exercise 4

Create the model shown in Figure 5-57. The dimensions of the model are shown in Figure 5-58. **(Expected time: 30 min)**



Figure 5-57 *Model for Exercise 4*

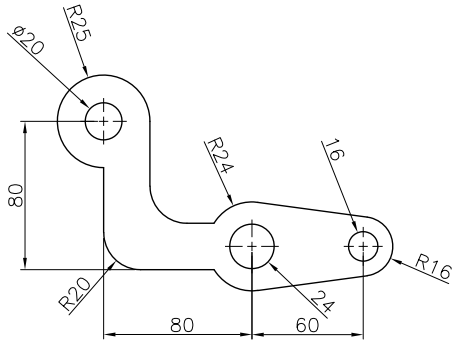


Figure 5-58 *Dimensions of the model for Exercise 4*

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Answers to Self-Evaluation Test

1. F, **2.** T, **3.** T, **4.** T, **5. Mass Properties**, **6. Draft On/Off**, **7. Horizontal Ordinate Dimension**, **8. Direction 2**, **9.** closed, open, **10. Reverse Direction**