



Chapter 5

Working with Additional Reference Planes

Learning Objectives

After completing this chapter, you will be able to:

- Understand the use of reference geometries.
- Create reference planes.
- Control the display of reference axes.
- Create new coordinate systems.
- Use additional termination options to create protrusion features.
- Create protruded and revolved cutouts.
- Include edges of the existing features as sketched entities in the current sketch.
- Work with advanced drawing display tools.

ADDITIONAL SKETCHING AND REFERENCE PLANES

As mentioned earlier, most mechanical designs consist of a number of sketched, reference, and placed features that are integrated together. In the previous chapter, you learned to create the base feature, which is the first feature in a model. After creating the base feature, you need to add more features to it. Most of the times, the additional features are not created on the default plane on which the base feature is created. For example, refer to the model shown in Figure 5-1.

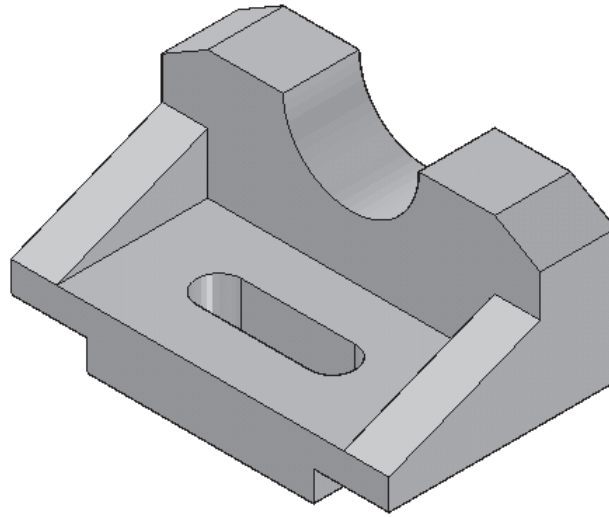


Figure 5-1 Model with multiple features

The base feature for this model is a protrusion feature, as shown in Figure 5-2. The sketch for this protrusion feature is drawn on the front plane.

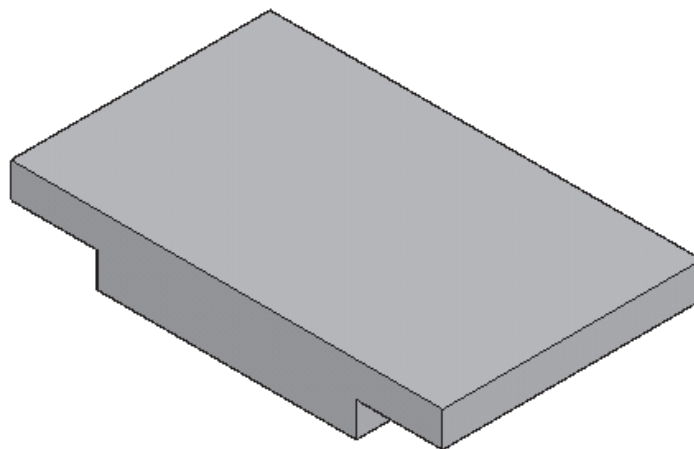


Figure 5-2 Base feature of the model

After creating the base feature, you need to create another protrusion feature, two rib features, and a cutout feature, see Figure 5-3. All these features are sketched features and require additional sketching or reference planes to draw their sketches.

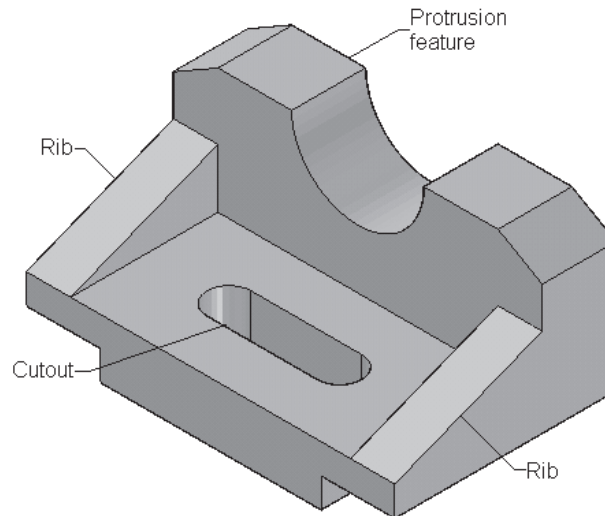


Figure 5-3 Various features in the model

This is the reason you require additional sketching planes or reference planes. By default, the top, right, and front planes are available in the part file. These default planes are called the base reference planes. You can use any of these reference planes or the planar faces of the base feature to draw the sketches of the additional features.

However, you cannot use the existing planes for drawing the sketch of a feature that is at a certain offset or angle from an existing reference plane or planar face of a feature. In this case, you need to create additional reference planes.

In addition to the base reference planes, you can create two more types of reference planes, which are discussed next.

Local Reference Planes

Local reference planes are the ones that are created while defining a feature. For example, whenever you invoke a sketched feature creation tool, the ribbon bar provides the tools to create reference planes. A reference plane created at this stage will be used to create only this particular feature. This is why these types of reference planes are called local reference planes. These planes are not displayed in the drawing window or in the **EdgeBar**.

Global Reference Planes

Global reference planes are the ones that are created separately as a feature using the tools available in the **Features** toolbar. These planes are displayed in the drawing window and in the **EdgeBar** and can be used to create multiple features.

CREATING REFERENCE PLANES

In Solid Edge, you are provided with seven options for creating global or local reference planes. If you select the option to create a reference plane from the **Create-From Options** drop-down list available in the ribbon bar in the **Plane or Sketch** step while creating a sketched feature, the result will be a local reference plane. But if you use the tools in the **Features** toolbar to create these features, the result will be a global reference plane. The seven methods of creating the reference planes are discussed next.

Creating a Coincident Plane

Toolbar: Features > Coincident Plane

Ribbon Bar: Plane or Sketch Step > Create-From Options > Coincident Plane



The **Coincident Plane** tool is used to create a reference plane that is coincident to a base reference plane, another reference plane, or a planar face of the model. While creating this plane, you can control the direction of the X axis of the plane.

To create a coincident plane, invoke the **Coincident Plane** tool; you will be prompted to click on a planar face or reference plane. Move the cursor over the reference plane or planar face on which you want to create the coincident plane; the selected plane or planar face will be highlighted and the edge that will define the X axis of the new plane will be shown in yellow. Also, the preview of the reference plane created using the currently highlighted items will be displayed in the drawing window.

You can select the other edges of the planar face or reference plane to define the X axis of the new plane by pressing the N or B key. Pressing the N key highlights the next edge to define the X axis of the new plane and pressing the B key highlights the previous edge to define the X axis. Figures 5-4 and 5-5 show the coincident planes being created on the same plane, but with different edges defining the X axis of the plane.

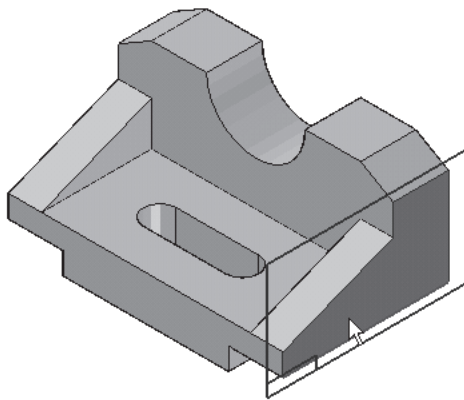


Figure 5-4 Creating a coincident plane with the bottom edge defining the orientation of the X axis

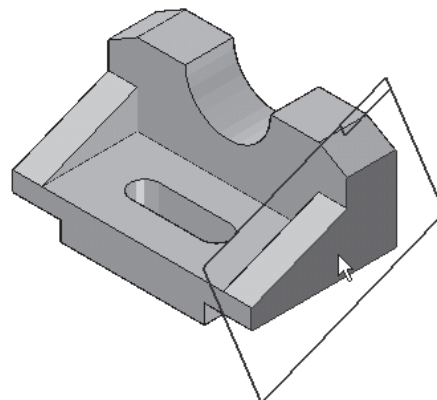


Figure 5-5 Creating a coincident plane with the inclined edge defining the orientation of the X axis

You can also toggle the positive X axis direction of the new plane by pressing the T key. For

example, if the reference plane is created, as shown in Figure 5-5, the positive X axis direction of this plane will be from the top endpoint to the bottom endpoint of the inclined edge. This is indicated by the small rectangle that is displayed at the origin of the plane. To reverse the direction of the positive X axis, press the T key; the direction of the positive X axis, which was pointing toward the left (Figure 5-5) will be reversed and will point toward the right, see Figure 5-6. The rectangle at the origin of the plane clarifies the direction of the positive X axis.

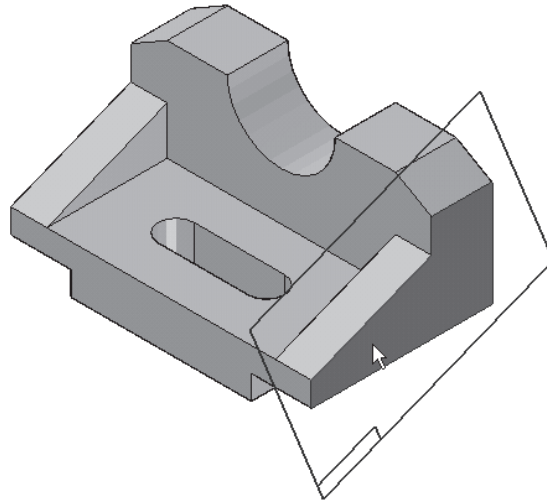


Figure 5-6 Preview of the reference plane after reversing the positive X axis direction

After you have selected all the options to create a reference plane, its preview will be displayed in the drawing window. Click at this stage to create the plane.



Tip. If the plane that you select to define the coincident plane does not have a linear edge, the X axis direction is defined using the base reference plane. However, you can use the N key to change the X axis direction of the new plane.

Creating a Parallel Plane

Toolbar: Features > Coincident Plane > Parallel Plane

Ribbon Bar: Plane or Sketch Step > Create-From Options > Parallel Plane



The **Parallel Plane** tool can be used to create a reference plane parallel to a selected base reference plane, another reference plane, or a planar face. The process of creating this plane works in two steps. In the first step, you need to select the reference plane or planar face to which the new plane will be parallel and also define the orientation of the positive X axis of the new reference plane. This step is the same as creating the coincident plane.

The second step is to define the location of the parallel plane. The location can be defined by entering the distance value or by using the keypoints such as endpoint, midpoint, center point, tangent point, and so on in the model.

To create a parallel plane, invoke the **Parallel Plane** tool. You will be prompted to click on a planar face or a reference plane. Using the procedure similar to creating a coincident plane, define the orientation of the new plane and then click. As soon as you click, the ribbon bar will be displayed with the option to define the location of the plane and you will be prompted to set the distance or key in the value.

You can enter the offset value directly in the **Distance** edit box or use the keypoints in the model to define the location of the parallel plane. To use the keypoints in the model, choose the **Keypoints** button in the ribbon bar to display the flyout that has the keypoint options such as endpoint, midpoint, center point, tangent point, and so on. Choose the required option from this flyout and then select that keypoint in the model. The new parallel plane will be placed at the selected keypoint. Figure 5-7 shows a parallel plane being created using the center keypoint from the model and Figure 5-8 shows a parallel plane being created using the tangent keypoint from the model.

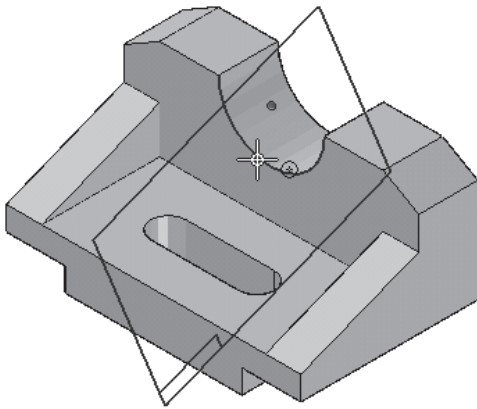


Figure 5-7 Using the center point in the model to define the location of the parallel plane

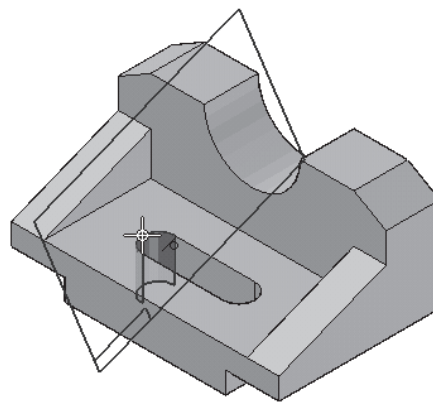


Figure 5-8 Using the tangent point in the model to define the location of the parallel plane

Creating an Angled Plane

Toolbar: Features > Coincident Plane > Angled Plane
Ribbon Bar: Plane or Sketch Step > Create-From Options > Angled Plane



The **Angled Plane** tool is used to create a reference plane that is at an angle to a selected plane and also passes through a specified edge, axis, or plane. On invoking this tool, you will be prompted to click on a planar face or a reference plane. The new reference plane will be defined at an angle to the selected plane. After selecting the plane, you will be prompted to click on the face, edge, or plane to be the base of the profile plane. This will define the edge or the plane through which the new plane will pass. You may need to use the **QuickPick** tool to select the required edge or plane.

Next, you need to define the direction of the positive X axis of the new plane. You will be prompted to click near the end of the axis for the reference plane orientation. As you move the cursor in the drawing window, the direction of the X axis will toggle. Click to define the positive X axis direction of the new plane.

Finally, you will be prompted to click to set the angle or key in a value. You can enter the angle value directly in the **Angle** edit box in the ribbon bar or use the keypoints to define the angle of the plane. Note that in this case, you can use only the endpoint, midpoint, or center point to define the plane. Figure 5-9 shows various selections to create an angled plane and also the preview of the resulting plane.

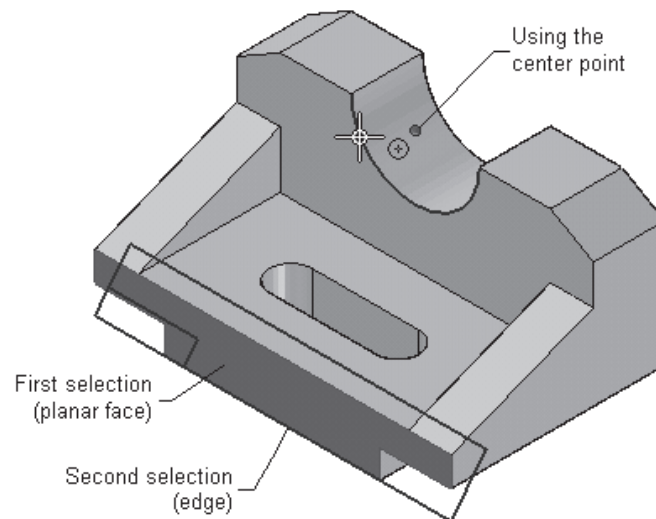


Figure 5-9 Various selections to create angled reference plane

Creating a Perpendicular Plane

Toolbar: Features > Coincident Plane > Perpendicular Plane

Ribbon Bar: Plane or Sketch Step > Create-From Options > Perpendicular Plane



The **Perpendicular Plane** tool is used to create a reference plane that is normal to a selected plane and passes through a specified edge or axis. To create a perpendicular plane, invoke this tool; you will be prompted to click on a planar face or a reference plane. The new reference plane will be defined normal to the plane that you select. After selecting the plane, you will be prompted to click on the face, edge, or plane to be the base of the profile plane. This will define the edge or the plane through which the new plane will pass. You may need to use the **QuickPick** tool to select the required edge or plane.

Next, you need to define the direction of the positive X axis of the new plane. You will be prompted to click near the end of the axis for the reference plane orientation. As you move the cursor in the drawing window, the direction of the X axis will toggle. Click to define the positive X axis direction of the new plane.

Finally, you will be prompted to click to set the angle or key in the value. You can move the cursor in the drawing window to define the side of the plane. Figure 5-10 shows various selections to create a perpendicular plane and the preview of the resulting plane.

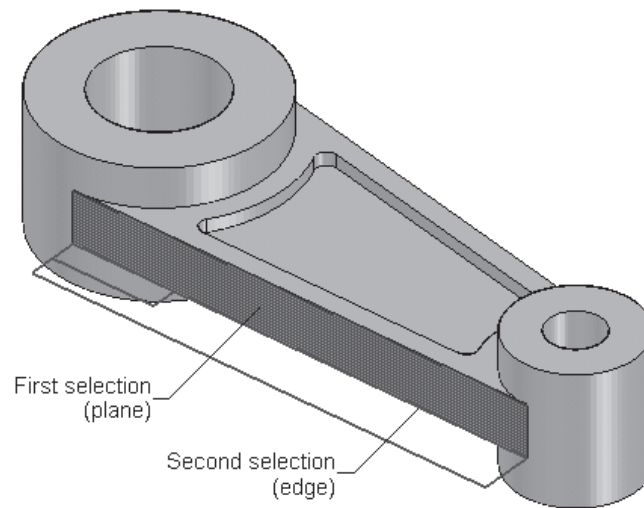


Figure 5-10 Various selections to create a perpendicular reference plane



Tip. You can also define any other angle in the **Angle** edit box while creating the perpendicular plane.

Creating a Coincident Plane by Axis

Toolbar: Features > Coincident Plane > Coincident Plane By Axis
Ribbon Bar: Plane or Sketch Step > Create-From Options > Coincident Plane By Axis



The **Coincident Plane By Axis** tool works in the same way as the **Coincident Plane** tool. The only difference is that in this tool, after selecting the plane, you will also be prompted to select the edge to define the direction of the positive X axis and the orientation of the plane. Unlike in the **Coincident Plane** tool, where you used the keyboard shortcuts to define these parameters, in this tool you need to select these parameters in the drawing window.

Creating a Plane Normal to an Edge or a Sketched Curve

Toolbar: Features > Coincident Plane > Plane Normal to Curve
Ribbon Bar: Plane or Sketch Step > Create-From Options > Plane Normal to Curve



The **Plane Normal to Curve** tool is used to create a plane that is normal to a selected sketched curve or an edge of the model. You can define a point along the curve where the normal plane needs to be placed. To create a plane normal to a curve or an edge of the model, invoke this tool; you will be prompted to click on the curve or edge for the normal plane. Once you move the cursor close to the sketched curve or edge of the model, the nearest endpoint of the curve or the edge will be highlighted. At this point, select the curve. The preview of the normal plane will be displayed and you will be prompted to click on a keypoint

or key in the offset distance. Also, the **Position** and **Distance** edit boxes will be displayed in the ribbon bar. These edit boxes are used to specify the location of the plane along the curve. The **Position** edit box defines the position in terms of percentage. The total length of the edge or the curve is taken as 1 and you are allowed to enter any value between 0 and 1. Similarly, the **Distance** edit box defines the distance of the normal plane from the endpoint that is highlighted while selecting the edge or the curve.

You can also define the location of the normal plane by specifying a point along the edge or the curve in the drawing window. Figure 5-11 shows a plane being created normal to a sketched curve.

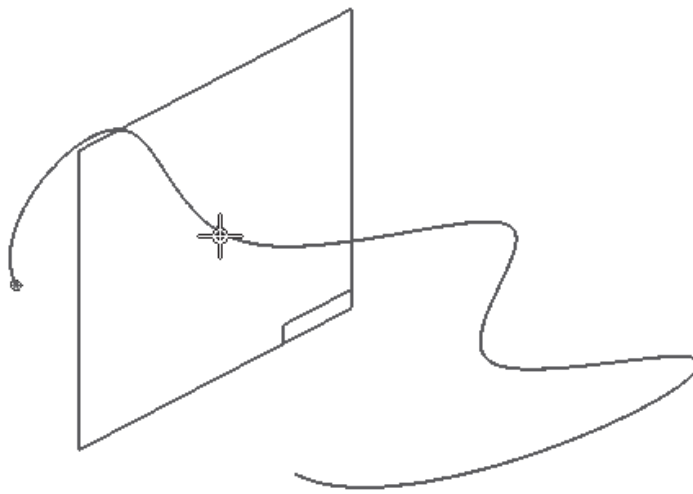


Figure 5-11 Creating a plane normal to a sketched curve

Creating a Plane Using Three Points

Toolbar: Features > Coincident Plane > Plane by 3 Points

Ribbon Bar: Plane or Sketch Step > Create-From Options > Plane by 3 Points



The **Plane by 3 Points** tool is used to create a reference plane by selecting three points. These three points define the origin of the plane, the direction of the positive X axis, and the direction of the positive Y axis. The length and width of the plane are determined by the distance of the second and third points from the origin.

To define a reference plane using three points, invoke the **Plane by 3 Points** tool; you will be prompted to click on a point to define the origin of the plane. Select a point in the model that you want to use as the origin of the new plane. You can use the **Keypoints** flyout from the ribbon bar to select the point. Next, you will be prompted to click on a point to define the base of the plane. Select a point that will define the direction of the positive X axis. Also, note that this point will define the length of the plane. Finally, you will be prompted to click on a point to complete the plane. Select a point that will define the direction of the positive Y axis and the width of the plane.

Figure 5-12 shows a plane being created using three points.

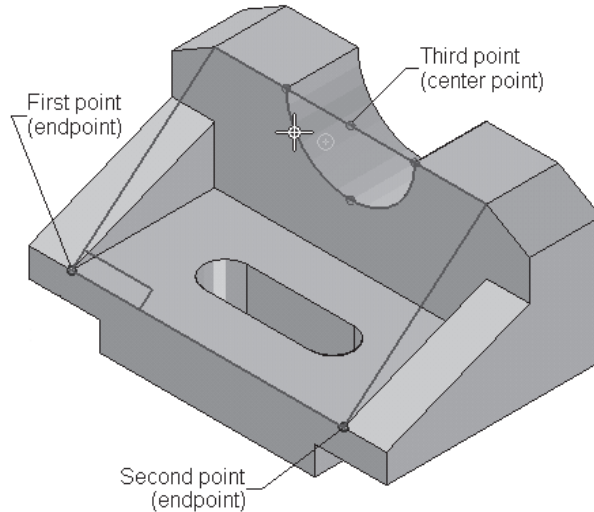


Figure 5-12 Creating a plane using three points



Note

While selecting the option to create reference planes from the **Create-From Options** drop-down list in the ribbon bar, the **Feature's Plane** and **Last Plane** options are also available. The **Feature's Plane** option uses the plane on which the profile of the selected feature was created and the **Last Plane** option uses the previous plane selected to create the feature.

Displaying the Reference Axes

Solid Edge automatically creates reference axes when you create a revolved feature, hole feature, or any other circular or semicircular feature. However, the display of these reference axes is turned off by default. To turn on the display of the axes, choose **Tools > Show All > Reference Axes**. The reference axes of all the revolved features will be displayed in the drawing window now. But the reference axes of the circular, semicircular, or hole features will not be displayed. To display the axes of these features, choose **Tools > Show All > Toggle Axes**. You will be prompted to click on a feature or surface containing the reference axis. Select the features whose reference axes you want to display. The reference axes of the selected features will be displayed.

Similarly, you can turn off the display of the reference axes by choosing **Tools > Hide All > Reference Axes**.



Note

Once you have toggled the display of the reference axes using the **Toggle Axes** option, you can use the **Tools > Hide All > Reference Axes** or **Tools > Show All > Reference Axes** options to turn off or on the display of the reference axes.

You can use the options in the **Show All** or **Hide All** cascading menu to modify the display of the other entities such as reference planes.

UNDERSTANDING COORDINATE SYSTEMS

Every part file that you start in Solid Edge has a coordinate system defined in it. This default coordinate system is called the base coordinate system. In Solid Edge, you can create additional coordinate systems according to your design requirements. These coordinate systems can be used as a reference to create reference planes, measure distances, copy parts, and so on.

Creating a Coordinate System

Toolbar: Features > Coordinate System



In Solid Edge, you can create coordinate systems using two options: by defining the orientations of any of the two axes using the edit boxes or by defining the orientation of any of the two axes by selecting the edges in the model. The options required to create a coordinate system can be selected from the **Coordinate System Options** dialog box (Figure 5-13), which is displayed when you invoke this tool.

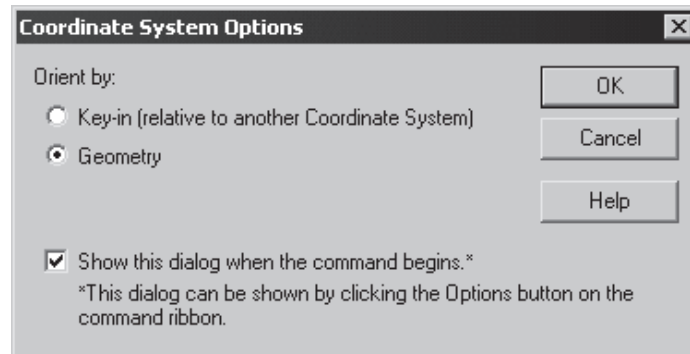


Figure 5-13 The Coordinate System Options dialog box

Select the required option from this dialog box and choose **OK**; the **Coordinate System** ribbon bar will be displayed. If you select the **Key-in (relative to another Coordinate System)** option, the ribbon bar will have two steps. If you select the **Geometry** option, this ribbon bar will have three steps for creating the coordinate system. All these steps are discussed next.

Origin Step

This step is common to both the options of creating a coordinate system and is active by default when the **Coordinate System** ribbon bar is displayed. In this step, you need to define the point where the origin of the coordinate system will be placed. To define the origin, you can select a keypoint in the model or enter the coordinates of the point in the **X**, **Y**, and **Z** edit boxes in the ribbon bar. By default, the points will be defined relative to the default coordinate system. This is because **Model Space** is selected from the **Relative to** drop-down list in the ribbon bar. However, if there are other coordinate systems in the current drawing, they will be listed in this drop-down list and you can define the coordinates of the points relative to them.

Orientation Step

This step is available only when you select the **Key-in (relative to another Coordinate System)** option. In this step, you need to define the orientation of the X, Y, and Z axes of the new coordinate system relative to those of the default coordinate system (model space) or any other coordinate system selected from the **Relative to** drop-down list. The coordinate system will be rotated about the axis by the angle that you define in the **X**, **Y**, and **Z** edit boxes. For example, if you enter **20** as the value in the **X°** edit box, then the new coordinate system will be rotated by 20-degrees about the X axis.



Note

Solid Edge uses the right-hand thumb rule to determine the direction of the rotation of the axes. The right-hand thumb rule states that if the thumb of the right hand points in the direction of the axes, then the direction of the curled fingers will point toward the direction of rotation.

Enter the rotation angles in the **X°**, **Y°**, and **Z°** edit boxes and choose the **Preview** button; the preview of the resulting coordinate system will be displayed. Choose the **Finish** button to create the coordinate system.

First Axis Step

This step is available only if you select the **Geometry** option from the **Coordinate System Options** dialog box and will be activated automatically as soon as you define the origin of the coordinate system. In this step, you need to define the orientation of the first axis of the coordinate system. By default, the **X-Axis** button is chosen from the ribbon bar. This is why the axis that you define will be taken as the X axis of the coordinate system. However, if you want to define any other axis first, you can choose the button of that axis from the ribbon bar.

You can select a point, linear element, planar face, or reference plane to define the axis. After selecting the element to define the first axis, choose the **Accept** button from the ribbon bar. You can also press ENTER or right-click to accept the selection. Next, you need to define the direction of the positive side. It is represented by an arrow along the entity that you selected to define the first axis. You can move the cursor to either side of the origin to define the direction of the positive side. Click on the desired side to accept the selection.

Second Axis Step

This step will be activated automatically as soon as you define the first axis of the coordinate system. In this step, you need to define the orientation of the second axis of the coordinate system. If you have defined the X axis of the coordinate system in the previous step, then you need to define the Y or Z axis in this step. You can choose the button of the axis that you want to define from the ribbon bar. Note that the button of the axis that is already defined is not enabled in the ribbon bar. The method of defining the axis is the same as that of the first axis.

As soon as you define the second axis, the coordinate system will be created and displayed in the model. Choose **Finish** from the ribbon bar to confirm the creation of the coordinate system. You can also choose the button of any of the steps to modify the selection made in that particular step. Figure 5-14 shows a coordinate system in the model.

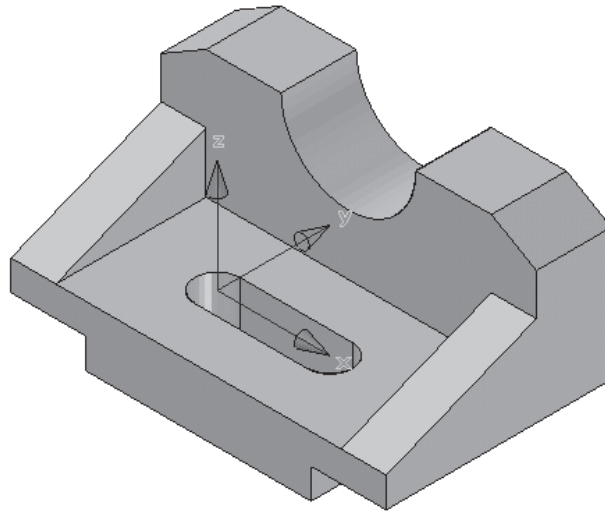


Figure 5-14 Coordinate system in the model

**Note**

Solid Edge uses the right-hand rule to determine the direction of the third axis. This rule states that if the thumb of the right hand points in the direction of the positive X axis and the first finger points in the direction of the positive Y axis, then the middle finger will point in the direction of the positive Z axis.

*The coordinate system is displayed as a feature in the **EdgeBar**.*

USING THE OTHER OPTIONS OF THE PROTRUSION TOOL

In the previous chapter, you learned about some options of the **Protrusion** tool. In this chapter, you will learn the remaining options of this tool.

Side Step



As mentioned in the previous chapter, the side step is required while creating additional features on the base feature using open sketches. In this step, you will be prompted to specify the side of the open sketch on which the material will be added. Figure 5-15 shows an open sketch and the side on which the material will be added and Figure 5-16 shows the model after creating the protrusion feature using the open sketch.

Extent Step



In the previous chapter, you learned about the symmetric and nonsymmetric finite extent options to create a feature. Some other options are also available in this step. These options are discussed next.

Through All



This option is used to create a feature by extruding the sketch through all the features available in the model. The feature will be terminated at the last face of the model with

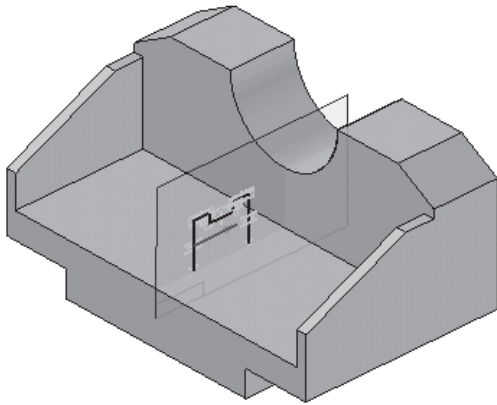


Figure 5-15 Open sketch and the side on which the material will be added

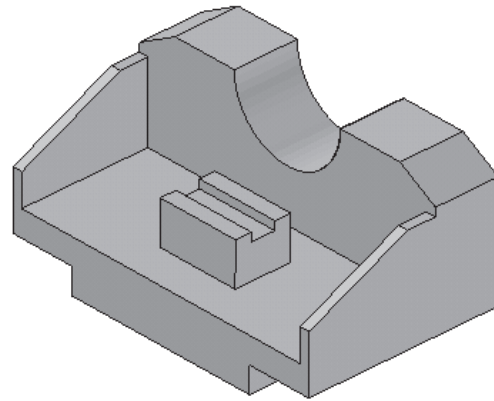


Figure 5-16 Resulting feature

which the sketch will intersect when extruded. On selecting this option, an arrow will be displayed on the sketch and you will be prompted to click to select the side. You can move the cursor on either side of the sketching plane to define the direction of extrusion. Note that if you select the side in which the sketch does not intersect with any face of the model, an error message will be displayed informing you that the operation was unsuccessful. Figure 5-17 shows a sketch extruded in the downward direction using the **Through All** option.

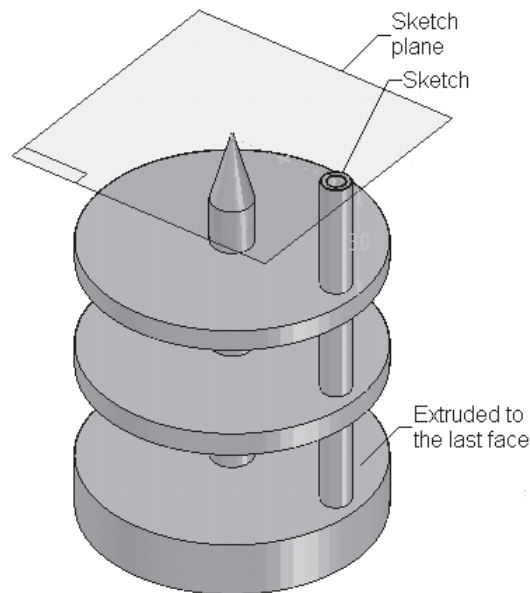


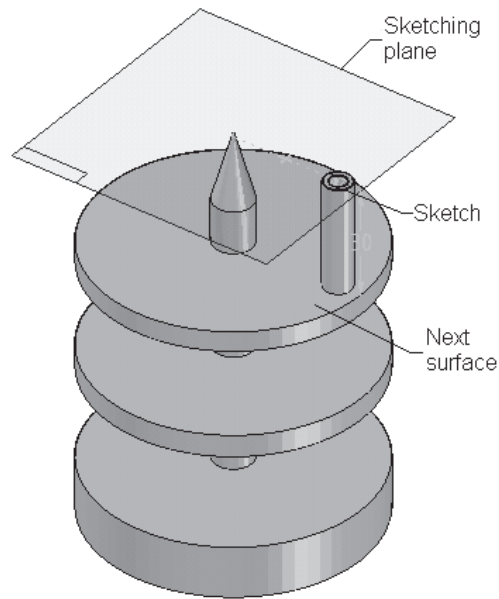
Figure 5-17 Extruding the sketch using the **Through All** option

Through Next



This option is used to create a feature by extruding the sketch up to the next surface of the model that the sketch will intersect when extruded. You will be prompted to define

the side of extrusion of the sketch. Figure 5-18 shows a sketch extruded in the downward direction up to the next surface.

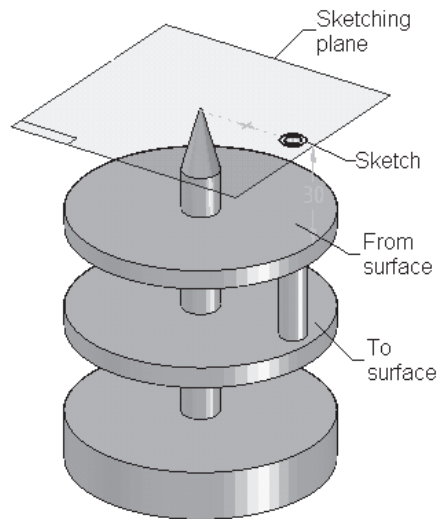


*Figure 5-18 Extruding the sketch using the **Through Next** option*

From/To Extent



This option is used to create a protrusion feature by extruding the sketch from a selected plane to another selected plane. When you select this option, the “**From**” Surface and “**To**” Surface buttons will be displayed in the ribbon bar and you will be prompted to specify the from and to surfaces. Figure 5-19 shows a sketch extruded using this option.



*Figure 5-19 Extruding the sketch using the **From/To Extent** option*

You can also define the offset values from the from and to surfaces by entering the value in the **Offset** edit box. For example, if you want to use this option with an offset, choose the **From/To Extent** button from the ribbon bar. You will be prompted to select the from surface. In the **Offset** edit box, enter the offset distance for the surface from which the feature should start and then select the surface. An arrow will be displayed and you will be prompted to select the side for the offset. Next, you will be prompted to select the surface up to which the feature will be created. Again, enter the offset distance and select the surface. An arrow will be displayed and you will be prompted to select a side for the offset. Move the cursor to define the direction of the offset. Figure 5-20 shows a protrusion feature created by defining the offset values for the from and to surfaces.

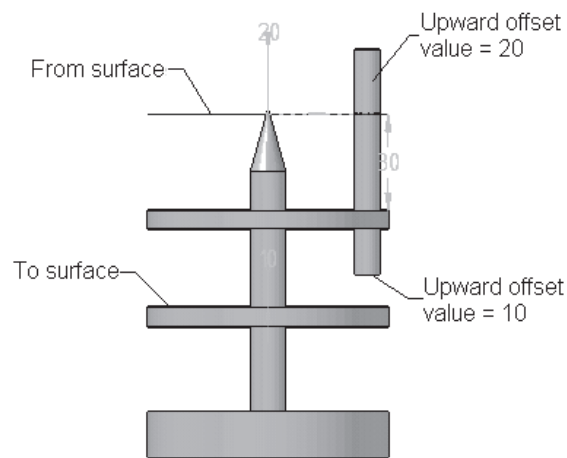


Figure 5-20 Creating a protrusion feature by defining offset values for the from and to surfaces

Note that you can also use the **Through All**, **Through Next**, and **From/To Extent** options to extrude the sketch on both sides of the sketching plane in combination with the **Non-symmetric Extent** option. Figure 5-21 shows a sketch extruded on both sides of the sketching plane using the **Through All** option in combination with the **Non-symmetric Extent** option.

Treatment Step



In this step, you can add a draft or crown to the protrusion feature. These features are added to aesthetically improve the design and for an easy removal of the model from its mold. By adding a draft, you can add a linear taper to the model, as shown in Figure 5-22. This figure shows a different draft applied to both sides of a nonsymmetric feature. By adding a crown, you can add a curved taper to the model, as shown in Figure 5-23. This figure shows different crowns applied to both sides of a nonsymmetric feature. Note that in both cases, the basic sketch is a circle.

To add a treatment to the feature, choose the **Treatment Step** button from the ribbon bar. The options available in the ribbon bar are discussed next.

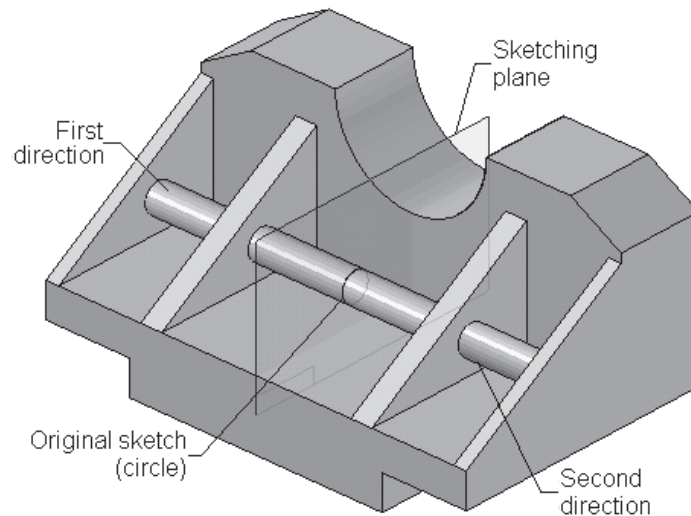


Figure 5-21 Sketch extruded using the **Through All** option on both sides of the sketching plane with the **Non-symmetric Extent** option

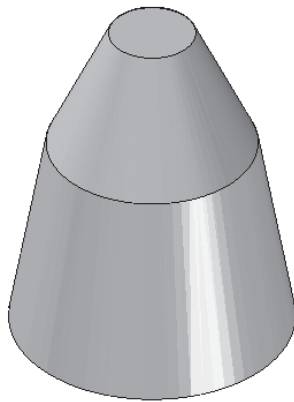


Figure 5-22 Different drafts applied to both sides of a protrusion feature

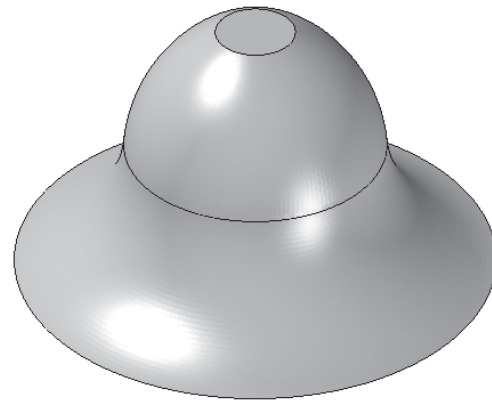


Figure 5-23 Different crowns applied to both sides of a protrusion feature

Treatment Options



This button is chosen to display the **Treatment Options** dialog box, as shown in Figure 5-24. By default, the **Treatment** step is not invoked automatically after you complete the **Extent** step. This is because the **Never prompt for treatment parameters** radio button is selected from this dialog box. You can select the **Always prompt for treatment parameters** radio button to invoke the **Treatment** step automatically after the completion of the **Extent** step.

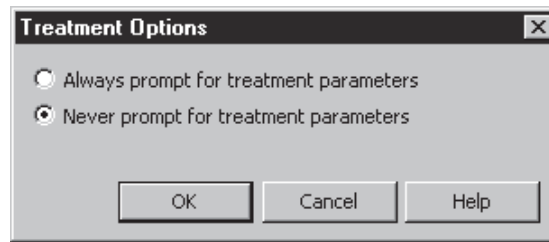


Figure 5-24 The *Treatment Options* dialog box

No Treatment



When you invoke the **Treatment** step, this button is chosen by default. As a result, no treatment is applied to the model.

Draft



The **Draft** button is chosen to add a draft to the protrusion feature. When you choose this button, the **Angle** edit box and the **Flip 1** button will be displayed in the ribbon bar. You can enter the draft angle in the **Angle** edit box. The default direction in which the draft will be created is displayed in the preview of the model. If you want to reverse the direction of the draft, choose the **Flip 1** button. If the draft was initially inward, the preview will now show the draft outward, indicating that the draft will be applied in the reverse direction now.

Crown



The **Crown** button is chosen to add a crown to the protrusion feature. When you choose this button, the **Crown Parameters** dialog box will be displayed. If you extrude the sketch symmetrically or nonsymmetrically to both sides of the sketching plane, the longer version of this dialog box will be displayed with the **Direction 1** and **Direction 2** areas, as shown in Figure 5-25. However, if you extrude the sketch in one direction, the shorter version of this dialog box will be displayed with only the **Direction 1** area. The options available in both areas of the **Crown Parameters** dialog box are the same and are discussed next.

Crown Type

The **Crown Type** drop-down list is used to select the technique of applying a crown to the feature. The options available in this drop-down list are discussed next.

No Crown

This option is used when you do not want to apply the crown in this direction.

Radius

This option is used to apply the crown by defining its radius. The radius value is specified in the **Radius** edit box, which is available below the **Crown Type** drop-down list.

Radius and take-off

This option is used to apply the crown by defining its radius and the takeoff angle.

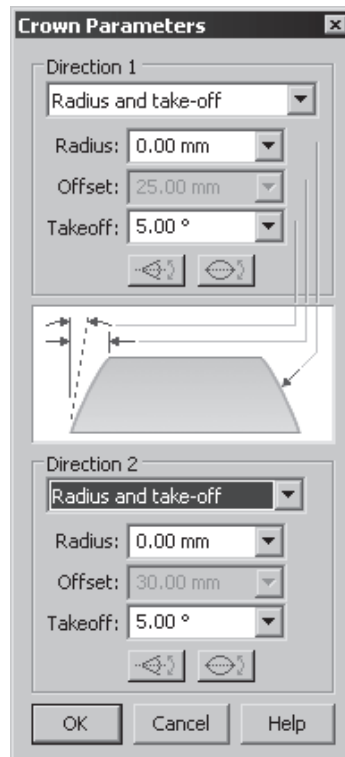


Figure 5-25 The **Crown Parameters** dialog box

These values are specified in the edit boxes that are available below the **Crown Type** drop-down list.

Offset

This option is used to apply the crown by defining the offset value between the sections at the start and the end of the crown. The offset value is specified in the **Offset** edit box available below the **Crown Type** drop-down list.

Offset and take-off

This option is used to apply the crown by defining the offset value between the sections at the start and the end of the crown and the takeoff angle. These values are specified in the edit boxes available below the **Crown Type** drop-down list.

Radius

The **Radius** edit box is used to specify the radius value of the crown and will be available only when you select the **Radius** or the **Radius and take-off** crown type.

Offset

The **Offset** edit box is used to specify the offset value of the crown and will be available only when you select the **Offset** or the **Offset and take-off** crown type.

Takeoff

The **Takeoff** edit box is used to specify the takeoff value of the crown and will be available only when you select the **Radius and take-off** or the **Offset and take-off** crown type.

Flip Side

This button is used to reverse the side on which the crown is applied. If the crown is applied inside the feature, choosing this button will apply the crown outside the feature.

Flip Curvature

The **Flip Curvature** button is used to reverse the curvature of the crown.

Preview Window

The **Preview** window displays the preview of various crown parameters that you define using the **Crown Parameters** dialog box.

Figures 5-26 and 5-28 show the preview of the crown feature and Figures 5-27 and 5-29 show the resulting crown feature created using the offset and take-off crown types, respectively.

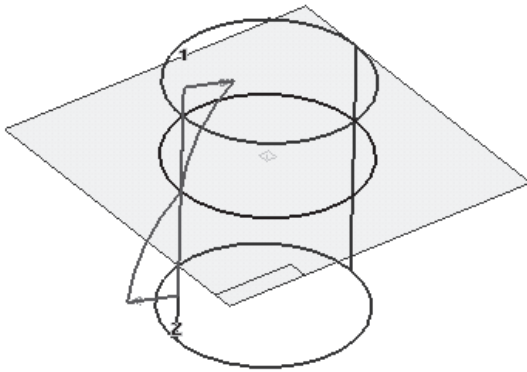


Figure 5-26 Preview of the crown showing the side of the crown

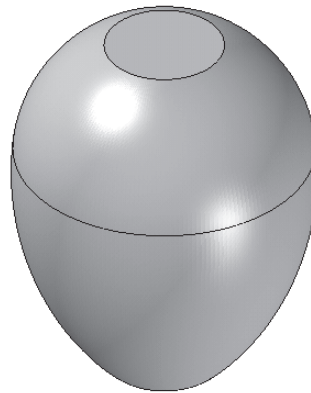


Figure 5-27 Feature with the resulting crown

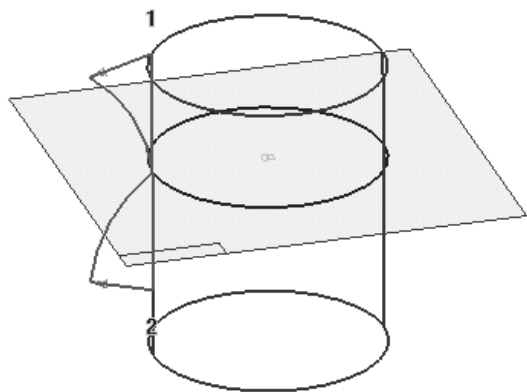


Figure 5-28 Preview of the crown showing the side of the crown

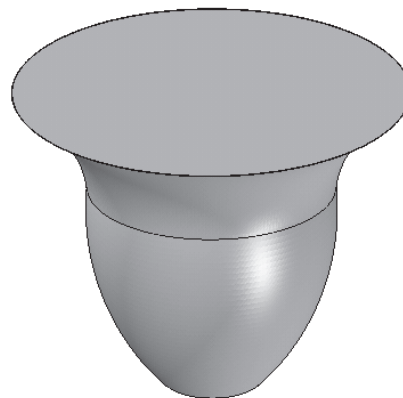


Figure 5-29 Feature with the resulting crown

Crown Parameters

The **Crown Parameters** button is chosen to redisplay the **Crown Parameters** dialog box for modifying the crown parameters.

CREATING CUTOUT FEATURES

Cutouts are created by removing the material, defined by a profile, from one or more existing features. In Solid Edge, you can create various types of cutouts such as extruded cutouts, revolved cutouts, swept cutouts, and so on. In this chapter, you will learn about the extruded or revolved cutouts. The remaining types of cutouts will be discussed in later chapters.

Creating Extruded Cutouts

Toolbar: Features > Cutout



Extruded cutouts are created by extruding a profile to remove the material from one or more features. Figure 5-30 shows the base feature and the sketch that will be used to create a cutout and Figure 5-31 shows the rotated view of the model after creating the cutout.

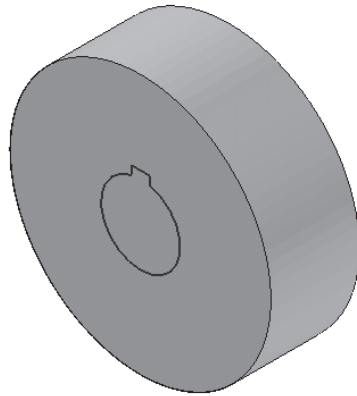


Figure 5-30 Base feature and the sketch for the cutout

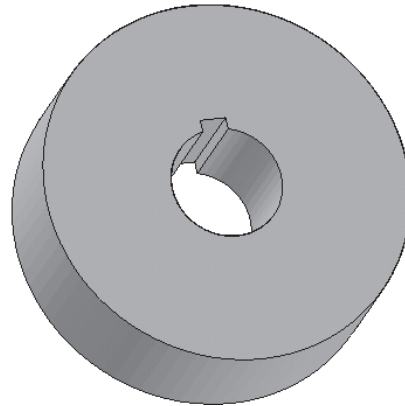


Figure 5-31 Rotated view of the model after creating the cutout

In Solid Edge, cutouts are created using the **Cutout** tool. This tool works in the same manner as the **Protrusion** tool. Note that while creating the cutouts, you can create open profiles and use the **Side** step extensively to define the direction of the material removal. Figures 5-32 through 5-35 show the side of the material removal and the resulting features that are created.

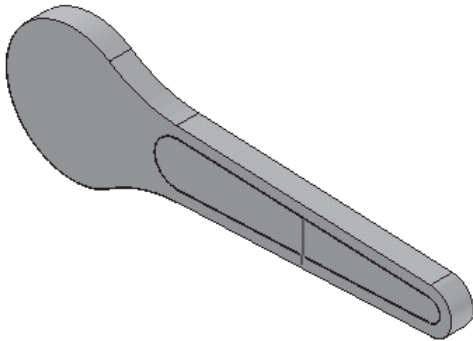


Figure 5-32 Sketch for the cutout and the direction for the cutout pointing inside the sketch

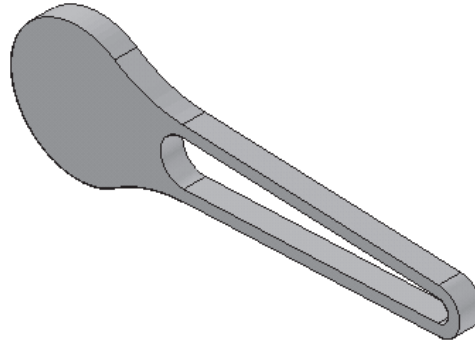


Figure 5-33 Resulting cutout created by removing the material inside the sketch

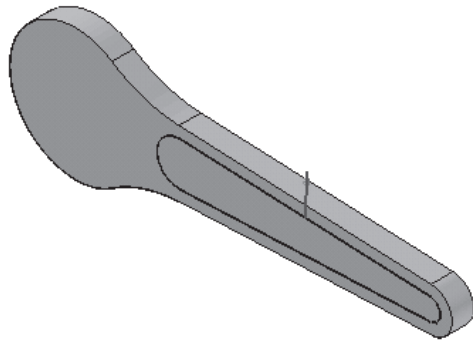


Figure 5-34 Sketch for the cutout and the direction for the cutout pointing outside the sketch

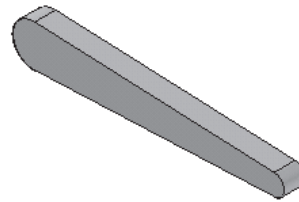


Figure 5-35 Resulting cutout created by removing the material outside the sketch

You can also use open profiles to create the cutouts. However, you need to carefully define the side of the material removal for the cutout, see Figures 5-36 through 5-39.

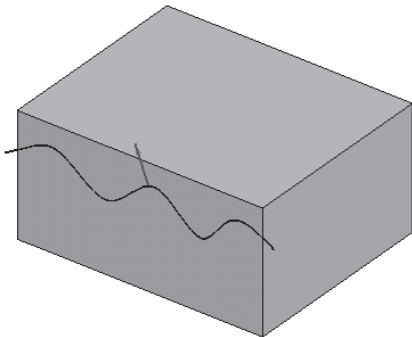


Figure 5-36 Open profile and the side of material removal

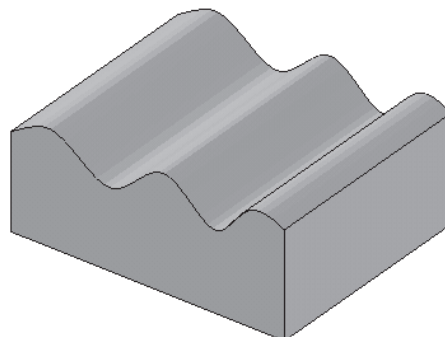


Figure 5-37 Resulting cutout created by removing the material above the open profile

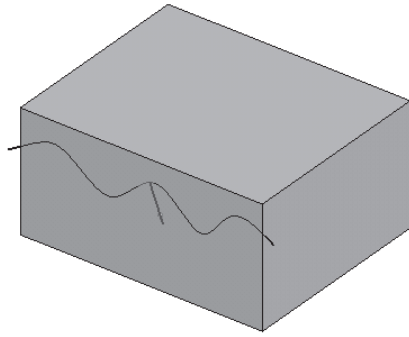


Figure 5-38 Open profile and the side of the material removal

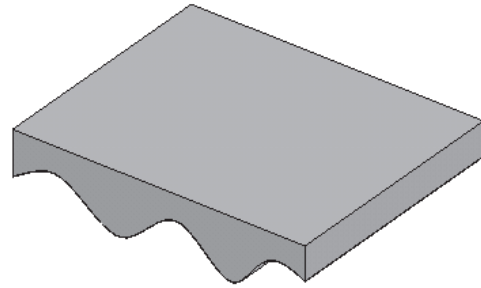


Figure 5-39 Resulting cutout created by removing the material below the open profile

Creating Revolved Cutouts

Toolbar: Features > Revolved Cutout



The **Revolved Cutout** tool is used to create a revolved cutout by removing the material defined by the sketch. This tool works in the same manner as the **Revolved Protrusion** tool, with the only difference being that this tool removes the material from an existing feature. Similar to the extruded cutouts, you can also use the **Side** step extensively in the revolved cutouts to specify the side of material removal. Figures 5-40 through 5-43 show the side of material removal and the resulting model after creating a symmetric semicircular revolved cutout.

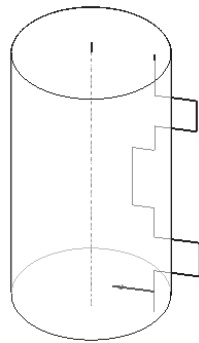


Figure 5-40 Open profile and the side of the material removal

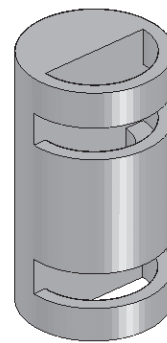


Figure 5-41 Resulting revolved cutout created by removing the material on the left of the profile

INCLUDING THE EDGES OF THE EXISTING FEATURES IN THE SKETCH

Toolbar: Draw > Include



Sometimes, while drawing the profiles of some features, you may need to use the edges of the existing features as sketched entities in the current profile. In Solid Edge, you can

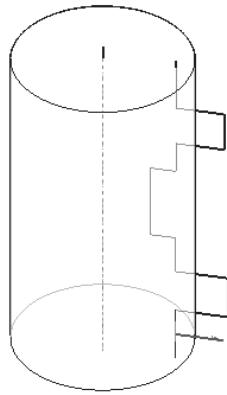


Figure 5-42 Open profile and the side of the material removal

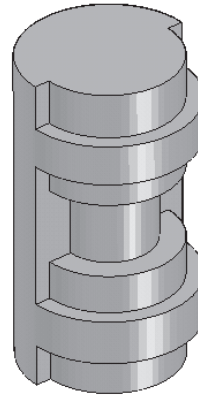


Figure 5-43 Resulting revolved cutout created by removing the material on the right of the profile

do this using the **Include** tool in the sketching environment. You can copy the edges of the existing features by projecting them exactly as they are on the current sketching plane or by copying them with some offset. Note that because you are projecting the entities that are already used in the model, you do not need to add dimensions to the projected entities. They will automatically take the dimensions from the original entities.

To project the geometries, invoke the sketching environment in the model with some existing features. Choose the **Include** button from the **Draw** toolbar to display the **Include Options** dialog box, as shown in Figure 5-44. The options available in this dialog box are discussed next.

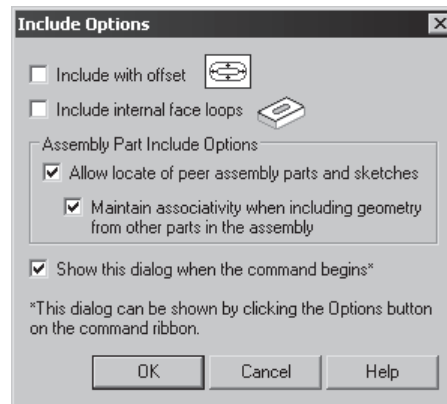


Figure 5-44 The **Include Options** dialog box

Include with offset

This check box is selected to project the geometries with some offset value. If this check box is selected, the offset options will be displayed in the ribbon bar after you select the edges.

Include internal face loops

If this check box is selected, the geometries of all the internal loops on a face will also be projected when you select a face to project the edges.

Assembly Part Include Options Area

The options available in this area are used in the assembly modeling environment. These options are discussed next.

Allow locate of peer assembly parts and sketches

If this check box is selected, you will be allowed to select the geometries from the other parts in the assembly.

Maintain associativity when including geometry from other parts in the assembly

If this check box is selected, the geometries that you project from the other parts in the assembly will be associative.

After setting the options in this dialog box, choose the **OK** button to display the **Include** ribbon bar. Depending on whether you have selected the option to include with an offset, this tool works in one or two steps, which are discussed next.

Select Step

This step allows you to select the geometries that you want to include in the profile. You can set the option for selecting the geometries using the **Select** drop-down list. Depending on the selection type selected, you may need to choose the **Accept** button after selecting the entities. If you have not selected the option to include the geometries with an offset, then exit this tool after selecting the entities. As mentioned earlier, you do not need to dimension the projected entities. The dimensions are adopted from the original geometries that are projected.

Offset Step

This step will be available only if you select the option of including the geometries with an offset. In this case, select the geometries in the **Select** step and then choose the **Accept** button from the ribbon bar. The **Offset** step will be invoked and the **Distance** edit box will be displayed in the ribbon bar. Enter the offset distance in this edit box and then move the cursor in the drawing window to specify the side for the offset. After selecting the side, click to project and offset the selected geometry.

Figure 5-45 shows a model after projecting some of the geometries from the existing features on to a reference plane created at an offset from the top face of the model.

ADVANCED DRAWING DISPLAY TOOLS

In earlier chapters, you learned about some of the basic drawing display tools. In this chapter, you will learn about the advanced drawing display tools, which are discussed next.

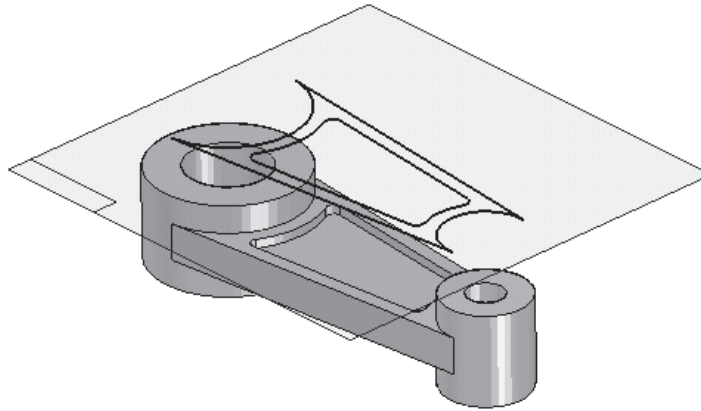


Figure 5-45 Geometries projected on a reference plane

Creating User-defined Named Views

As mentioned earlier, you can use the **Named Views** drop-down list in the **Main** toolbar to invoke the standard named views. In Solid Edge, you can also create user-defined standard views that are automatically added to the **Named Views** drop-down list. As a result, you can invoke these views whenever required.

To create the user-defined named views, set the current view to the view that you want to save using the **Rotate** tool or any other drawing display tool. After setting the view, choose **View > Named Views** from the menu bar. The **Views** dialog box will be displayed with six standard named views. Enter the name of the new user-defined view in the **Name** column of the seventh row. You can also enter the description about the view in the **Description** column. After entering the information, exit the dialog box. The view that you defined will be displayed in the **Named Views** drop-down list and can be selected from there.

USING COMMON VIEWS

Toolbar: Main > Rotate > Common Views



Solid Edge provides you with the **Common Views** tool, which is a very user-friendly tool to set the current view to some standard common views. You can also invoke this tool by right-clicking in the drawing window when the **Select** tool is active. From the shortcut menu, choose **Common Views**. The **Common Views** dialog box will be displayed with a cube in it. You can use the vertices or faces of this cube to rotate the view of the model. As you move the cursor over any vertex or face of the cube in the **Common Views** dialog box, you will be informed about how the view will be rotated by a message that is displayed in this dialog box. While working with the **Common Views** tool, you can press the HOME key at any point of time to invoke the standard isometric view.

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 5-46. Its dimensions are given in the views shown in Figure 5-47. After creating the model, save it with the name `|Solid Edge|c05|c05tut1.par`. (Expected time: 45 min)

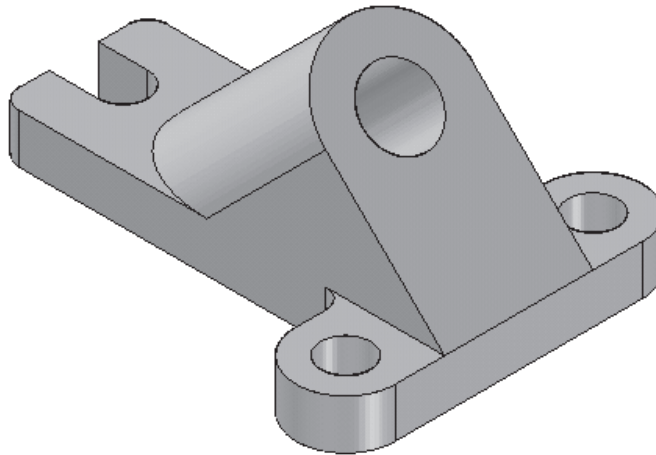


Figure 5-46 Model for Tutorial 1

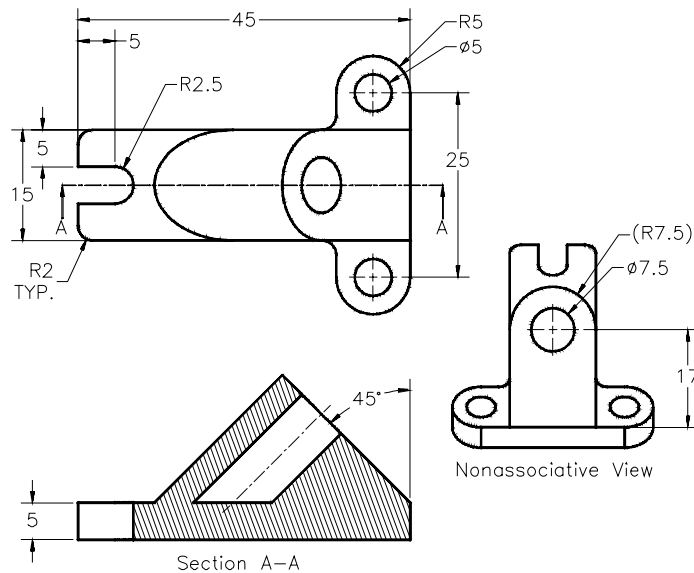


Figure 5-47 Dimensions to create the model

Whenever you start creating a model, you first need to determine the number of features in it and then the sequence in which they will be created. The model for Tutorial 1 is a combination of two protrusion features. The base feature will be created on the top plane. The second feature will be created at an angled plane created at 45-degrees.

The following steps outline the procedure to create the given model:

- a. Create the base feature with two holes on the top plane, refer to Figure 5-49.
- b. Define a new reference plane at an angle of 45-degrees to the right edge of the base feature and use it to draw the profile for the second feature, refer to Figure 5-52.
- c. Extrude the profile up to the next face to complete the feature, refer to Figure 5-53.

Creating the Base Feature

As mentioned earlier, the base feature is a protrusion feature whose profile will be created on the top plane.

1. Start Solid Edge in the **Part** environment and then select the top plane as the sketching plane for the protrusion feature.
2. Draw the profile for the base feature using various sketching tools.
3. Add the required relationships and dimensions to the sketch, as shown in Figure 5-48.

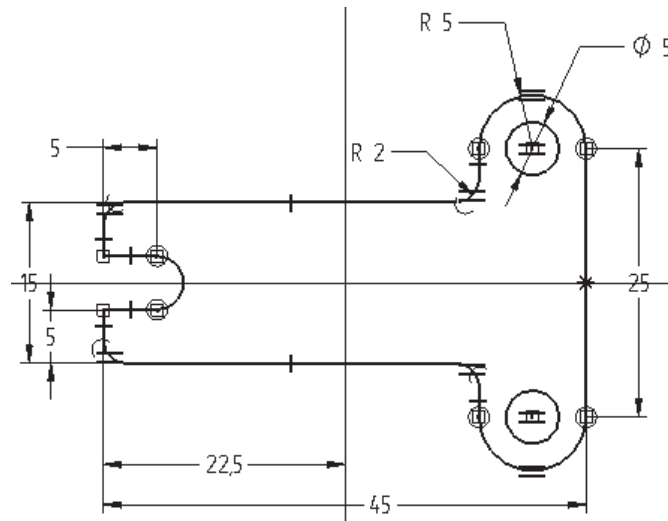


Figure 5-48 Dimensioned profile for the base feature

4. Exit the sketching environment; the **Extent** step is automatically invoked and you are prompted to click to set the distance or key in the value.
5. Enter **5** as the value in the **Distance** edit box in the ribbon bar and click anywhere in the drawing window above the top plane to specify the side of the feature creation.

The preview of the base feature is displayed.

6. Choose **Finish** and then choose **Cancel** from the ribbon bar. The base feature of the model is shown in Figure 5-49.

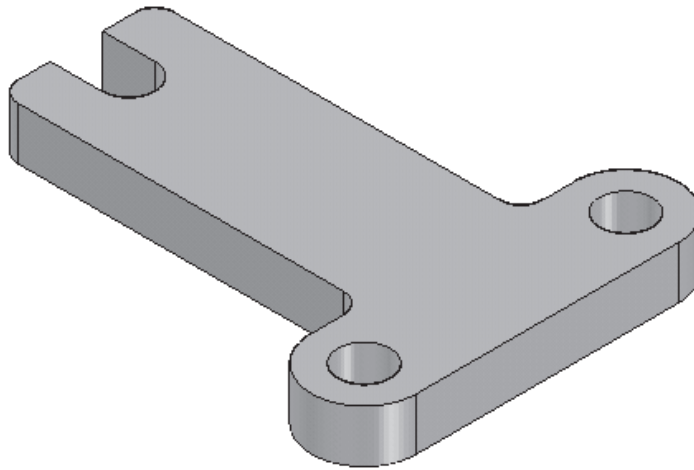


Figure 5-49 Base feature of the model

Creating the Second Feature

The second feature needs to be created using an angled reference plane, which in turn will be created using the right edge of the top face of the base feature. Because this angled reference plane will be used to create only one feature, it is recommended that you create this plane from within the **Protrusion** tool.

1. Invoke the **Protrusion** tool to display the **Protrusion** ribbon bar.
2. From the **Create-From Options** drop-down list, select the **Angled Plane** option. You are prompted to click on a planar face or a reference plane.

To create an angled plane, you first need to select a reference plane or a planar face of the model from which the new plane will be at an angle. In this case, you need to select the top face of the base feature to define the angled plane.

3. Select the top face of the base feature, as shown in Figure 5-50.

The face is highlighted and you are prompted to click on the face, edge, or plane to be the base of the profile plane. You need to select an edge through which the angled plane will pass. In this case, the right edge of the top face of the base feature will be selected as the edge through which the plane will pass.

4. Select the edge through which the angled plane will pass, see Figure 5-50.

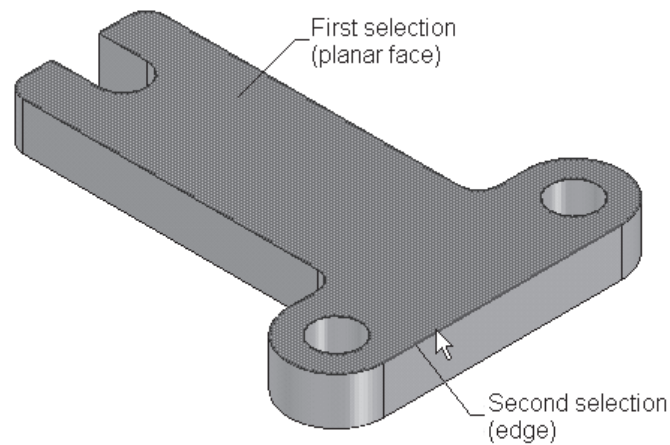


Figure 5-50 Making selections to create an angled plane

Next, you are prompted to click near the end of the axis for the reference plane orientation.

5. Click in the lower half of the drawing window to make sure that the orientation of the positive X axis is from the left to right.

As soon as you specify the orientation of the plane, a preview of the resulting plane is displayed and the **Angle** and **Step** edit boxes are displayed in the ribbon bar. When you move the cursor in the drawing window, the preview of the resulting plane is modified accordingly.

6. Enter **45** as the value in the **Angle** edit box and then click close to the right face of the base feature to define the plane.

The plane is defined and is selected as the sketching plane. Also, the sketching environment is invoked where you can draw the profile for the second feature.

To create the profile for the second feature, you can project the entities from the base feature and then modify them to complete the profile.

7. Choose the **Include** button from the **Draw** toolbar; the **Include Options** dialog box is displayed.
8. Make sure the **Include with offset** and **Include internal face loops** check boxes are cleared in this dialog box. Choose **OK** from this dialog box and then select the edges to project, as shown in Figure 5-51.



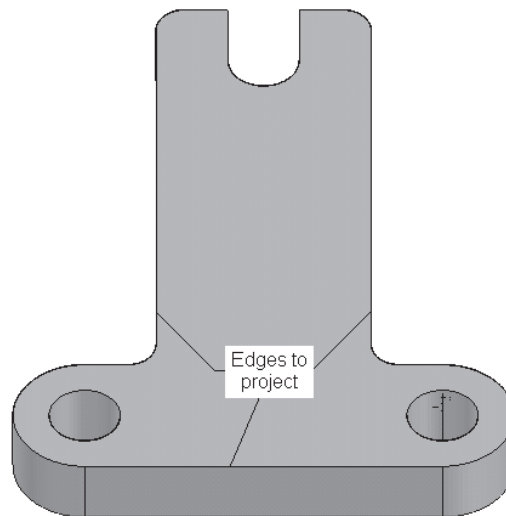


Figure 5-51 Selecting the entities to project

9. Trim and extend the projected entities using the sketching tools and then draw a tangent arc to complete the profile of the base feature, refer to Figure 5-52.

Note that if you use the projected entities in the form in which they are projected, you do not need to dimension them. However, if you modify them by trimming and extending, you need to add the dimensions to them.

10. Add a circle to the profile and then add the required dimensions to the profile and the circle, as shown in Figure 5-52.

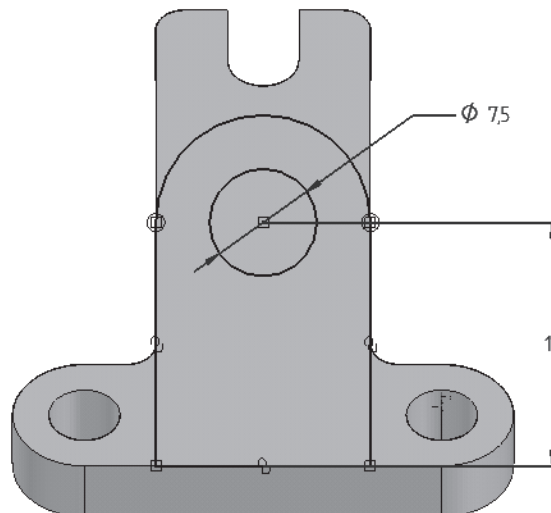


Figure 5-52 Profile for the second feature

11. Exit the sketching environment; the **Extent** step is automatically invoked and you are prompted to click to set the distance or key in the value.
12. Choose the **Through Next** button from the ribbon bar. You are prompted to click to select the side.
13. Click below the angled plane to define the side of the feature creation. The preview of the second feature is displayed merging with the base feature.
14. Choose **Finish** and then choose **Cancel** from the ribbon bar.
15. Press and hold the SHIFT key down and select the three base reference planes from the **EdgeBar**. Right-click and choose **Hide** from the shortcut menu to turn off the display of the base reference planes. The final model for Tutorial 1 is shown in Figure 5-53.

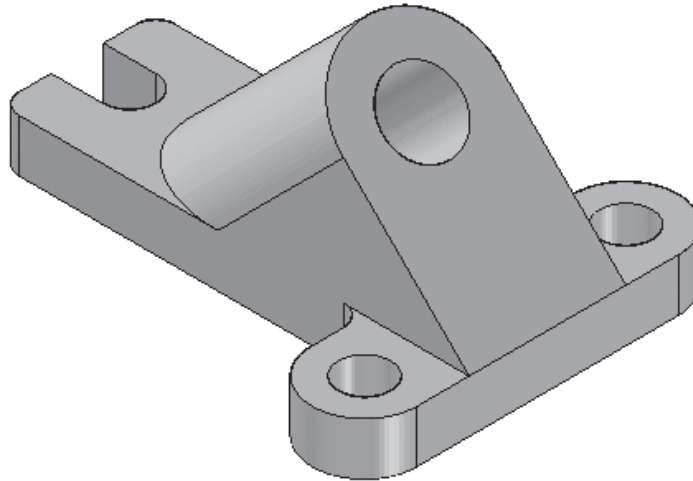


Figure 5-53 Final model for Tutorial 1

Saving the Model

1. Save the model with the name given below and then close the file.

\\Solid Edge\\c05\\c05tut1.par



Note

*The holes shown in the model for Tutorial 1 can also be drawn directly using the **Hole** tool. The use of this tool will be discussed in later chapters.*

Tutorial 2

In this tutorial, you will create the model shown in Figure 5-54. Its dimensions are given in the views shown in Figure 5-55. After creating the model, save it with the name *\\Solid Edge\\c05\\c05tut2.par*. **(Expected time: 45 min)**

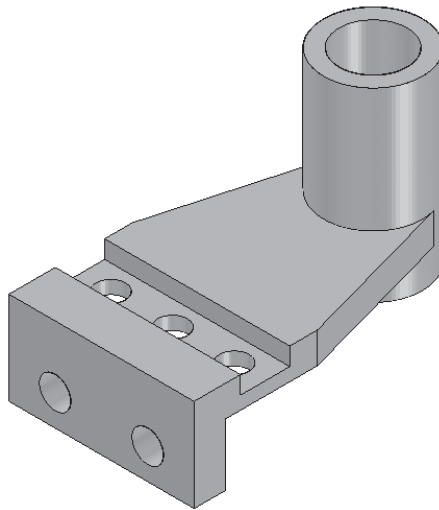


Figure 5-54 Model for Tutorial 2

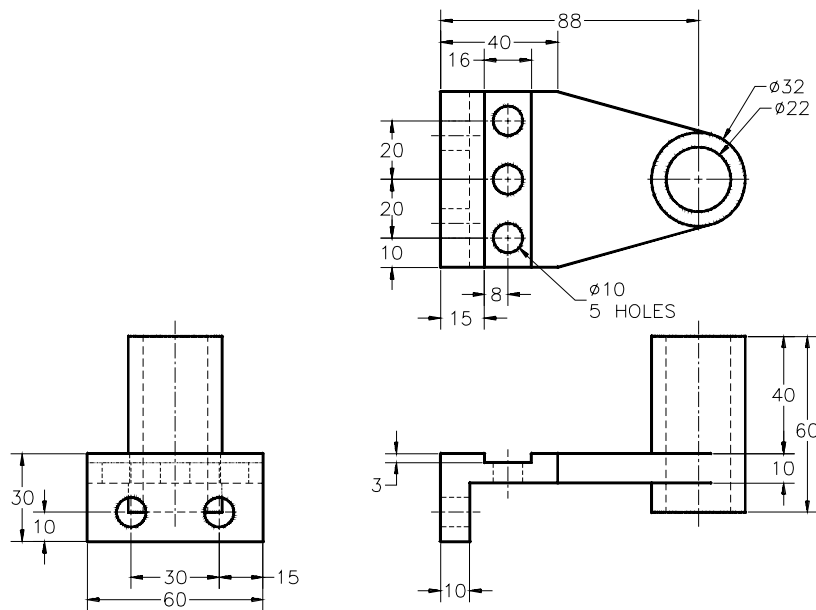


Figure 5-55 Dimensions of the model

As mentioned earlier, whenever you start creating a model, you need to first determine the number of features in it and then the sequence in which they will be created. The model for Tutorial 2 is a combination of three protrusion features and three cutouts to define the holes. The base feature will be created on the right plane. The second feature will be created on the top face of the base feature. The third feature will be created by defining a reference plane at an offset of 10 units from the bottom face of the second feature. The cutouts will be created on the planar face of the protrusion features.

The following steps outline the procedure to create the given model:

- a. Create the base feature on the right plane, refer to Figure 5-57.
- b. Select the top planar face of the base feature as the sketching plane and then create the second feature, refer to Figure 5-60.
- c. Define a reference plane at an offset of 10 units from the bottom face of the second feature and use it to create the third feature, refer to Figure 5-61.
- d. Create two holes on the left face of the base feature using the **Cutout** tool, as shown in Figure 5-63.
- e. Similarly, create the remaining cutouts to complete the model, refer to Figure 5-64.

Creating the Base Feature

As mentioned earlier, the base feature is a protrusion feature and so the profile of this feature will be created on the right plane.

1. Start a new part file and then select the right plane as the sketching plane for the protrusion feature.
2. Draw the profile for the base feature using various sketching tools.
3. Add the required relationships and dimensions to the sketch, as shown in Figure 5-56.

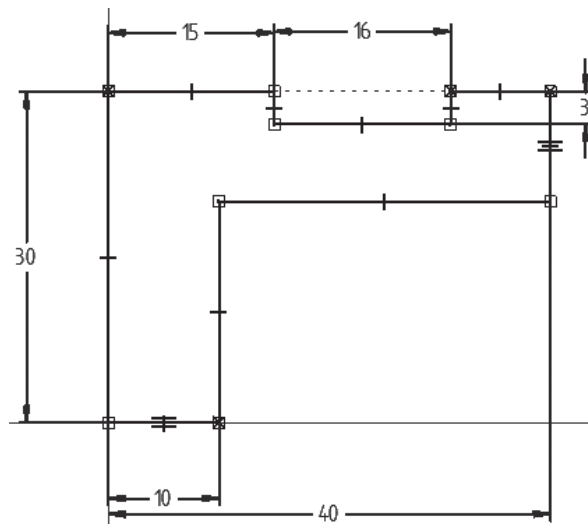


Figure 5-56 Dimensioned profile for the base feature

4. Exit the sketching environment; the **Extent** step is automatically invoked and you are prompted to click to set the distance or key in the value.
5. Choose the **Symmetric Extent** button and then enter **60** as the value in the **Distance** edit box in the ribbon bar; the preview of the base feature is displayed.
6. Choose **Finish** from the ribbon bar. The base feature of the model is shown in Figure 5-57.

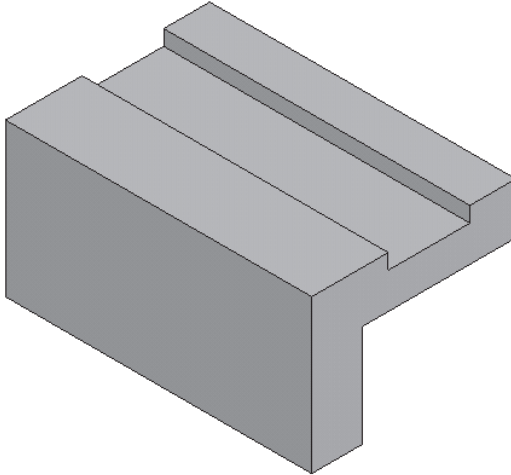


Figure 5-57 Base feature of the model

Creating the Second Feature

As mentioned earlier, the second feature is a protrusion feature and its profile is created on the top face of the base feature. Because you did not exit the **Protrusion** command, it is still active and you are prompted to click on the planar face or reference plane.

1. Move the cursor over the top face of the base feature and then press the N key to orient the sketching plane, as shown in Figure 5-58.
2. Draw the profile for the second feature using various sketching tools and then add the required dimensions and relationships to the profile, as shown in Figure 5-59. Note that you do not need to draw the vertical line on the left to close the sketch. While creating the feature, you can define the side of material addition to complete the feature.
3. Exit the sketching environment. Because the profile for the second feature is open from the left, the **Side** step is invoked and you need to click to accept the displayed side or select the other side in the view.

You need to add the material inside the profile. Therefore, you need to ensure that the arrow points inside the sketch to create this feature.

4. Move the cursor inside the profile to ensure that the arrow points inside. Click to define the direction and exit this step.

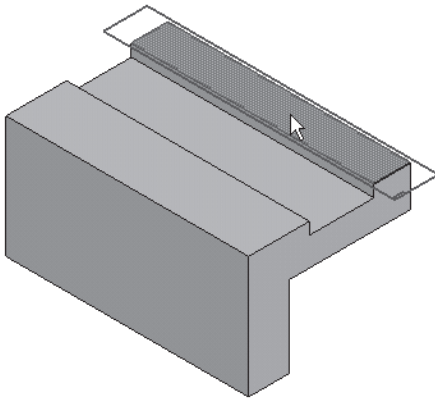


Figure 5-58 Specifying the orientation of the sketching plane for the second feature

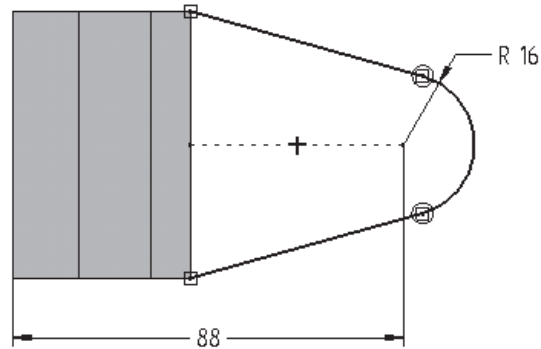


Figure 5-59 Open profile for the second feature

The **Extent** step is automatically invoked and you are prompted to click to set the distance or key in the value.

5. Choose the **From/To Extent** button from the ribbon bar; you are prompted to select the “from” surface or right-click to extrude from the profile plane.

Because you want to start the extrusion from the plane on which the profile is created, you can right-click to select it.

6. Right-click to select the profile plane as the plane from which the extrusion will start; you are prompted to select the “to” surface.
7. Using the **QuickPick** tool, select the immediate bottom face of the base feature that is at a distance of 10 units from the top face.

The preview of the model is displayed in the drawing window.

8. Choose **Return** from the ribbon bar to complete the feature. The model, after creating the second feature, is shown in Figure 5-60.

Creating the Third Feature

The third feature is also a protrusion feature and its profile is created on a reference plane created at an offset of 10 units from the bottom face of the second feature.

1. Select the **Parallel Plane** option from the **Create-From Options** drop-down list in the ribbon bar and then using the **QuickPick** tool, select the bottom face of the second feature.
2. Enter **10** as the value in the **Distance** edit box. Move the cursor to the lower half of the drawing window and click to define the plane.

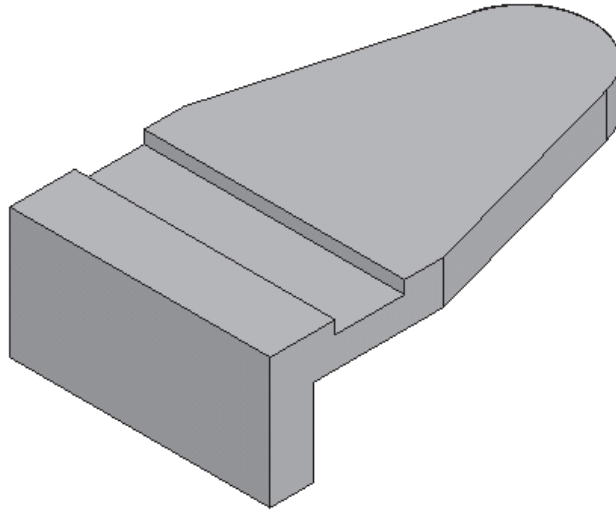



Figure 5-60 Model after creating the second feature

The sketching environment is invoked. The profile for the third feature is a circle with the diameter equal to the diameter of the arc in the second feature.

3. Draw a circle concentric with the arc in the second feature and with 32 diameter. You can locate the center of the arc by moving the cursor once over it.
4. Exit the sketching environment and choose the **Finite Extent** button from the ribbon bar.
5. Enter **60** as the value in the **Distance** edit box. Move the cursor in the upper half of the drawing window and click to define the feature.
6. Choose **Finish** and then choose **Cancel** to create the feature and exit the **Protrusion** tool. The model, after creating the third protrusion feature, is shown in Figure 5-61.

Creating a Cutout on the Left Face of the Base Feature

The fourth feature is a cutout that is required to define a hole on the left face of the base feature. The profile of this cutout is created on the left face of the base feature.

1. Choose the **Cutout** button from the **Features** toolbar. The **Cutout** ribbon bar is displayed with the **Plane or Sketch** step active. The **Coincident Plane** option is selected from the **Create-From Options** ribbon bar. 
2. Select the left face of the base feature to define a sketching plane on this face. The sketching environment is invoked.

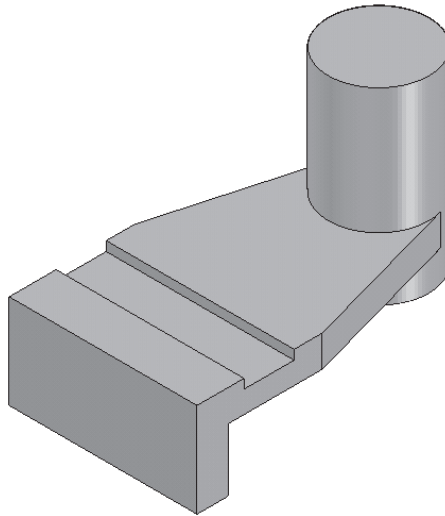


Figure 5-61 Model after creating the third feature

3. Draw the profile of the cutout. The profile consists of two circles each with a diameter of 12, as shown in Figure 5-62.
4. Exit the sketching environment and choose the **Through Next** button from the ribbon bar.
5. Move the cursor to the right of the model and left click to specify the direction of the cutout feature. The model, after creating the cutout feature, is shown in Figure 5-63.

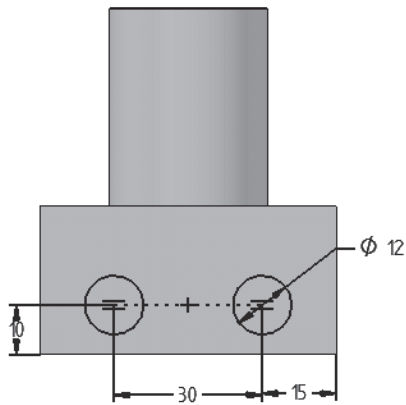


Figure 5-62 Profile for the cutout

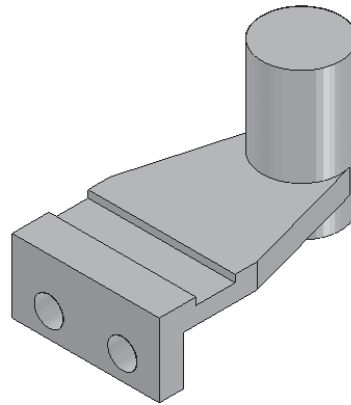


Figure 5-63 Model after creating the cutout

Creating the Remaining Features

1. Similarly, create the remaining two cutouts. The model, after creating all features, is shown in Figure 5-64.

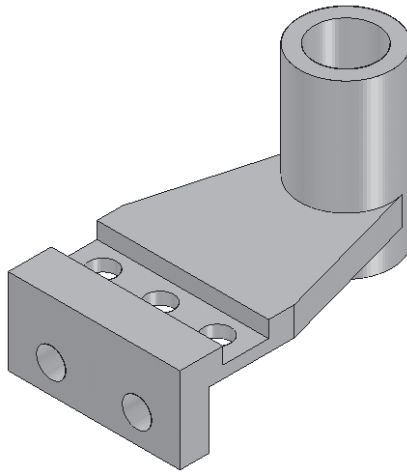


Figure 5-64 Final model for Tutorial 2

Saving the Model

1. Save the model with the name given below and then close the file.

\\Solid Edge\\c05\\c05tut2.par

Tutorial 3

In this tutorial, you will create the model shown in Figure 5-65. Its dimensions are given in the views shown in Figure 5-66. After creating the model, save it with the name *\\Solid Edge\\c05\\c05tut3.par*. (Expected time: 45 min)

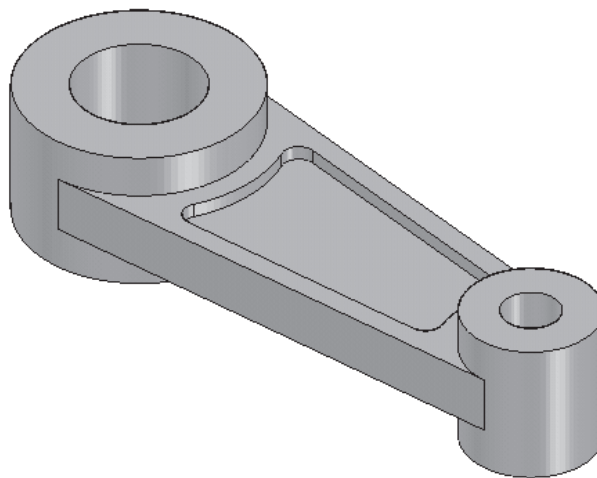


Figure 5-65 Model for Tutorial 3

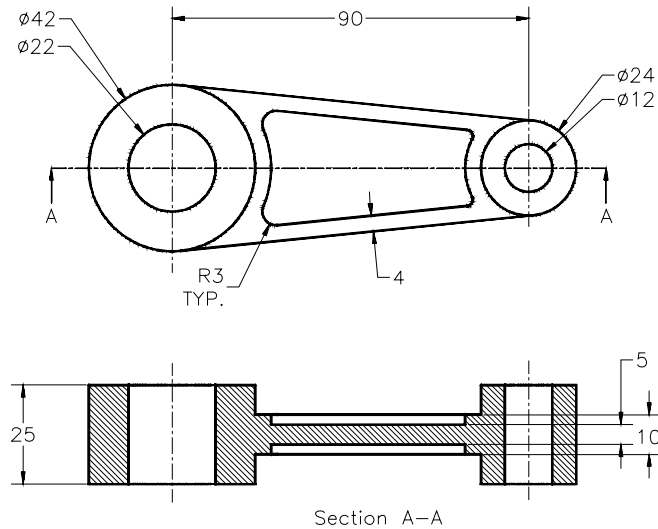


Figure 5-66 Dimensions of the model for Tutorial 3

The model for Tutorial 3 is a combination of three protrusion features and one cutout. The base feature will be created on the top plane and extruded symmetrically. The second feature will also be created on the top plane and extruded symmetrically. Next, you will create a sketch on the top plane that will be used to create a cutout as well as a protrusion feature in the second feature.

The following is the list of steps that are required to create the given model:

- Create the base feature on the top plane, refer to Figure 5-68.
- Create the second feature also on the top plane, refer to Figure 5-70.
- Invoke the **Sketch** tool and draw a sketch on the top plane, refer to Figure 5-71.
- Create a cutout in the second feature using the sketch, refer to Figure 5-72.
- Create a protrusion feature using the same sketch to complete the model, refer to Figure 5-73.

Creating the Base Feature

- Start a new part file and then select the top plane as the sketching plane for the protrusion feature.
- Draw the profile for the base feature using various sketching tools.
- Add the required relationships and dimensions to the sketch, as shown in Figure 5-67.
- Exit the sketching environment and then extrude the sketch symmetrically through a distance of 25 units. The base feature of the model is shown in Figure 5-68.

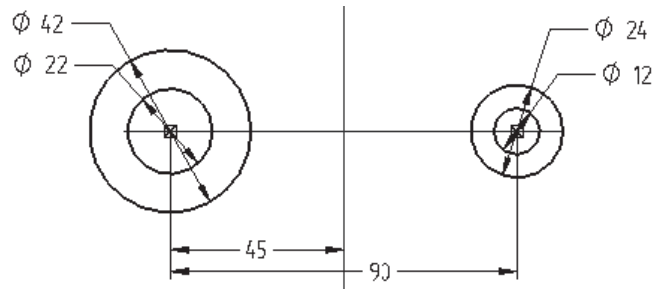


Figure 5-67 Profile for the base feature

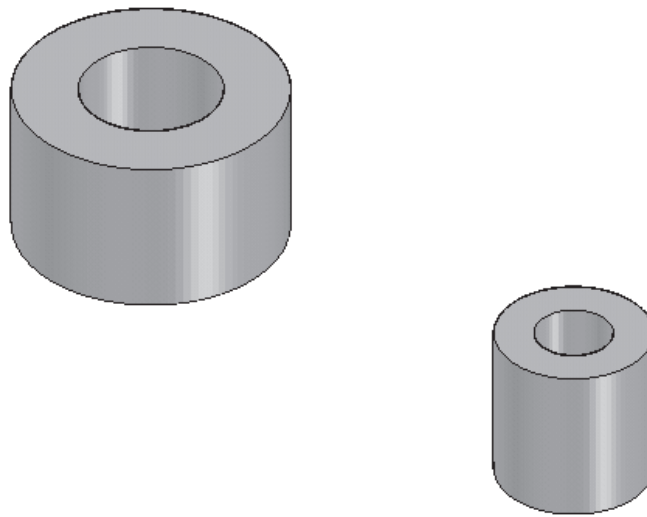


Figure 5-68 Base feature of the model

Creating the Second Feature

The second feature is also created on the top plane. To draw the profile of this feature, you will have to project the outer circles of the base feature and then draw tangent lines on those circles.

1. Choose the **Include** button from the **Draw** toolbar to display the **Include Options** dialog box.



2. Ensure the **Include with offset** and **Include internal face loops** check boxes are cleared. Exit the dialog box and then select the outer circles from the base feature to project them on the current sketching plane.
3. Draw tangent lines on these circles and then trim the unwanted portion of the profile. Because you used the projected entities, you do not need to dimension the profile. The profile for the second feature is shown in Figure 5-69.
4. Exit the sketching environment and then extrude the sketch symmetrically through a distance of 10 units. The model, after creating this feature, is shown in Figure 5-70.

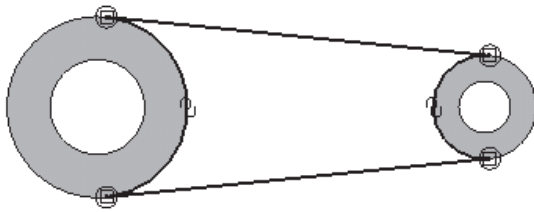


Figure 5-69 Profile for the second feature

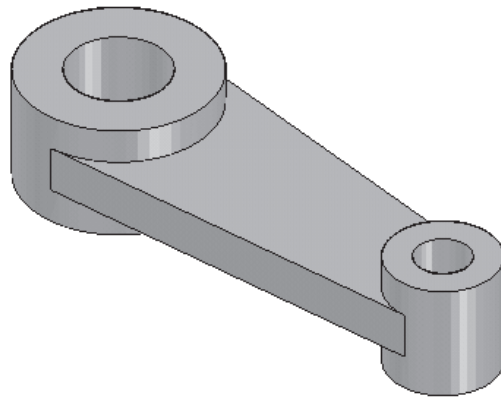



Figure 5-70 Model after creating the feature

Creating the Sketch for the Third and Fourth Features

Next, you need to create a sketch that will be used as a profile for the third and fourth features. This sketch is also created on the top plane. To create this sketch, you need to project the edges of the second feature with an offset of 4 units.

1. Choose the **Sketch** button from the **Features** toolbar and select the top plane as the sketching plane.
2. Choose the **Include** button from the **Draw** toolbar to display the **Include Options** dialog box. 
3. Select the **Include with offset** check box and make sure that the **Include internal face loops** check box is cleared.
4. Exit the dialog box and then select **Loop** from the **Select** drop-down list.
5. Move the cursor close to the upper inclined edge of the second feature and click when the complete loop, defined to create the second feature, is highlighted. The complete loop is projected.
6. Right-click to proceed to the **Offset** step. Enter **4** as the value in the **Distance** edit box and

move the cursor to the center of the model. Click when the offset arrow points inside the model.

The outer loop is projected with an offset of 4 units.

7. Add a fillet of radius 3 to the vertices of the projected loop. The sketch is shown in Figure 5-71.

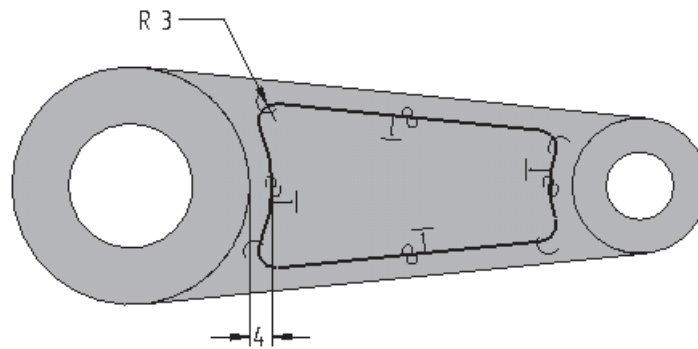


Figure 5-71 Sketch after adding the fillet

8. Exit the sketching environment and then exit the **Sketch** tool. This sketch is saved as **Sketch 1** in the **EdgeBar**.

Creating the Cutout

Next, you need to create a cutout using the sketch created in the previous step.

1. Choose the **Cutout** button from the **Features** toolbar.
2. Select the **Select from Sketch** option from the **Create-From Options** drop-down list in the ribbon bar.
3. Select the sketch and then right-click to proceed to the **Extent** step.
4. Choose the **Symmetric Extent** button from the ribbon bar and then enter **10** as the value in the **Distance** edit box.
5. Exit the **Cutout** tool. The model, after creating the cutout, is shown in Figure 5-72. This figure also shows the sketch because its display is not turned off.



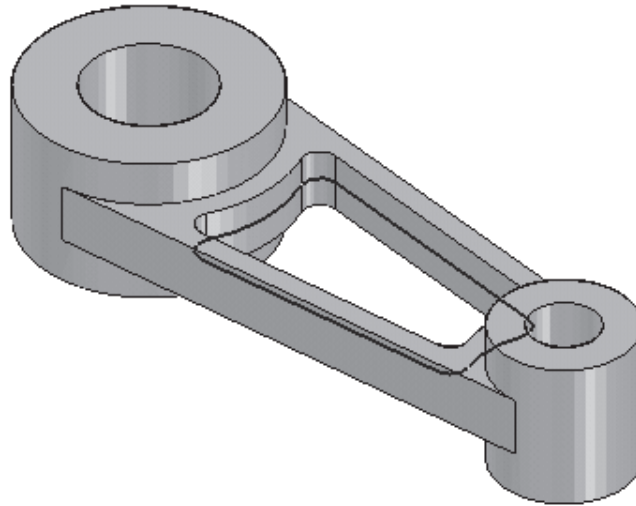


Figure 5-72 Model after creating the cutout

Creating the Protrusion Feature

1. Choose the **Protrusion** button from the **Features** toolbar and select the **Select from Sketch** option from the **Create-From Options** drop-down list.
2. Select the sketch and then right-click to proceed to the **Extent** step.
3. Choose the **Symmetric Extent** button from the ribbon bar and then enter **5** as the value in the **Distance** edit box.
4. Choose **Finish** and then choose **Cancel** from the ribbon bar to exit the **Protrusion** tool.

Before you save and close the file, it is recommended that you turn off the display of the sketches in the model.

5. Right-click on **Sketch 1** in the **EdgeBar** and choose **Hide** from the shortcut menu. The display of the sketch is turned off. The final model for Tutorial 3 is shown in Figure 5-73.

Saving the Model

1. Save the model with the name given below and then close the file.

\\Solid Edge\\c05\\c05tut3.par

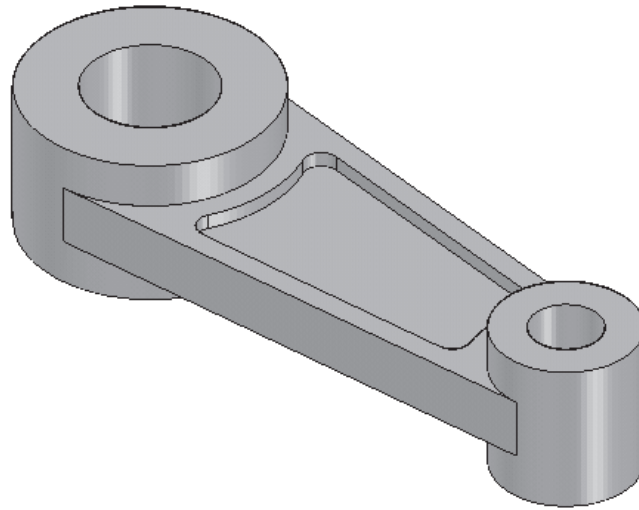


Figure 5-73 Final model for Tutorial 3

Self-Evaluation Test

Answer the following questions and then compare your answers with those given at the end of the chapter:

1. The **Treatment** step is automatically invoked while creating the protrusion features. (T/F)
2. The reference planes that are available by default are called the base reference planes. (T/F)
3. The **Parallel Plane** tool can be used to create a reference plane parallel to a selected base reference plane, other reference plane, or planar face. (T/F)
4. Cutouts are created by removing the material, defined by a profile, from one or more existing features. (T/F)
5. While creating the cutouts, you can create open profiles and use the _____ step extensively to define the direction of the material removal.
6. The _____ tool is used to create a plane that is normal to a selected sketched curve or an edge of the model.
7. In Solid Edge, you can create coordinate systems using _____ options.
8. The _____ tool is used to create a revolved cutout by removing the material defined by the sketch.
9. If the plane that you select to define the coincident plane does not have a linear edge, the _____ direction is defined using the base reference plane.

10. While creating the protrusion feature, you can use the _____ step to add a draft or a crown to the protrusion feature.

Review Questions

Answer the following questions:

- In Solid Edge, using which one of the following tools can you project the edges of the existing features on the current sketching plane?
(a) **Project** (b) **Include**
(c) **Insert** (d) None
- In Solid Edge, using which one of the following dialog boxes can you create user-defined standard views that are automatically added to the **Named Views** drop-down list?
(a) **Views** (b) **Named Views**
(c) **Drawing Views** (d) None
- Which one of the following is not a type of reference plane?
(a) **Base reference planes** (b) **Global reference planes**
(c) **Local reference planes** (d) **Sample reference planes**
- Which reference planes are created separately as features using the tools available in the **Features** toolbar to create reference planes?
(a) **Base reference planes** (b) **Global reference planes**
(c) **Local reference planes** (d) **Sample reference planes**
- Which rule is used by Solid Edge to determine the direction of rotation of the axes?
(a) **Right-hand thumb** (b) **Right-hand**
(c) **Left-hand thumb** (d) **Left-hand**
- Which two of the following options are also available while selecting the option to create reference planes from the **Create-From Options** drop-down list in the ribbon bar?
(a) **Feature's Plane** (b) **Base Plane**
(c) **Last Plane** (d) **Blank Plane**
- The **Parallel Plane** tool is used to create a reference plane that is coincident to a base reference plane, another reference plane, or a planar face of the model. (T/F)
- You cannot use open profiles to create cutouts. (T/F)
- The **Plane by 3 Points** tool is used to create reference planes by selecting three points. (T/F)

10. Solid Edge automatically creates reference axes when you create a revolved feature, hole feature, or any other circular or semicircular feature. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 5-74. Its dimensions are given in the views shown in Figure 5-75. After creating the model, save it with the name given below.

\\Solid Edge\c05\c05exr1.par

(Expected time: 30 min)

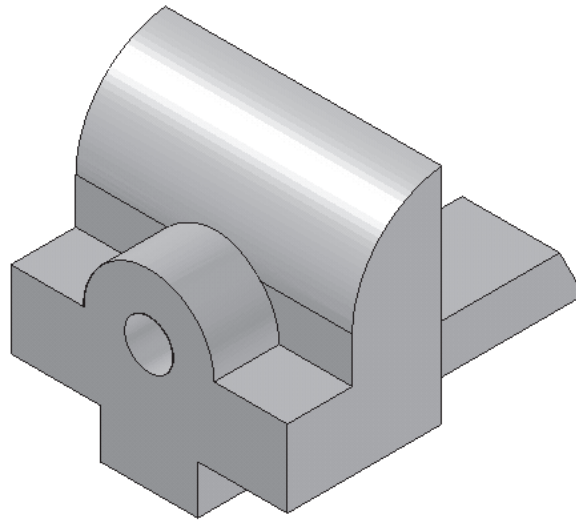


Figure 5-74 Model for Exercise 1

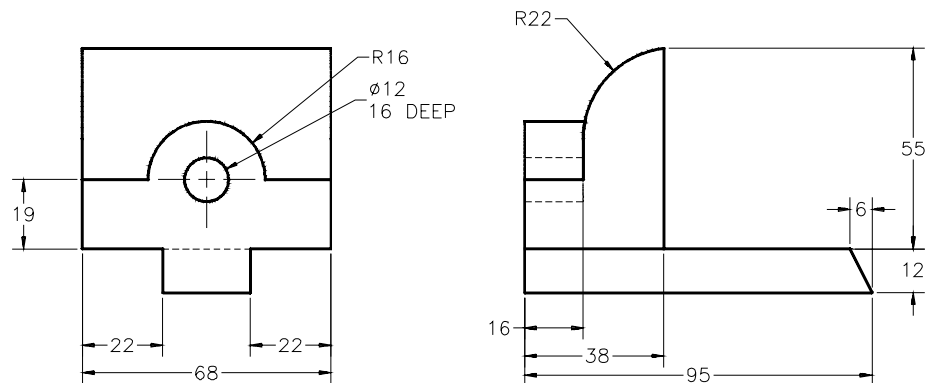


Figure 5-75 Dimensions of the model for Exercise 1

Exercise 2

Create the model shown in Figure 5-76. Its dimensions are given in the views shown in Figure 5-77. After creating the model, save it with the name given below.

\\Solid Edge\c05\c05exr2.par

(Expected time: 30 min)

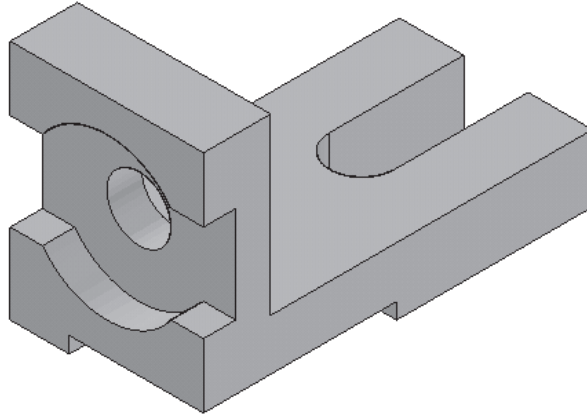


Figure 5-76 Model for Exercise 2

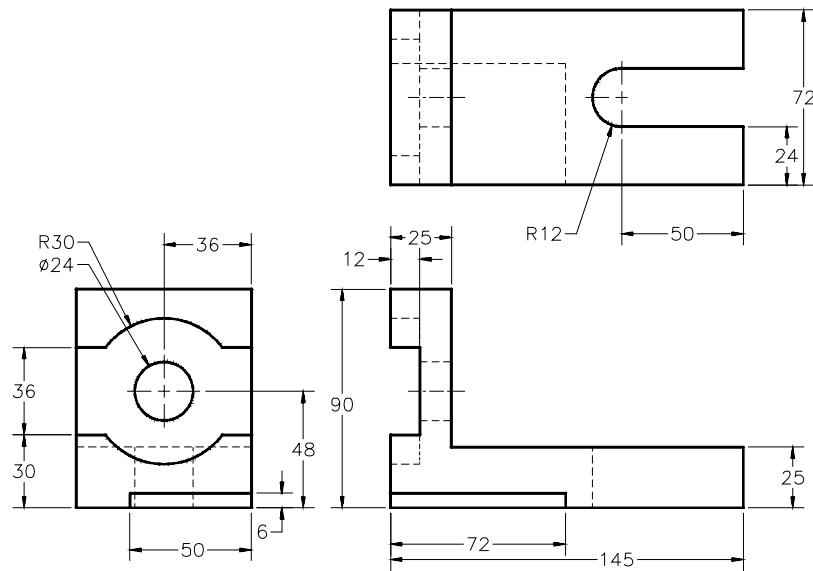


Figure 5-77 Model for Exercise 2

Answers to Self-Evaluation Test

1. F, 2. T, 3. T, 4. T, 5. Side, 6. Plane Normal to Curve, 7. two, 8. Revolved Cutout,