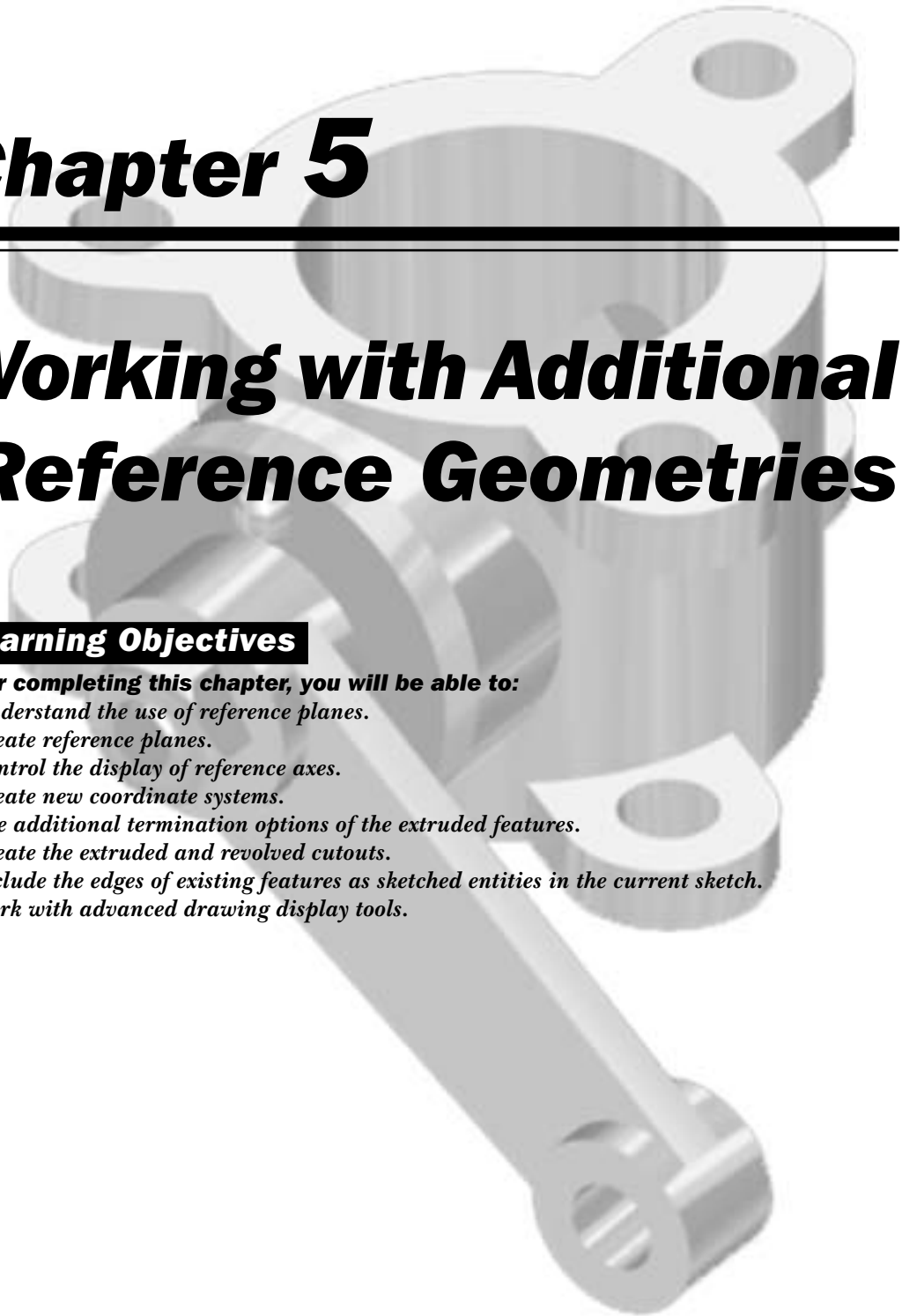
Chapter 5

Working with Additional Reference Geometries



Learning Objectives

After completing this chapter, you will be able to:

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

ADDITIONAL SKETCHING AND REFERENCE PLANES

As mentioned earlier, most of the mechanical designs consist of a number of sketched, reference, and placed features that are integrated together. In the previous chapter, you learned how 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. Mostly, additional features are not created on the default plane on which the base feature is created, refer to the models shown in Figures 5-1 and 5-2.

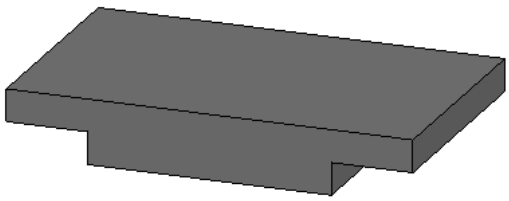


Figure 5-1 *Base feature of the model*

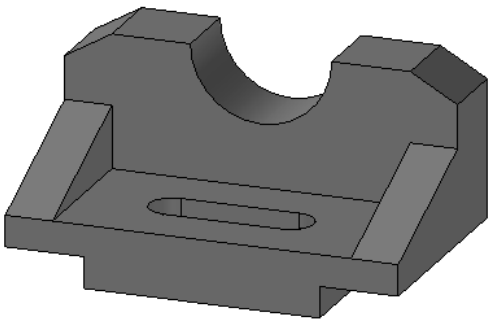


Figure 5-2 *Model with multiple features*

Figure 5-1 is a base feature, which is an extrude feature, created on the default plane, and Figure 5-2 shows a model with the same base feature but with additional features created on it.

After creating the base feature, you need to create another extruded feature, two rib features, and a cut feature, refer to Figure 5-3. All these features are sketched features and require additional sketching or reference planes to draw their sketches. 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 those that are created while defining a feature in the Part environment. For example, whenever you invoke a tool to create a sketch-based feature in the **Ordered** environment, the command bar provides tools to create reference planes. The reference plane created at this stage will be used to create only this particular feature. Therefore, these types of reference planes are called local reference planes. These planes are not displayed in the drawing window or in the **PathFinder**.

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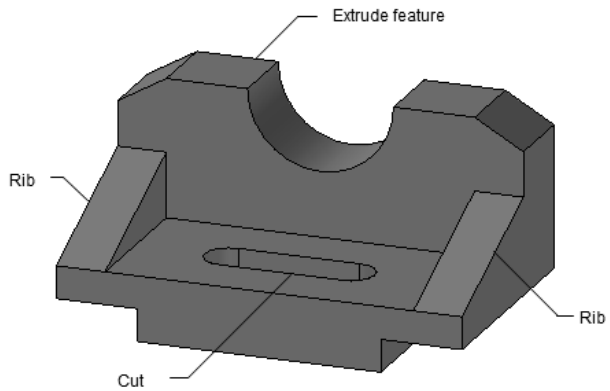


Figure 5-3 *Various features in the model*

Global Reference Planes

The planes that are created separately as features with help of the tools in the **Reference** group of the **Ribbon** are called global reference planes. These planes are displayed in the drawing window as well as in the docking window and can be used to create additional features.

CREATING REFERENCE PLANES (SYNCHRONOUS)

In Solid Edge, you can create various type of local or global reference planes. The procedures to create various types of reference planes are discussed next.

Creating a Coincident Plane

Ribbon: Home > Planes > Coincident Plane



You can create a reference plane coincident to a base reference plane, any other reference plane, or the planar face of a model using the **Coincident Plane** tool.

To create a coincident plane, invoke the **Coincident Plane** tool; you will be prompted to click on a planar face or a reference plane. Move the cursor on the reference plane or planar face on which you want to create the coincident plane; the plane will be highlighted in orange. Also, the preview of the reference plane being created will be displayed in the drawing window. Click at this point; the plane coincident to the selected entity will be created.

You can select the other edges of the planar face or the reference plane to define the X-axis of the new plane by pressing the N or the 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.

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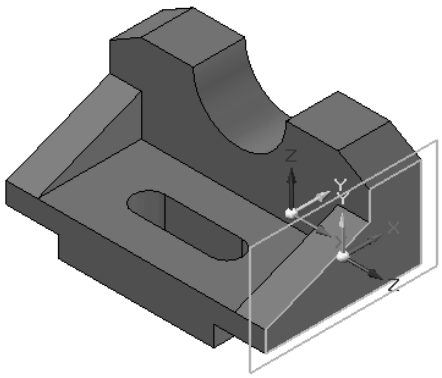


Figure 5-4 *Creating a coincident plane with the bottom edge defining the orientation of the X-axis*

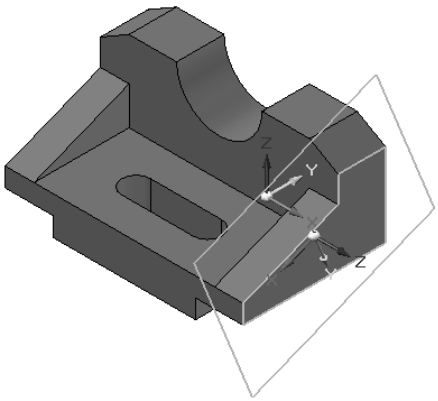


Figure 5-5 *Creating a coincident plane with the inclined edge defining the orientation of the X-axis*

You can also toggle the X-axis direction of a new plane to a positive or negative X-axis by pressing the T key. For example, if a 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 a triad displayed at the center 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 right (Figure 5-5) will be reversed and will point toward the left, refer to Figure 5-6. The triad at the origin of the plane indicates the direction of the positive X-axis.

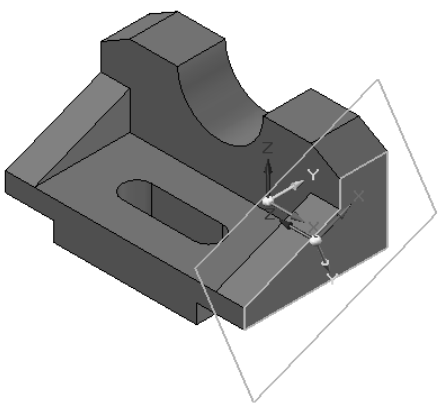


Figure 5-6 *Preview of the reference plane after reversing the positive X-axis direction*

After you have selected all the options for creating a reference plane, its preview will be displayed in the drawing window. At this stage, click in the drawing area to create the plane.

After creating a coincident reference plane, the steering wheel and the **Move QuickBar** will be displayed with the new reference plane.

Modifying Planes Using the Steering Wheel

The steering wheel is used to modify the position of a plane or face. Various components of the steering wheel are shown in Figure 5-7. If you press and drag the origin knob, the steering wheel will move to a new location. To move the selected entity along the primary or secondary axis, press and drag the primary or secondary axis arrow. In order to set the direction of the primary or secondary axis, press and drag the primary or secondary bearing that is attached

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to the primary or secondary axis. You can specify the exact distance value in the edit box that is displayed while moving the face. On moving a selected face, its adjacent faces are updated automatically. The tool plane of the steering wheel allows the selected entities to move along the tool plane. The torus of the steering wheel is used for modifying the inclination of the selected entities.

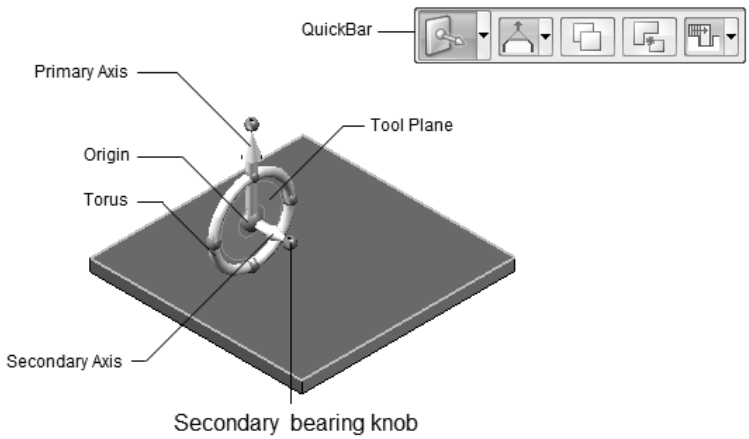


Figure 5-7 *The steering wheel and the* ***QuickBar***

Creating a Plane Normal to an Edge or a Sketched Curve

Ribbon: Home > Planes > Planes drop-down > Normal to Curve



You can create a plane that is normal to a selected sketched curve or an edge of the model using the **Normal to Curve** tool. 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 the edge for the normal plane. Once you move the cursor close to the sketched curve or the 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 command 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 5-8 shows a plane being created normal to a sketched curve.

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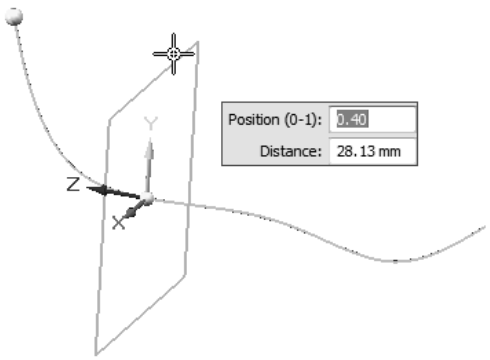


Figure 5-8 *Creating a plane normal to a sketched curve*

Creating a Plane Using Three Points

Ribbon: Home > Planes > Planes drop-down > By 3 Points



You can create a reference plane on any three specified points by using the **By 3 Points** tool. The 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 **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** drop-down from the command 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 5-9 shows a plane being created using three points.

Creating a Tangent Plane

Ribbon: Home > Planes > Planes drop-down > Tangent



You can create a reference plane tangent to a selected curved surface using the **Tangent** tool. To create a tangent plane, invoke this tool; you will be prompted to click on a curved face. Select the curved surface from the drawing area. Next, you need to click on the curved surface at the required angle or you can enter the value of angle in the Edit box; the 3D indicator with three axes and a origin will be displayed on the preview of the plane. The indicator allows you to rotate the view freely in 3D space or around any of these three axes. Figure 5-10 shows the tangent plane being created on the curved surface. A tangent plane can be created on a cylinder, sphere, cone, torus, or b-line surface.

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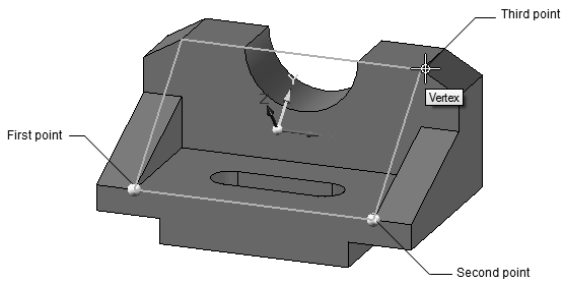


Figure 5-9 *Creating a plane using three points*

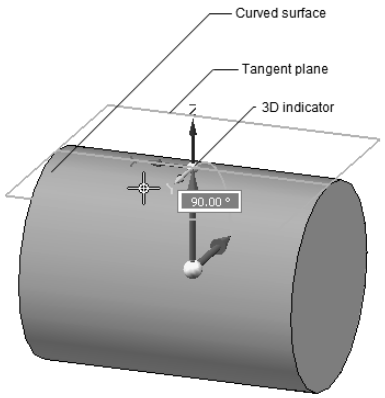


Figure 5-10 *Creating a tangent plane*

In addition to the tools mentioned earlier, there are some other tools that can be used for creating reference planes. But these tools are available only in the **Ordered Part** environment. In order to use these tools, you need to switch over to the **Ordered Part** environment. However, you can create these types of reference planes in the **Synchronous Part** environment by using the steering wheel. The procedures to create reference planes in the **Ordered Part** environment are discussed next.

Creating a Parallel Plane

Ribbon: Home > Planes > Planes drop-down> Parallel



In **Ordered Part** environment, you can create a reference plane parallel to a selected base reference plane, another reference plane, or a planar face using the **Parallel Plane** tool. To create a parallel plane, invoke the **Parallel** tool from the **Planes** drop-down, as shown in Figure 5-8; you will be prompted to click on a planar face or a reference plane. Define the orientation of the new plane and then click; the command 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 command bar to display the drop-down that has the keypoint options such as endpoint, midpoint, center point, tangent point, and so on. Choose the required option from this drop-down and then select the corresponding keypoint in the model; the new parallel plane will be placed at the selected keypoint. Figure 5-11 shows a parallel plane being created by using the center keypoint in the model and Figure 5-12 shows a parallel plane being created by using the tangent keypoint in the model.

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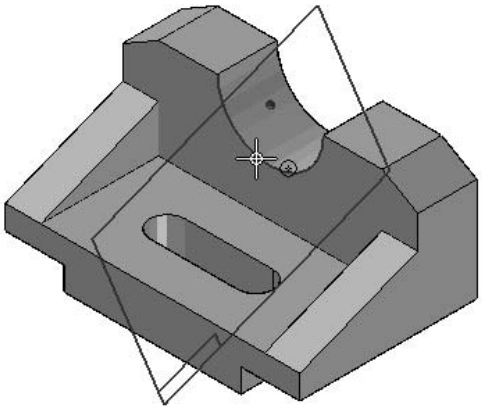


Figure 5-11 *Creating a parallel plane by defining the center point*

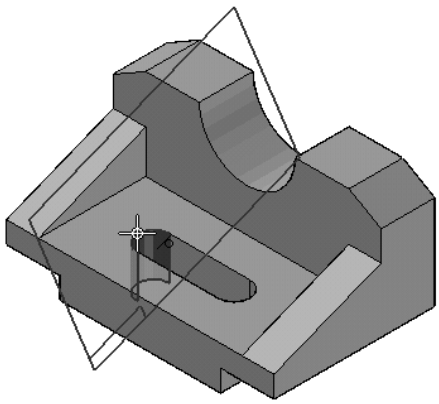


Figure 5-12 *Creating a parallel plane by defining the tangent point*

You can also create a plane parallel to a planar face in the **Synchronous Part** environment. To do so, create a coincident plane on the selected planar face; a steering wheel will be displayed. Click on the primary axis and drag it; a edit box will be displayed. Enter a value in it to define the location of the plane and press ENTER; plane will be moved to the desired location.

Creating an Angled Plane

Ribbon: Home > Planes > Planes drop-down > Angled



In the **Ordered Part** environment, you can create a reference plane that is at an angle to the selected plane and passes through a specified edge, axis, or plane using the **Angled** tool. 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 form 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. To invoke this tool, place the cursor over the entity for a while; a mouse symbol will be displayed. Right-click; the **QuickPick** list box will be displayed with all entities that can be selected at that location. Select the required entity from this list box.

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 of the command 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 5-13 shows various selections to create an angled plane and also the preview of the resulting plane.

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You can also create a plane at an angle to a planar face in the **Synchronous Part** environment using the steering wheel. To do so, create a plane coincident to the selected planar face; the steering wheel will be displayed on the plane. Click and drag the origin knob of the steering wheel and place it on the edge through which the new plane will pass. On doing so, the primary or secondary axis of the steering wheel will coincide with the edge. Next, click on the torus of the steering wheel; a edit box will displayed. Specify a value in it and press ENTER; a plane will be created at an angle specified in the edit box. When you enter 90 in the edit box, a perpendicular plane will be created.

Creating a Perpendicular Plane (Ordered)

Ribbon: Home > Planes > Planes drop-down > Perpendicular



The **Perpendicular** 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 a face, edge, or a plane to form 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 the 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 5-14 shows various selections to create a perpendicular plane and the preview of the resulting plane.

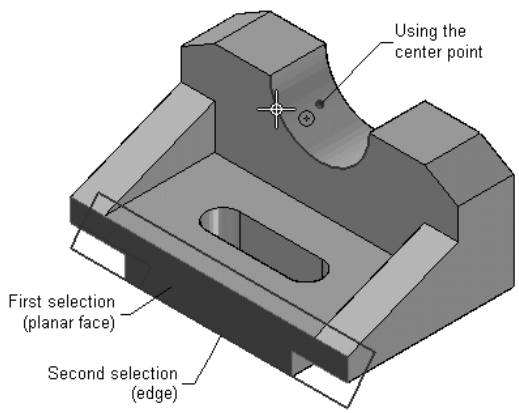


Figure 5-13 *Various selections to create an angled reference plane*

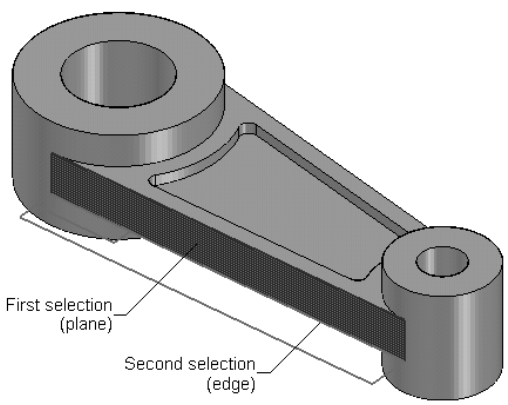


Figure 5-14 *Various selections made to create a perpendicular reference plane*

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Creating a Plane Coincident by Axis (Ordered)

Ribbon: Home > Planes > Planes drop-down > Coincident by Axis



The **Coincident 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 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.



Note

*While selecting the option to create reference planes in the **Ordered Part** environment from the **Create-From Options** drop-down list in the command 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 (Ordered)

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 the display of the axes on, right-click in the drawing window; a shortcut menu will be displayed. Choose **Show All > Reference Axes** from the shortcut menu; the reference axes of all the revolved features will be displayed in the drawing window. But the reference axes of the circular, semicircular, or hole features will not be displayed. To display the axes of these features, choose **Show All > Toggle Axes** from the shortcut menu that will be displayed on right-clicking in the drawing window; you will be prompted to click on the feature or the surface containing 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 the display of the reference axes off by choosing **Hide All > Reference Axes** from the shortcut menu that is displayed on right-clicking in the drawing window.



Note

*Once you have toggled the display of reference axes using the **Toggle Axes** option, you can turn the display of reference axes on or off. To do so, choose **Hide All > Reference Axes** or **Show All > Reference Axes** from the shortcut menu that is displayed on right-clicking in the drawing window.*

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

UNDERSTANDING COORDINATE SYSTEMS

Each 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

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coordinate systems according to your design requirements. These coordinate systems can be used as references to create reference planes, measure distances, copy parts, and so on.

Creating a Coordinate System

Ribbon: Home > Planes > Coordinate System



In **Ordered Part** environment, you can create coordinate systems in two ways: 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 are available in the **Coordinate System Options** dialog box, refer to Figure 5-15. To invoke this dialog box, choose **Coordinate System** from the **Planes** group of the **Home** tab.

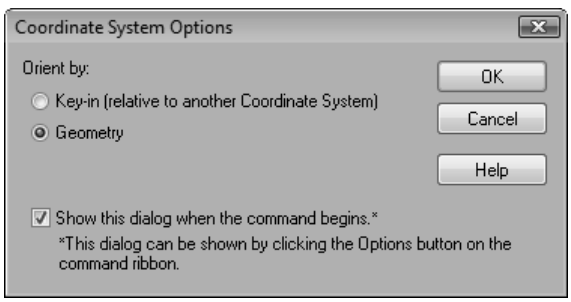


Figure 5-15 *The* ***Coordinate System Options*** *dialog box*

Select the **Key-in (relative to another Coordinate System)** radio button, if you want to create a coordinate system with respect to an existing coordinate system. Select the **Geometry** radio button, if you want to create a coordinate system by specifying the axis with respect to an existing geometry. When you select the **Key-in (relative to another Coordinate System)** and choose **OK**; the **Coordinate System** command bar will be displayed. The following steps will be available in the command bar.

Origin Step

This step is common to both the options for creating a coordinate system and is active by default when the **Coordinate System** command 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 command bar. By default, the points will be defined with respect to the default coordinate system. This is because **Model Space** is selected from the **Relative to** drop-down list in the command 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 with respect to them.

Orientation Step

In this step, you need to define the orientation of the X, Y, and Z axes of the new coordinate system with respect to those of the default coordinate system (model space) or any other

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coordinate system selected from the **Relative to** drop-down list. The coordinate system will be rotated around 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 **Xº** edit box, then the new coordinate system will be rotated by 20-degree around 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 points in the direction of the axes, then the direction of the curled fingers points toward the direction of rotation.

Enter the rotation angles in the **Xº**, **Yº**, and **Zº** 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.

On selecting the **Geometry** radio button from the **Coordinate System Options** dialog box, the following steps will be displayed in the command bar:

Origin Step

This step is similar to the **Origin Step** described in the previous option.

First Axis Step

This step 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 command bar. Therefore, 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 as the first axis, you can choose the button corresponding to that axis from the command bar.

You can select a point, linear element, planar face, or reference plane to define the axis after selecting the corresponding option from the **Select** drop-down list. After selecting the element to define the first axis, choose the **Accept** button from the command 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 corresponding to the axis that you want to define from the command bar. Note that the button of the axis, which is already defined, is not enabled in the command bar. The method of defining the axis is the same as that of defining 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 command bar to confirm the creation of the coordinate

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system. You can also choose the button of any of the steps to modify the selection made in that particular step.

Note

1. 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.

2. The coordinate system is displayed as a feature in the docking window.

In the **Synchronous Part** environment, you can create a new coordinate system by specifying its position in the graphics area. To do so, choose the **Coordinate System** tool in the **Planes** group; a coordinate system will be attached to the cursor and you will be prompted to specify the origin of the new coordinate system. Next, click in the graphics area; the new coordinate system will be placed in the graphics window. You can also snap to the keypoints in the model space to place a coordinate system.

USING THE OTHER OPTIONS OF THE EXTRUDE TOOL

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

Side Step

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 5-16 shows an open sketch and the side on which the material will be added and Figure 5-17 shows the model after creating the protrusion feature using the open sketch.

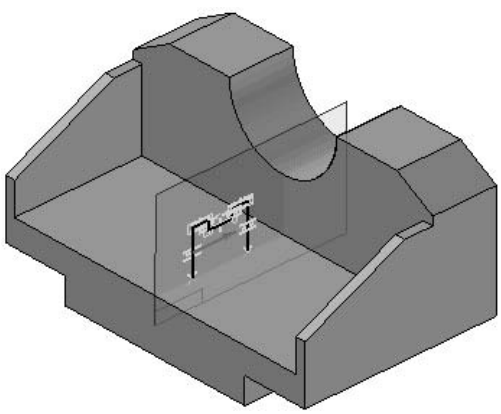


Figure 5-16 *Open sketch and the side on which the material will be added*

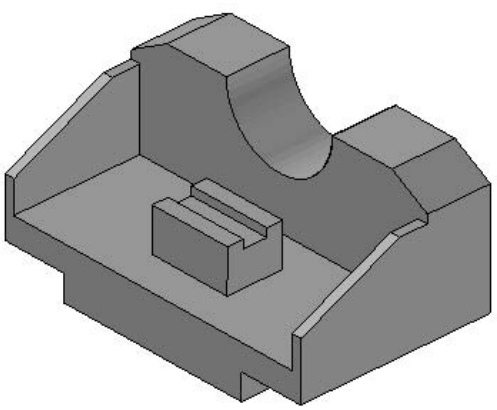


Figure 5-17 *Resulting feature*

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Extent Type

In the previous chapter, you learned about the use of the **Finite** and **From-To** options of the **Extent Type** flyout while creating a extruded feature. There are some more options available in the command bar that enables you to define the depth of the extrusion. These options become available while creating the second feature. 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. On selecting this option, an **Extrude** handle will be displayed on the sketch and you will be prompted to click to select the side. Click on the Extrude Handle and 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 5-18 shows a sketch extruded in the downward direction 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. On selecting this option, you will be prompted to define the side of extrusion of the sketch. Figure 5-19 shows a sketch extruded in the downward direction up to the next surface.

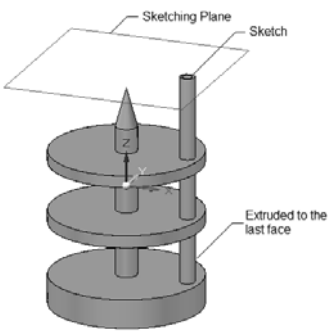


Figure 5-18 *Extruding the sketch downward using the* ***Through All*** *option*

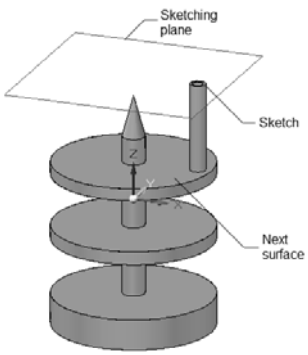


Figure 5-19 *Extruding the sketch using the* ***Through Next*** *option*

While using the **From/To Extent** option in the **Ordered Part** environment, you can also define the offset values from the **From** surface and the **To** surface by entering the value in the **Offset** edit box before selecting the respective surface. For example, if you want to use this option with an offset, choose the **From/To Extent** button from the command 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. After selecting this side, 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 extruded feature created by defining the offset values for the 'From' and 'To' surfaces.

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Note that you can also use the **Through All**, **Through Next**, and **From-To Extent** options to extrude the sketch on both sides. 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**.

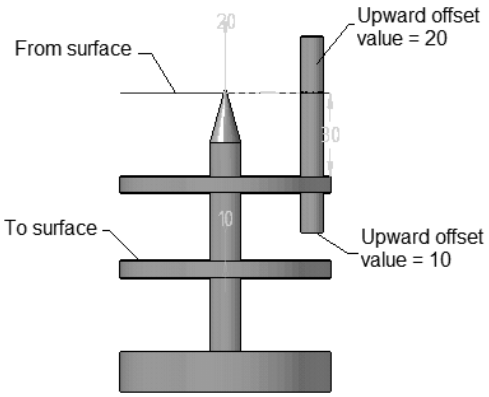


Figure 5-20 *Creating a protrusion feature by defining offset values for the* ***From*** *and* ***To*** *surfaces*

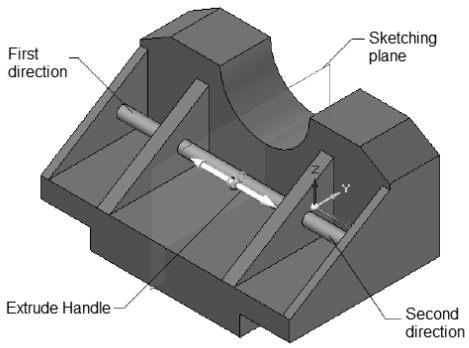


Figure 5-21 *Sketch extruded using the* ***Through All*** *option on both sides of the sketching plane in combination with the* ***Non-symmetric Extent*** *option*

Treatments drop-down

You can add a draft or a crown to a protrusion feature by choosing the required option from the **Treatments** drop-down in the **Extrude QuickBar**. The draft or crown features are added to improve the design aesthetically and to remove the model from its mold easily. By adding a draft, you can add a linear taper to the model, as shown in Figure 5-22. This figure shows different drafts applied to both sides of a non-symmetric 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 non-symmetric feature. Note that in both cases, the basic sketch is a circle.

The options in the **Treatments** drop-down are discussed next.

No Treatment



This option is chosen by default in the **Treatments** drop-down. As a result, no treatment is applied to the model.

Draft



The **Draft** tool is chosen to add a draft to the extruded feature. When you choose this button, the **Angle 1** edit box and the **Flip 1** button will be displayed in the command bar. You can enter the draft angle in the **Angle 1** 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. In case of symmetric extend extrude features, the **Angle 1**, **Angle 2**, **Flip 1**, and **Flip 2** edit boxes are displayed in the command bar.

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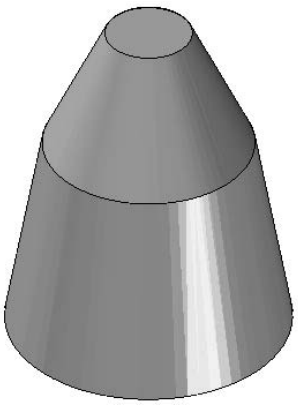


Figure 5-22 *Different drafts added to both sides of a protrusion feature*

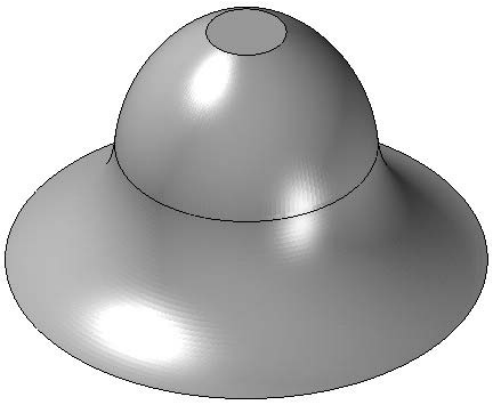


Figure 5-23 *Different crowns added to both sides of a protrusion feature*

Crown



The **Crown** button is chosen to add a crown to the extruded feature. When you choose this button, the **Crown Parameters** dialog box will be displayed. If you extrude the sketch symmetrically or non-symmetrically on both sides of the sketching plane, the dialog box will be displayed with the **Direction 1** and **Direction 2** areas, as shown in Figure 5-24. However, if you extrude the sketch only in one direction, only the **Direction 1** area of this dialog box will be active. 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.

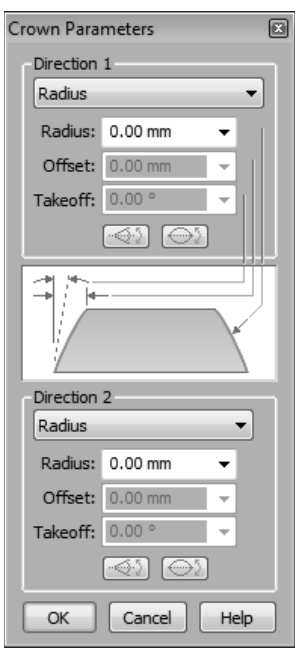


Figure 5-24 *The **Crown Parameters** dialog box*

No Crown

This option is used when you do not want to apply the crown in any 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 take off angle. 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 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 take off angle and offset value between the sections at the start and end of the crown. 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 option.

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 option.

Takeoff
The **Takeoff** edit box is used to specify the angle 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.

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 the later chapters.

Creating Extruded Cutouts (Ordered)

Ribbon: Home > Solids > Cut



Extruded cutouts are created by extruding a profile to remove the material from one or more features. Figure 5-25 shows the base feature and the sketch that will be used to create a cutout and Figure 5-26 shows the model after creating the cutout.

In Solid Edge, the cutouts are created using the **Cut** tool. This tool works in the same manner as the **Extrude** tool. Note that while creating the cutouts, you can create close profiles and use the **Side Step** extensively to define the direction of the material removal. Figures 5-27 through 5-30 show the side of the material removal and the resulting features.

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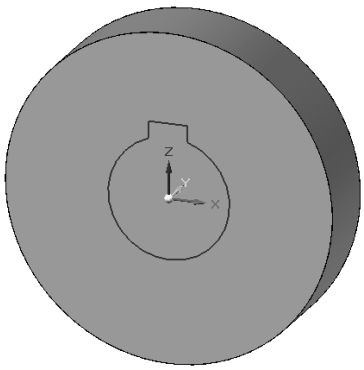


Figure 5-25 *Base feature and the sketch for the cutout*

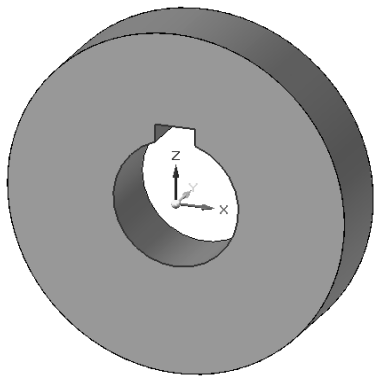


Figure 5-26 *Model after creating the cutout*

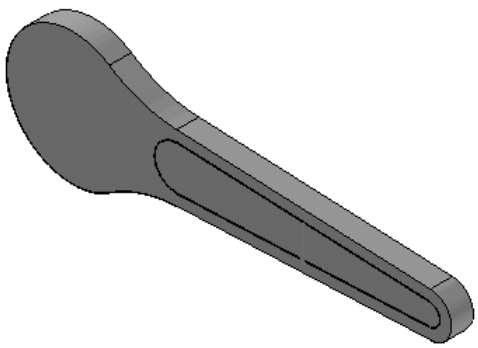


Figure 5-27 *Sketch for the cutout and the direction for the cutout pointing inside the sketch*

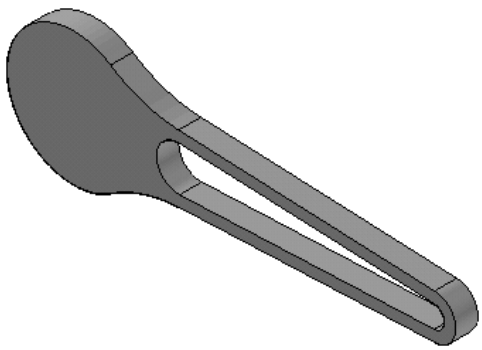


Figure 5-28 *Cutout created by removing the material from inside the sketch*

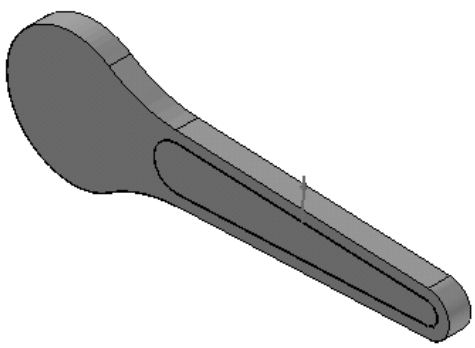


Figure 5-29 *Sketch for the cutout and the direction for the cutout pointing outside the sketch*

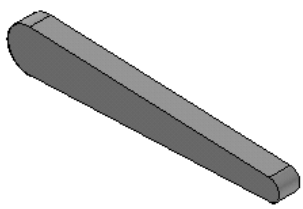


Figure 5-30 *Cutout created by removing the material from outside the sketch*

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 5-31 through 5-34.

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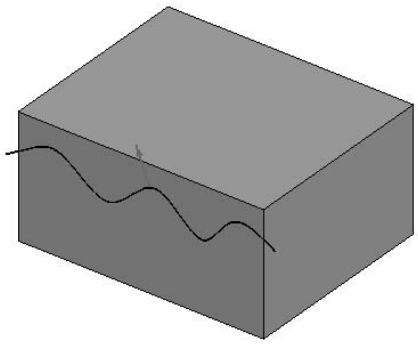


Figure 5-31 *Open profile and the side of material removal*

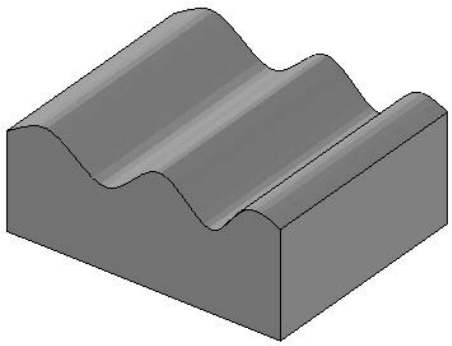


Figure 5-32 *Resulting cutout created by removing the material above the open profile*

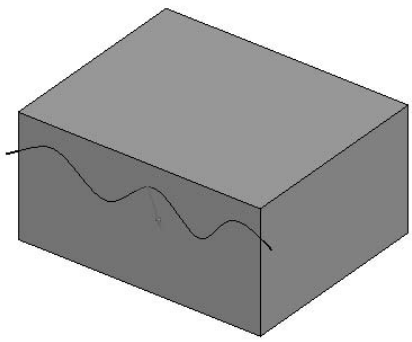


Figure 5-33 *Open profile and the side of material removal*

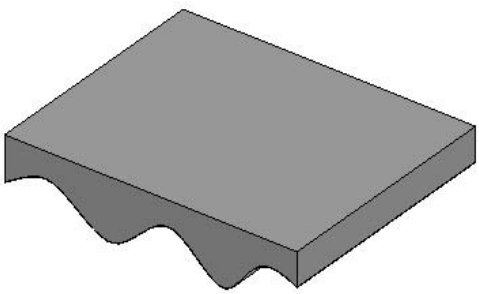


Figure 5-34 *Resulting cutout created by removing the material below the open profile*

Creating Revolved Cutouts (Ordered)

Ribbon: Home > Solids > Revolved Cut



The **Revolved Cut** tool is used to create a revolved cutout by removing the material defined by the sketch. This tool will be active only when a feature exits in the modeling window. This tool works in the same manner as the **Revolve** tool with the only difference being that the **Revolved Cut** tool removes the material only 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 5-35 through 5-38 show the side of material removal and the resulting models after creating a symmetric semicircular revolved cutout.

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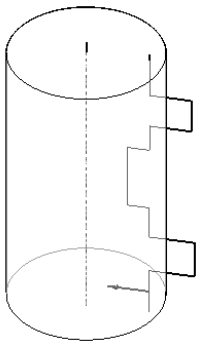


Figure 5-35 *Open profile and the side of the material removal*

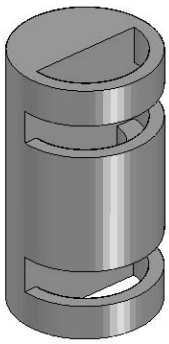


Figure 5-36 *Resulting revolved cutout created by removing the material on the left of the profile*

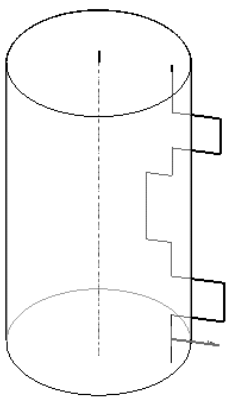


Figure 5-37 *Open profile and the side of the material removal*

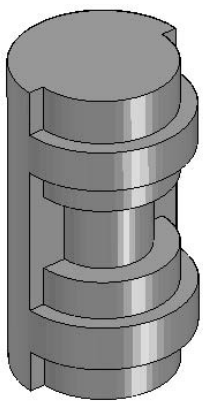


Figure 5-38 *Resulting revolved cutout created by removing the material on the right of the profile*

Creating Cutouts (Synchronous)

Just as adding material to a solid by using the Extrude and Revolve handles, you can also remove material from a solid. To do so, first draw a profile at the required position and then select the profile; an Extrude handle and a **QuickBar** will be displayed. Click on the handle; the preview of extrusion will be displayed by default. But to create a cut, you need to choose **Cut** from the **Add/Cut** flyout in the **QuickBar**. Next, you can specify the extent type, symmetry, draft and crown parameters for the cut in the same way as you did while extruding the profile. Similarly, you can create revolved cutouts using the revolve handle.

Open/Closed sketch

This option is used to specify whether the edges of an adjacent model are considered as a part of the sketch region.

Open

When you choose this option, the edges of the adjacent model are ignored. Figure 5-39 shows the cutout created by using the **Open** sketch option.

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Closed

When you choose this option, the edges of the adjacent model are considered as a part of the profile. Figure 5-40 shows the cutout created by using the **Closed** sketch option.

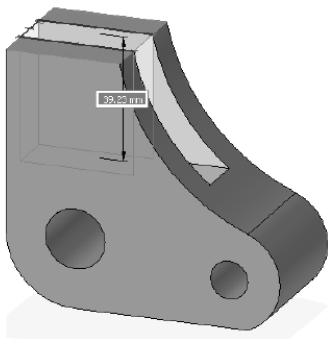


*Figure 5-39 Cutout created by using **Open** sketch option*

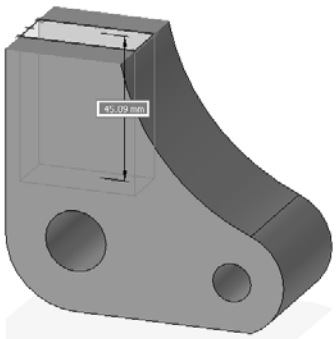


*Figure 5-40 Cutout created by using **Closed** sketch option*

USING THE EDGES OF EXISTING FEATURES

Ribbon: Home > Draw > Project to Sketch

Sometimes, while drawing the profiles of some features, you may need to use the edges of the existing features as the sketched entities in the current profile. In Solid Edge, you can do this using the **Project to Sketch** tool after selecting a sketching plane. 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 entities in the model, invoke **Project to Sketch** tool from the **Draw** group; the **Project to Sketch** command bar will be displayed. You can set project options by choosing the **Project options** button. On choosing this button, the **Project to Sketch Options** dialog box will be displayed, as shown in the Figure 5-41. The options in the dialog box are discussed next.

Project with offset

This check box is used to project the geometries with some offset value. If this check box is selected, the options to offset the projected geometry will be displayed in the command bar.

Project 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.

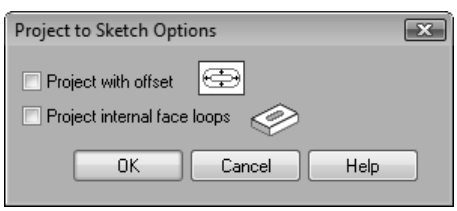


*Figure 5-41 The **Project to Sketch Options** dialog box*

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After setting the options in this dialog box, choose the **OK** button to display the **Project to Sketch** command bar. If you have selected the **Project with offset** check box, this tool works in one or two steps, which are discussed next.

Select

By default, this button is chosen in the **Project to Sketch** command bar. As a result, you can select the geometries that you want to include in the profile. Also, you can select an option from the **Select** drop-down list based on which the geometries will be included in the profile. Select the required option to select the corresponding entities. You need to choose the **Accept** button after selecting the entities, if you have selected the option to project the geometries with an offset. Else, the **Accept** button does not get activated and the selecting entities get included. 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 that allows you to project the geometries with an offset. In this case, select the geometries using the **Select** tool and then choose the **Accept** button from the command bar; the **Offset** step will be invoked and the **Distance** edit box will be displayed in the command 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 5-42 shows a model after projecting some of the geometries from the existing features on a reference plane created at an offset from the top face of the model.

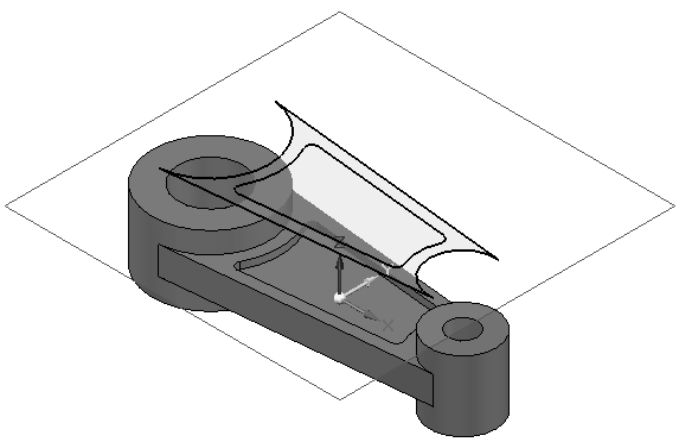


Figure 5-42 *Geometries projected on a reference plane*

PROJECTING EDGES (ORDERED)

Ribbon: Home > Draw > Include

To project geometries onto a sketching plane, choose the **Include** tool from the **Draw** group; the **Include Options** dialog box will be displayed, as shown in Figure 5-43. Next, in this dialog box, set the options to project the geometries onto the sketching plane.

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The options in **Include Options** dialog box are discussed next.

The **Include with offset** and **Include internal face loops** are same as the **Project with offset** and **Project internal face loops** options discussed earlier.



Figure 5-43 *The* ***Include Options*** *dialog box*

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.

ADVANCED DRAWING DISPLAY TOOLS

In the 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.

Creating User-defined Named Views

As mentioned earlier, the **View Orientation** palette in the status bar is used to invoke the standard named views. In Solid Edge, you can also create user-defined standard views. These views are automatically added to the **View Orientation** palette, and therefore, can be invoked whenever required.

To create the user-defined named views, set the current view to the view to be saved using the **Rotate** tool or any other drawing display tool. After setting the view, choose **View Orientation > View Manager** from the status bar; the **Named 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 of the **Named Views** dialog box. You can also enter a brief description about the view in the **Description** column. After entering the required information, close the dialog box; the newly created view will be displayed in the **View Orientation** palette and can be selected from this palette.

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Using Common Views

Ribbon: View > Orient > Common Views



Solid Edge provides you with the **Common Views** tool, which is a very user-friendly tool and is used to set the current view to some standard common views. You can invoke this tool from the **Ribbon** or by right-clicking in the drawing window when the **Select** tool is active. Then, choose **Common Views** from the shortcut menu displayed; the **Common Views** dialog box will be displayed with a cube in it. You can use this cube to rotate the view of the model by clicking on the corners or faces of the cube. As you move the cursor over any vertex or face of the cube in the **Common Views** dialog box, a message will be displayed in this dialog box that will inform you about the direction of rotation of the view. 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 — Synchronous

In this tutorial, you will create the model shown in Figure 5-44. The dimensions of the model are given in Figure 5-45. After creating the model, save it with the name *c05tut1.par* at the following location:

\Solid Edge\c05 **(Expected time: 45 min)**

Before you create the model, first you 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 the combination of three extruded features and three cutouts (holes). The base feature will be created on the YZ plane. The second feature will be created on the top face of the base feature. The third extruded feature will be created on the reference plane created 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.

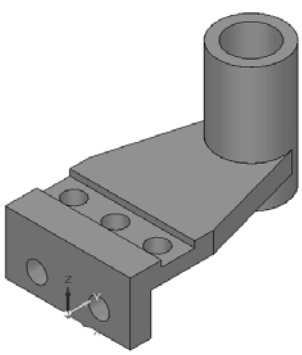


Figure 5-44 *Model for Tutorial 1*

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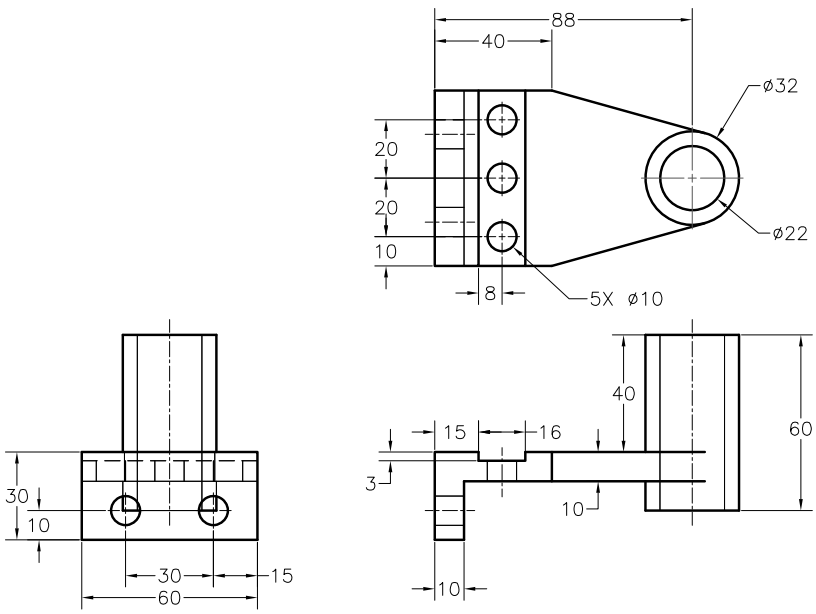


Figure 5-45 *Dimensions of the model*

The following steps are required to complete this tutorial:

a. Create the base feature on the right plane, refer to Figures 5-46 and 5-47.
b. Select the top planar face of the base feature as the sketching plane and then create the second feature, refer to Figures 5-48 through 5-51.
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 5-52.
d. Create two holes on the left face of the base feature using the **Cut** tool, refer to Figure 5-54.
e. Create the remaining cutouts to complete the model, refer to Figure 5-55.

Creating the Base Feature

As mentioned earlier, the base feature is an extruded feature and the profile of this feature will be created on the right plane.

1. Start a new part file and then choose the **Line** tool from the **Draw** group of the **Home** tab; the **Line** command bar is displayed and the alignment lines are attached to the cursor.

   Next, you need to define the YZ plane to draw the profile of the base feature.

2. As you move the cursor toward the base coordinate system, the nearest principal plane gets highlighted. Wait for a while and then right-click; the **QuickPick** list box is displayed with the nearest possible selection. Next, select the YZ plane to sketch.

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3. Change the orientation of the view normal to the sketching plane by using the **Sketch View** tool from the **Status Bar**.

4. Draw the profile for the base feature and then add the required relations and dimensions by using the tools from the **Sketching** tab, as shown in Figure 5-46. Change the orientation of the view to isometric.

5. Choose the **Select** tool from the **Home** tab and select the middle portion of the sketch; an extrude handle and a **QuickBar** are displayed.

6. Choose the **Symmetric** button from the **QuickBar** and drag the arrow in the drawing window; the preview of extrusion and an edit box are displayed in the drawing window.

7. Enter **60** in the edit box and press ENTER; an extruded feature is created. Figure 5-47 shows the base feature of the model after hiding the dimensions.

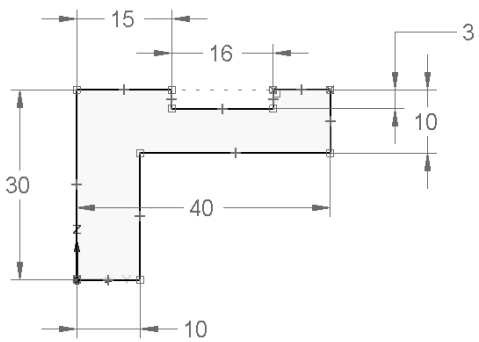


Figure 5-46 *Dimensioned profile for the base feature*

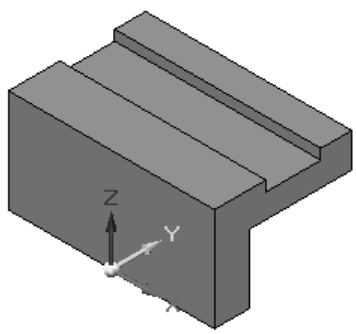


Figure 5-47 *Base feature of the model*

Creating the Second Feature

As mentioned earlier, the second feature is also an extruded feature and its profile is created on the top face of the base feature.

1. Choose the **Circle by Center Point** tool from the **Draw** group; you are prompted to select a planar face or the principal plane for sketching the profile.

2. Move the cursor on the top face of the base feature and click on the lock symbol, refer to Figure 5-48.

3. Draw the profile for the second feature by using the sketching tools and then add the required dimensions and relationships to the profile, as shown in Figure 5-49. Note that you do not need to draw the horizontal line to close the sketch.

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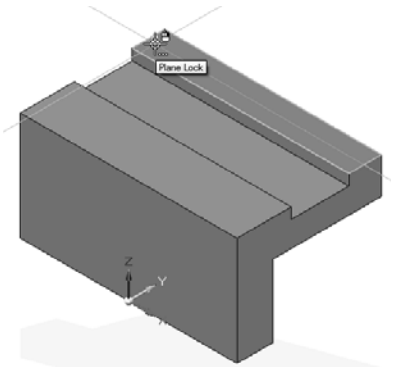


Figure 5-48 *Specifying the sketching plane for the second feature*

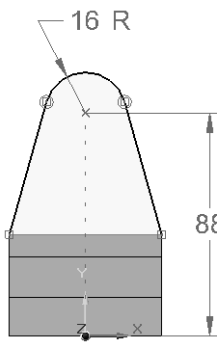


Figure 5-49 *Open profile for the second feature*

4. Click on the lock symbol on the right of the graphics window; the sketching plane is unlocked. Next, orient the view to isometric.

5. Invoke the **Select** tool and then click inside the closed region; an extrude handle is displayed.

   You need to add material inside the sketch region. Therefore, you need to click on the down arrow of the extrude handle.

6. Choose the **From-To** option from the **Extent Type** drop-down in the **QuickBar**; you are prompted to select the **To** surface because the profile plane is automatically selected as the **From** surface.

7. Place the mouse on the right face for few seconds; the mouse symbol is attached to the cursor. Next, right-click to invoke the **QuickPick** list box. In this list box, select the bottom face of the base feature, as shown in Figure 5-50.

   The model after creating the second feature is shown in Figure 5-51.

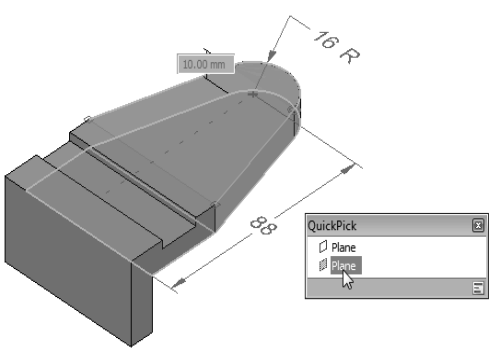


Figure 5-50 *Preview of the second feature*

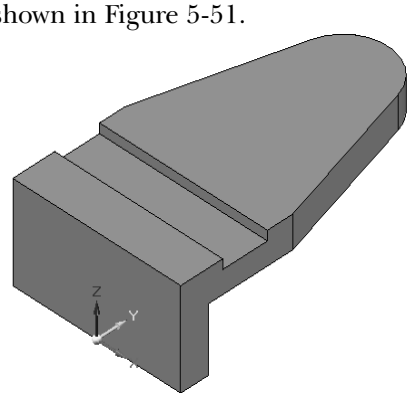


Figure 5-51 *Model after creating the second feature*

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Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

Creating the Third Feature

The third feature is also an extruded feature. Its profile can be created on the top face of the second feature and then extruded along both sides of the top face. However in this tutorial, you will create a profile on a reference plane at an offset of 10 units from the bottom face of the second feature.

1. Choose the **Coincident Plane** tool from the **Planes** group; you are prompted to click on the planar face. Rotate the model and select the bottom face of the second feature; a coincident plane is created on the bottom face of the second feature. Also, a steering wheel is displayed on the newly created plane.

2. Select the primary axis of the steering wheel. Next, drag the cursor downward and then enter **10** in the edit box; a parallel plane is created at an offset from the bottom face of the second feature.

3. Draw a circle of diameter 32 units on the new reference plane. The circle should be concentric with the arc in the second feature.

4. Choose the **Select** tool from the **Home** tab and select the middle portion of the sketch; a steering wheel and an **Extrude QuickBar** are displayed.

5. Make sure the **Finite** option is chosen in the **Extent Type** drop-down, and then drag the extrude handle upward; the preview of extrusion and an edit box are displayed in the drawing window.

6. Enter **60** in the edit box and press ENTER. The model after creating the third protrusion feature is shown in Figure 5-52.

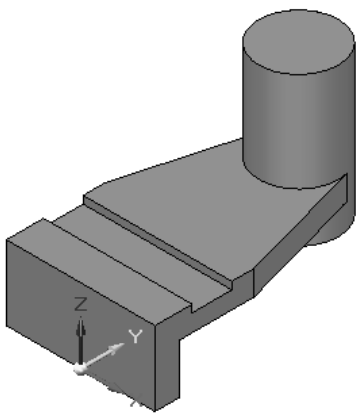


Figure 5-52 *Model after creating the third feature*

Creating a Cutout on the Left Face of the Base Feature

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

1. Choose the **Circle by Center Point** tool from the **Draw** group; you are prompted to click on the planar face.

2. Select the left face of the base feature to define a sketching plane.

3. Draw the profile for the cutout. The profile consists of two circles, each with a diameter of 10 mm, refer to Figure 5-53.

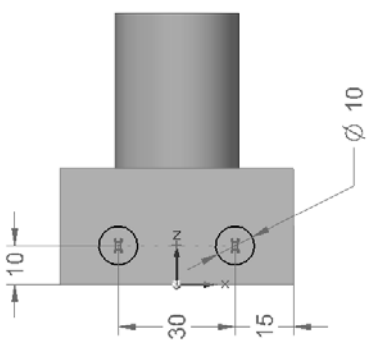


Figure 5-53 *Profile for the cutout*

4. Choose the **Extrude** tool from the **Solids** group; the **Extrude** command bar is displayed and you are prompted to select the sketch. Next, select the circles and choose the **Through Next** option from the **Extent Type** drop-down in the command bar; the **Add/Cut** flyout is activated.

5. Choose the **Remove** material option from the **Add/Cut** flyout.

6. Move the cursor to the right of the model and right-click to specify the direction of the cutout feature creation. The model after creating the cutout feature is shown in Figure 5-54.

Creating the Remaining Features

1. Create the remaining two cutouts. The model after creating all features is shown in Figure 5-55.

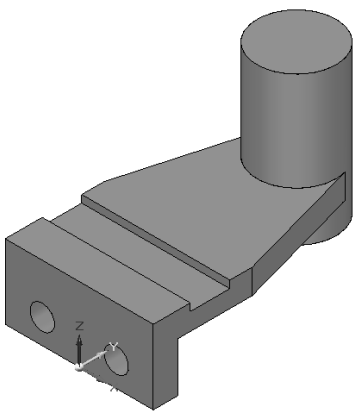


Figure 5-54 *Model after creating the cutout feature*

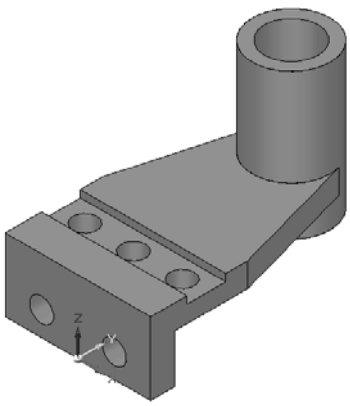


Figure 5-55 *Final model for Tutorial 1*

Saving the Model

1. Save the model with the name *c05tut1.par* at the location given below and then close the file.

\Solid Edge\c05

Tutorial 2 Ordered

In this tutorial, you will create the model shown in Figure 5-56 in the **Ordered Part** environment. The dimensions of the model are given in Figure 5-57. After creating the model, save it with the name *c05tut2.par* at the location given below:

\Solid Edge\c05 **(Expected time: 45 min)**

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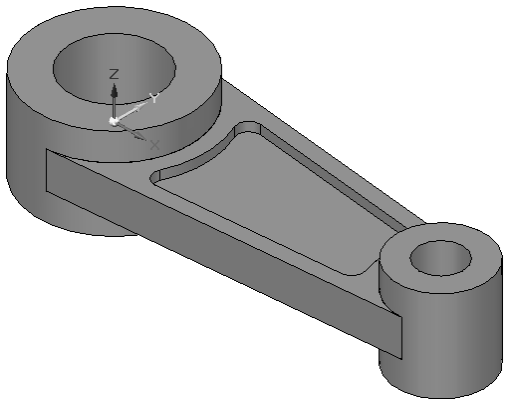


Figure 5-56 *Model for Tutorial 2*

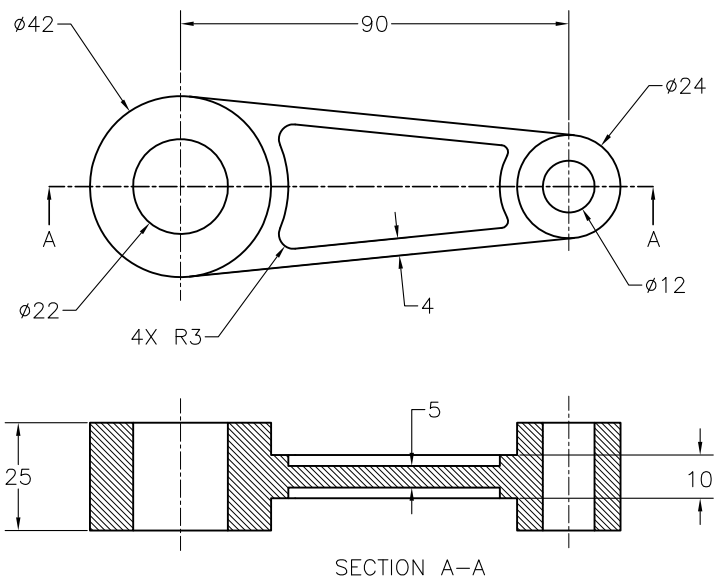


Figure 5-57 *Dimensions of the model for Tutorial 2*

The model for Tutorial 2 can be created with various combinations of features. However, in this tutorial, the model will be created by using the combination of three extruded 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 then extruded symmetrically. Next, you will create a sketch on the top plane to create a cutout and then an extruded feature.

The following steps are required to complete this tutorial:

a. Start a new part file and switch to the **Ordered Part** environment.

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b. Create the base feature on the top plane, refer to Figure 5-59.
c. Create the second feature on the top plane, refer to Figure 5-61.
d. Project the edges of the base feature, refer to Figure 5-62.
e. Create a cutout in the second feature by using the sketch, refer to Figure 5-63.
f. Create an extruded feature by using the same sketch to complete the model, refer to Figure 5-64.

Creating the Base Feature

1. Start a new part file and then select the **Ordered** radio button from the **Model** group of the **Tools** tab; the **Ordered Part** environment is invoked.

2. Choose the **Sketch** tool from the **Home** tab; you are prompted to click on the planar face or the reference plane.

3. Turn on the visibility of the Base Reference Planes by selecting the check box adjacent to it and then select the top plane as the sketching plane for creating the extrude feature.

4. Draw the profile for the base feature by using the sketching tools and then add the required relationships and dimensions to the profile, as shown in Figure 5-58.

5. Exit the sketching environment and then extrude the sketch symmetrically through a distance of 10 units. The base feature of the model is shown in Figure 5-59.

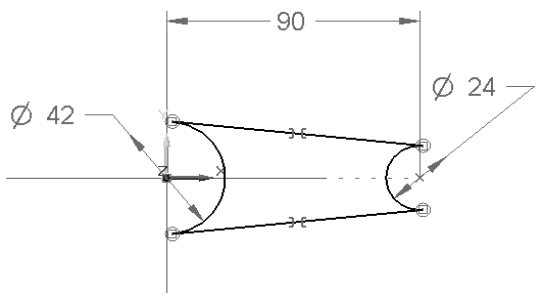


Figure 5-58 *Profile for the base feature*

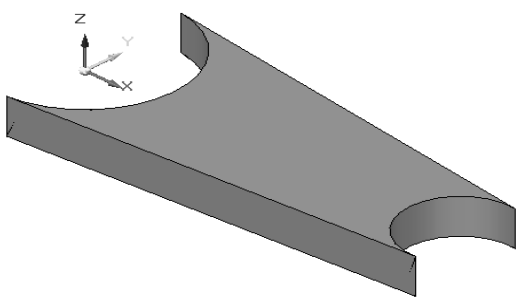


Figure 5-59 *Base feature of the model*

Creating the Second Feature

The second feature will also be created on the top plane.

1. Invoke the **Sketch** tool and select the top plane as the sketching plane. Draw the profile for the second feature by using the sketching tools and then add the required relationships and dimensions to the sketch, as shown in Figure 5-60.

2. Exit the sketching environment and then extrude the sketch symmetrically through a distance of 25 units. The model after creating this feature is shown in Figure 5-61.

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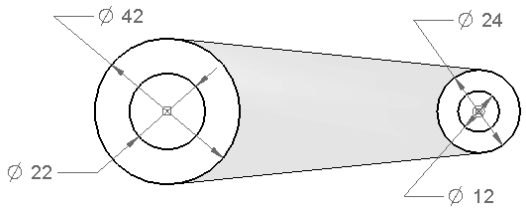


Figure 5-60 *Profile for the second feature*

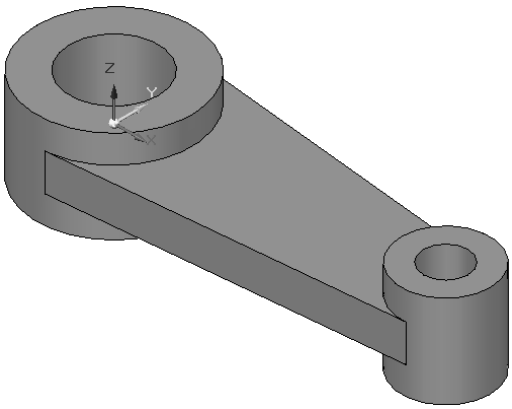


Figure 5-61 *Model after creating the second feature*

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 needs to be created on the top plane. To create this sketch, you need to project the edges of the second feature to an offset of 4 units.

1. Invoke the **Sketch** tool and select the top plane as the sketching plane.

2. Choose the **Include** tool from the **Draw** group; the **Include Options** dialog box is displayed.

3. Select the **Include with offset** check box and make sure that the **Include internal face loops** check box is clear.

4. Choose **OK** to exit the dialog box and then select **Loop** from the **Select** drop-down list in the **Include** command bar.

5. Move the cursor close to the first feature and click when the complete loop, defined to create the first feature, is highlighted; the complete loop is projected.

6. Right-click to proceed to the **Offset** step. Enter **4** in the **Distance** edit box and press ENTER. Next, move the cursor to the center of the model. Click when the offset arrow points inward in the model; the outer loop is projected at an offset of 4 units.

7. Add a fillet of radius 3 units to all vertices of the projected loop.

8. Apply dimensions and constraints to the sketch to fully define it, as shown in Figure 5-62.

9. Exit the sketching environment and then the **Sketch** command bar. This sketch is termed as **Sketch 3** in the **PathFinder**.

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Creating the Cutout

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

1. Choose the **Cut** tool from the **Solids** group; you are prompted to select a sketch.

2. Select the **Select from Sketch** option from the **Create-From Options** drop-down list in the command bar.

3. Select the sketch and then right-click to proceed to the **Extent Step**.

4. Choose the **Symmetric Extent** button from the command bar and then enter **10** in the **Distance** edit box. Then, press ENTER.

5. Choose **Finish** and then **Cancel** to exit the **Cut** tool. The model after creating the cutout is shown in Figure 5-63.

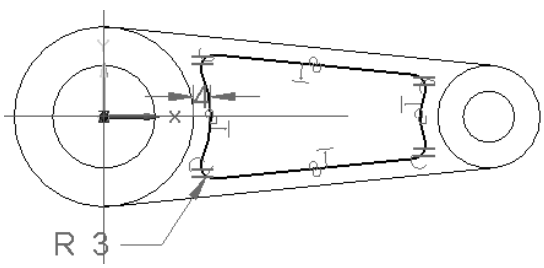


Figure 5-62 *Sketch after adding the fillet, dimensions, and constraints*

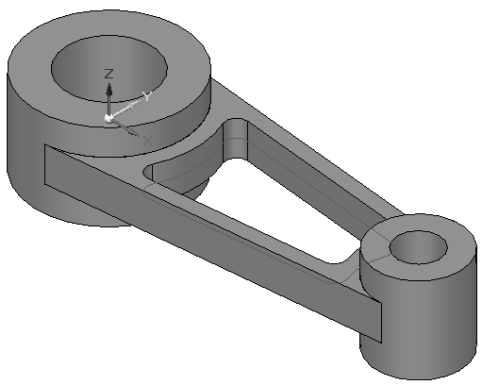


Figure 5-63 *Model after creating the cutout*

Creating the Protrusion Feature

1. Choose the **Extrude** tool from the **Solids** tool and then select the **Select from Sketch** option from the **Create-From Options** drop-down list.

2. Select **Sketch 3** and then right-click to proceed to the **Extent Step**.

3. Choose the **Symmetric Extent** button from the command bar and then enter **5** in the **Distance** edit box.

4. Choose **Finish** and then **Cancel** from the command bar to exit the **Extrude** tool.

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

5. Clear the check box beside **Sketch 3** to hide the sketch. Similarly, you can hide planes and other sketches. The final model for Tutorial 2 is shown in Figure 5-64.

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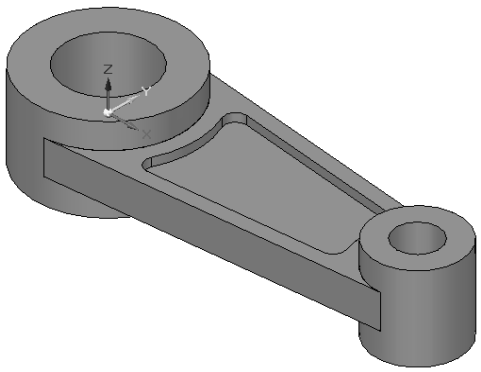


Figure 5-64 *Final model for Tutorial 2*

Saving the Model

1. Save the model with the name *c05tut2.par* at the location given below and then close the file.

   \Solid Edge\c05

Tutorial 3 Synchronous

In this tutorial, you will create the model shown in Figure 5-65. Its dimensions are given in Figure 5-66. After creating the model, save it with the name *c05tut3.par* at the location given below:

\Solid Edge\c05 **(Expected time: 45 min)**

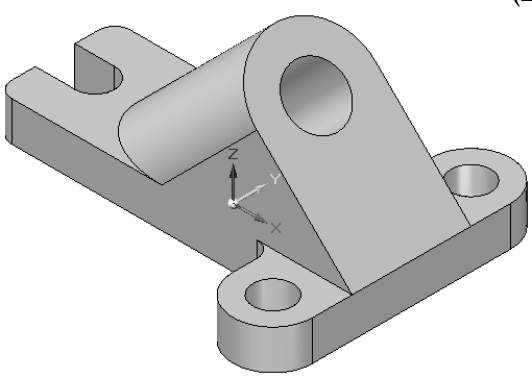


Figure 5-65 *Model for Tutorial 3*

Before you create a model, first you need to determine the number of features in it and then the sequence in which they will be created. The model for Tutorial 3 is a combination of two extruded features. The base feature will be created on the XY plane. The second feature will be created on a plane that is created at 45 degrees to the XY plane.

The following steps are required to complete this tutorial:

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a. Create the base feature with two holes on the XY plane, refer to Figure 5-68.
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 5-71.
c. Extrude the profile up to the next face to complete the feature, refer to Figure 5-72.

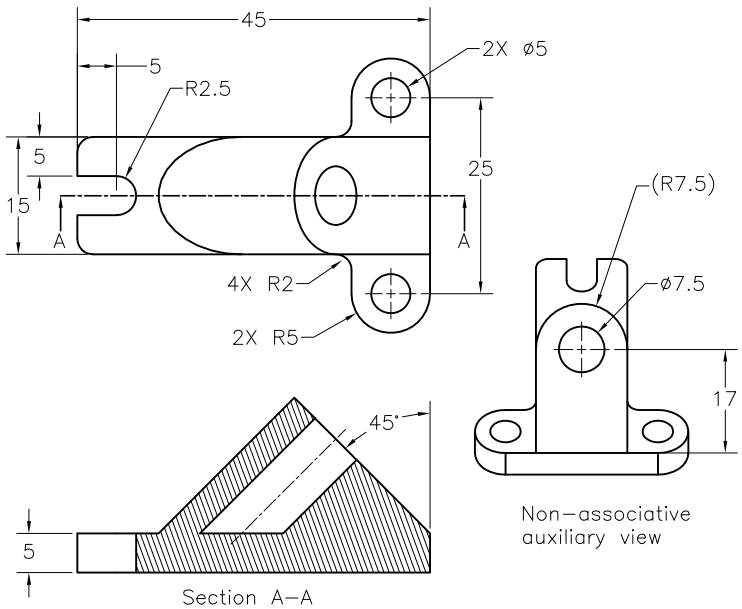


Figure 5-66 *Dimensions of the model for Tutorial 3*

Creating the Base Feature

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

1. Start a new part file and then choose the **Line** tool from the **Draw** group in the **Home** tab; the **Line** command bar is displayed and the alignment lines are attached to the cursor.

2. Select the XY plane as the sketching plane for creating the protrusion feature.

3. Change the orientation of the view normal to the sketching plane by using the **Sketch View** tool from the **Status Bar**.

4. Draw the profile for the base feature by using the sketching tools and then add the required relationships and dimensions to the sketch, as shown in Figure 5-67.

5. Choose the **Extrude** tool from the **Solids** group; the **Extrude** command bar is displayed.

6. Select the sketch and right-click in the graphics window; the preview of the extrude feature is displayed.

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7. Enter **5** in the edit box and press ENTER; an extruded feature is created, as shown in Figure 5-68.

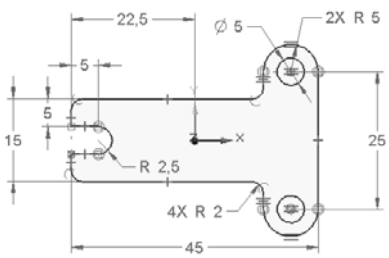


Figure 5-67 *Dimensioned profile of the base feature*

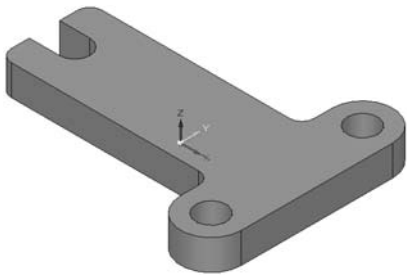


Figure 5-68 *Base feature of the model*

Creating the Second Feature

The second feature needs to be created by using an angled reference plane, which in turn will be created by using the right edge of the top face of the base feature.

1. Choose the **Coincident Plane** tool from the **Planes** group; you are prompted to click on the planar face. Rotate the model and select the bottom face of the second feature; a coincident plane is created on the bottom face of the second feature. Also, a steering wheel is displayed on the newly created plane.

2. Press and hold the Origin knob of the steering wheel and move the steering wheel toward the top edge of the face, and then place it, as shown in Figure 5-69.

3. Click on the torus of the steering wheel to rotate the plane. Next, specify the angle by entering **45** in the edit box. After specifying the value, an angled plane is created.

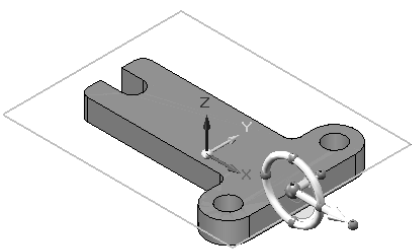


Figure 5-69 *Positioning the steering wheel to create an angled plane*

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

4. Choose the **Project to Sketch** tool from the **Draw** group and select the angled plane; the **Project to Sketch** command bar is displayed.

5. Select the **Project Options** button in the command bar; the **Project to Sketch Options** dialog box is displayed. Make sure the **Project with offset** and **Project internal face loops** check boxes are cleared in this dialog box. Next, choose **OK** in this dialog box and then select the edges to project, as shown in Figure 5-70.

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6. Trim and extend the projected entities by using the sketching tools and then draw a tangent arc to create a closed profile, as shown in Figure 5-71.

7. Add a circle to complete the profile and then add the required dimensions to both the profile and the circle, refer to Figure 5-71.

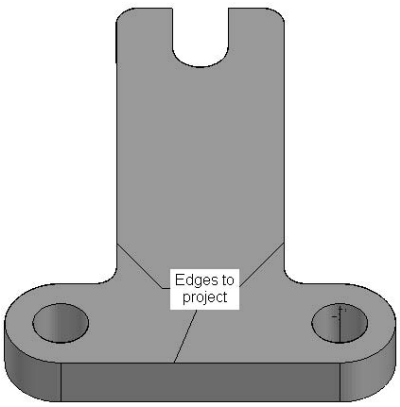


Figure 5-70 *Selecting the entities to project*

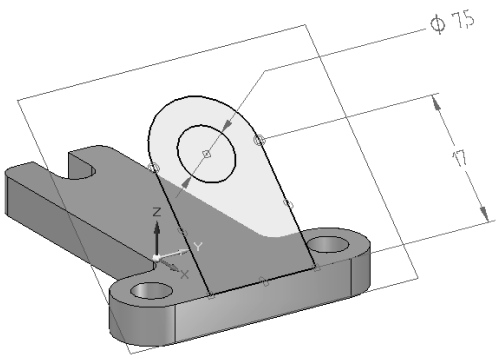


Figure 5-71 *Profile for the second feature*

8. Choose the **Select** tool from the **Home** tab and select the middle portion of the sketch; an extrude handle and a **QuickBar** are displayed.

9. Choose the **Through Next** option from the **Extent type** drop-down in the **QuickBar** and click on the downward arrow of the extrude handle; the preview of the second feature is displayed merging with the base feature.

10. Left-click in the graphics window to create the second feature of the model. The final model for Tutorial 3 is shown in Figure 5-72.

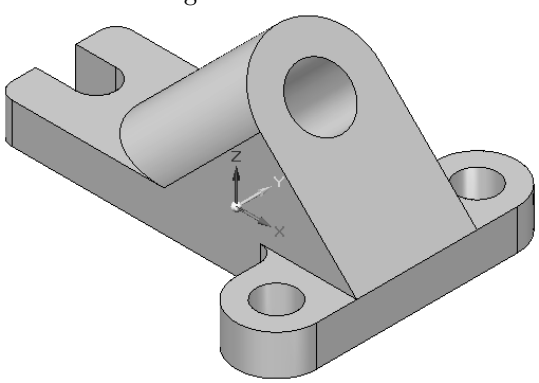


Figure 5-72 *Final model for Tutorial 3*

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Saving the Model

1. Save the model with the name *c05tut3.par* at the location given below and then close the file.

   \Solid Edge\c05



Note

*The holes shown in the model in Tutorial 3 can also be created by using the **Hole** tool. The use of this tool will be discussed in later chapters.*

Self-Evaluation Test

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

1. In the **Synchronous Part** environment, the steering wheel can be used to create parallel planes as well as angle planes. (T/F)

2. Cutouts are created by removing the material defined by a profile from one or more existing features. (T/F)

3. While creating cutouts, you can create open profiles and use the _____ extensively to define the direction of material removal.

4. The _____ tool is used to create a plane normal to a selected sketched curve or an edge of a model.

5. In Solid Edge, you can create coordinate systems by using _____ options.

6. The _____ tool is used in the **Ordered Part** environment to create a revolved cutout by removing the material defined by a sketch.

7. If a plane selected to define the coincident plane does not have a linear edge, the _____ direction is defined by using the base reference plane.

8. While creating an extruded feature, you can use the _____ to add a draft or a crown to it.

9. A coordinate system is displayed as a feature in the docking window. (T/F)

10. You can create an extruded feature at an offset distance from the sketching plane while using the **From/To Extent** option. (T/F)

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Review Questions

Answer the following questions:

1. Which of the following tools in Solid Edge is used to project the edges of a existing feature on the current sketching plane?

   (a) **Project to Sketch** (b) **Include**
   (c) **Insert** (d) both (a) and (b)

2. Which 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

3. The **Coincident Plane** tool is used to create a reference plane that is coincident to a base reference plane, another reference plane, or the planar face of a model. (T/F)

4. You cannot use open profiles to create cutouts. (T/F)

5. The **By 3 Points** tool is used to create reference planes by selecting three points. (T/F)

6. The _____ of the steering wheel is used to create a parallel plane in the **Synchronous Part** environment

7. In the **Ordered Part** environment, select the _____radio button to create a coordinate system with respect to an existing coordinate system.

8. The _____ option is used to apply the crown by defining its radius and take off angle.

9. The _____ check box needs to be selected in the **Include Options** dialog box to select geometries from other parts in an assembly.

10. The _____option in the **Create-From Options** drop-down list is used to create sketches on the plane on which the profile of the selected feature was created.

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Exercises

Exercise 1

Create the model shown in Figure 5-73. The dimensions of the model are given in the views shown in Figure 5-74. After creating the model, save it with the name *c05exr1.par* at the location given below:

\Solid Edge\c05 **(Expected time: 30 min)**

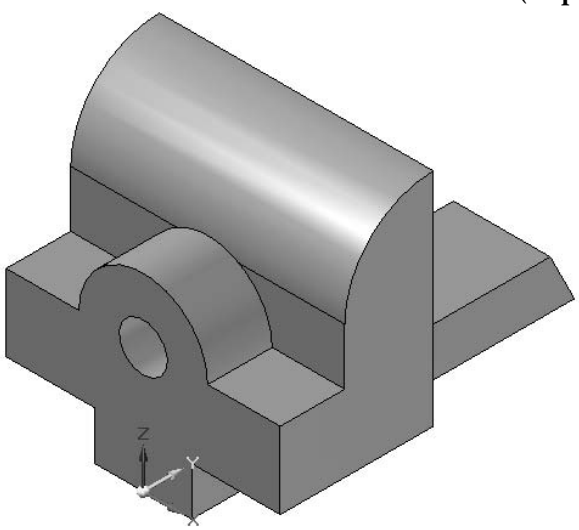


Figure 5-73 *Model for Exercise 1*

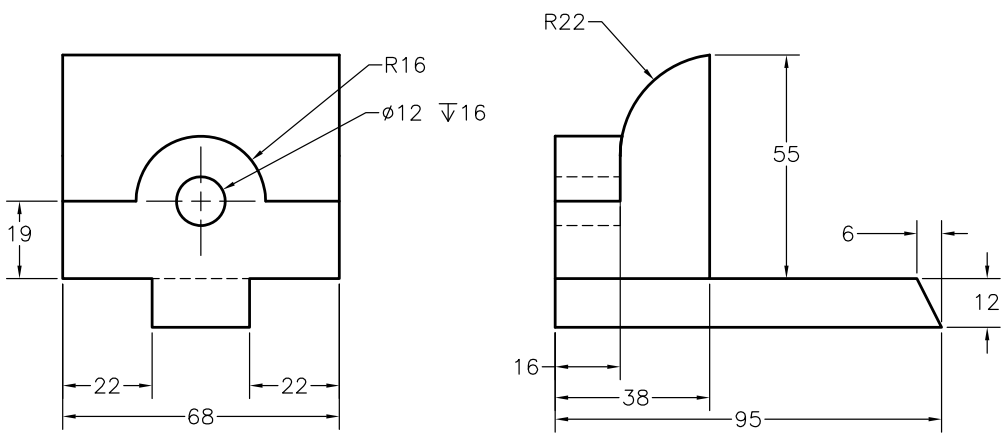


Figure 5-74 *Dimensions of the model for Exercise 1*

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Exercise 2

Create the model shown in Figure 5-75. Its dimensions are given in the views shown in Figure 5-76. After creating the model, save it with the name *c05exr2.par* at the location given below:

\Solid Edge\c05 **(Expected time: 30 min)**

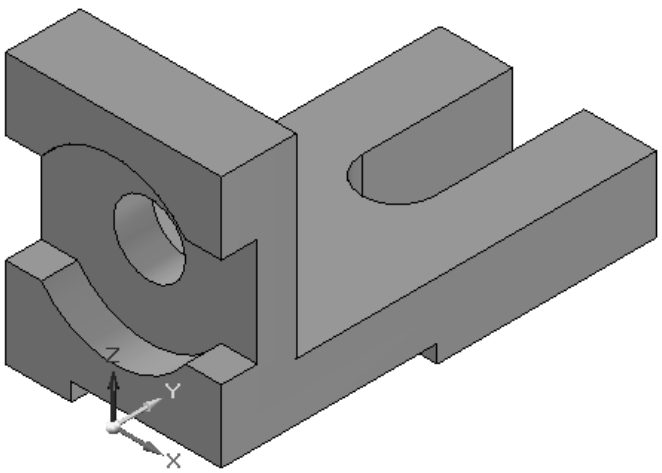


Figure 5-75 *Model for Exercise 2*

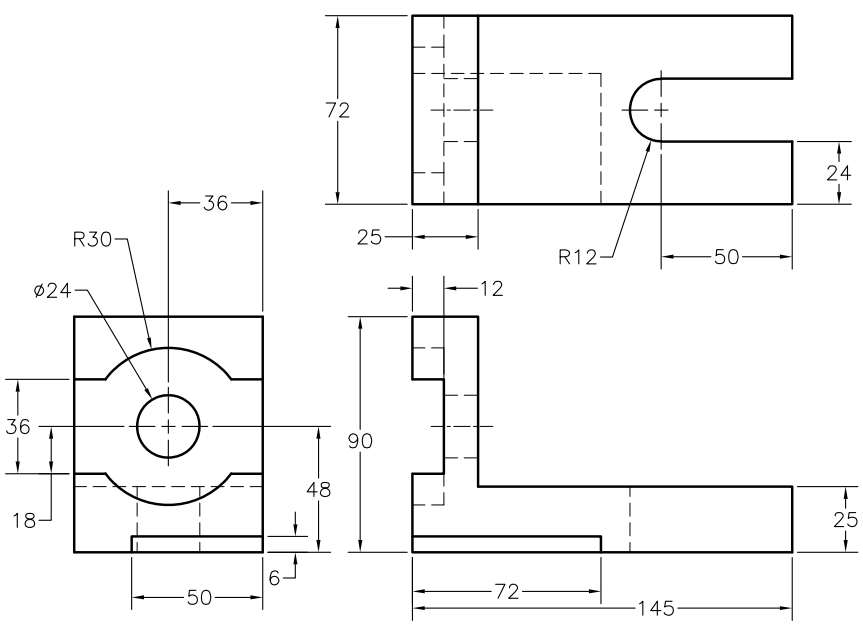


Figure 5-76 *Dimensions of the model for Exercise 2*

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Exercise 3

Create the model shown in Figure 5-77. Its dimensions are also given in the same figure.

(Expected time: 20 min)

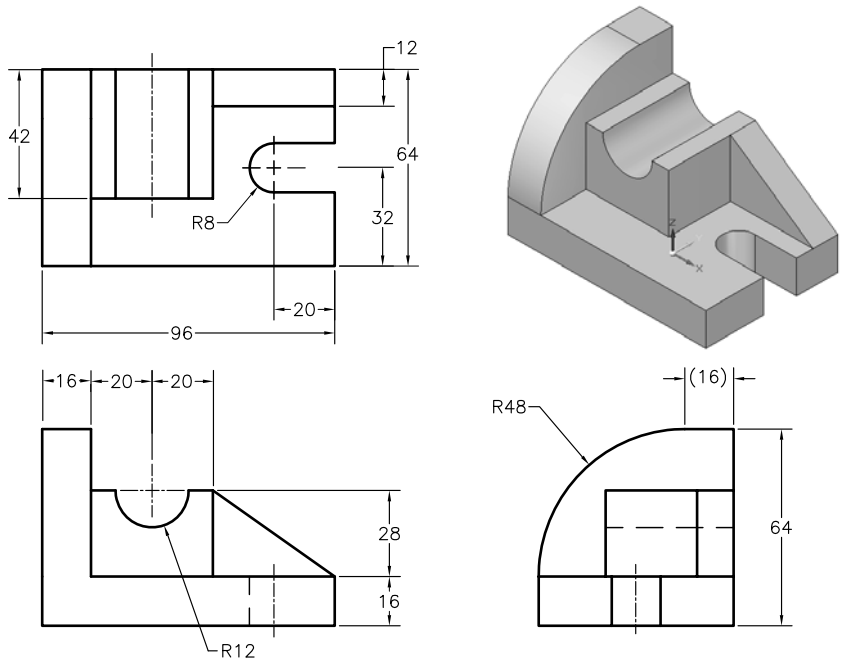


Figure 5-77 *Model and its dimensions for Exercise 3*

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Answers to Self-Evaluation Test

1. T, **2.** T, **3. Side Step**, **4. Normal To Curve**, **5.** two, **6. Revolved Cut**, **7.** X-axis, **8. Treatment Step**, **9.** T, **10.** T