CHAPTER 3

Features

Objectives

- Learn about the **Features** tools.
- Learn how to draw 3D objects.
- Learn how to use **Features** tools to create objects.
- Work with millimeter dimensions.

3-1 INTRODUCTION

This chapter introduces the **Features** tools. Several examples are included that show how to apply the tools to create objects.

3-2 EXTRUDED BOSS/BASE

The **Extruded Boss/Base** tool is used to add thickness or height to an existing 2D sketch. The examples presented use metric dimensions.

To Work with Dimensions in Millimeters

1. Click the **Tools** heading at the top of the drawing screen.

   See Figure 3-1.

2. Click the **Options** . . . tool.

   The **Document Properties** box will appear.

3. Click the **Document Properties** tab.
4. Click **Units.**
5. Click the **MMGS (millimeter, gram, second)** button.
6. Click **OK.**

   The system is now calibrated for millimeters.

To Use the Extruded Boss/Base Tool

1. Start a new drawing and draw a **60 × 100** rectangle in the top plane.

   See Figure 3-2.

2. Add the **Standard View** toolbar to your screen and click the **Isometric** icon.
3. Click the **Features** tool.
4. Click the **Extruded Boss/Base** tool.

   The **Extrude Properties Manager** will appear.

5. Define the extrusion height as **40.00mm.**

   A real-time preview will appear.

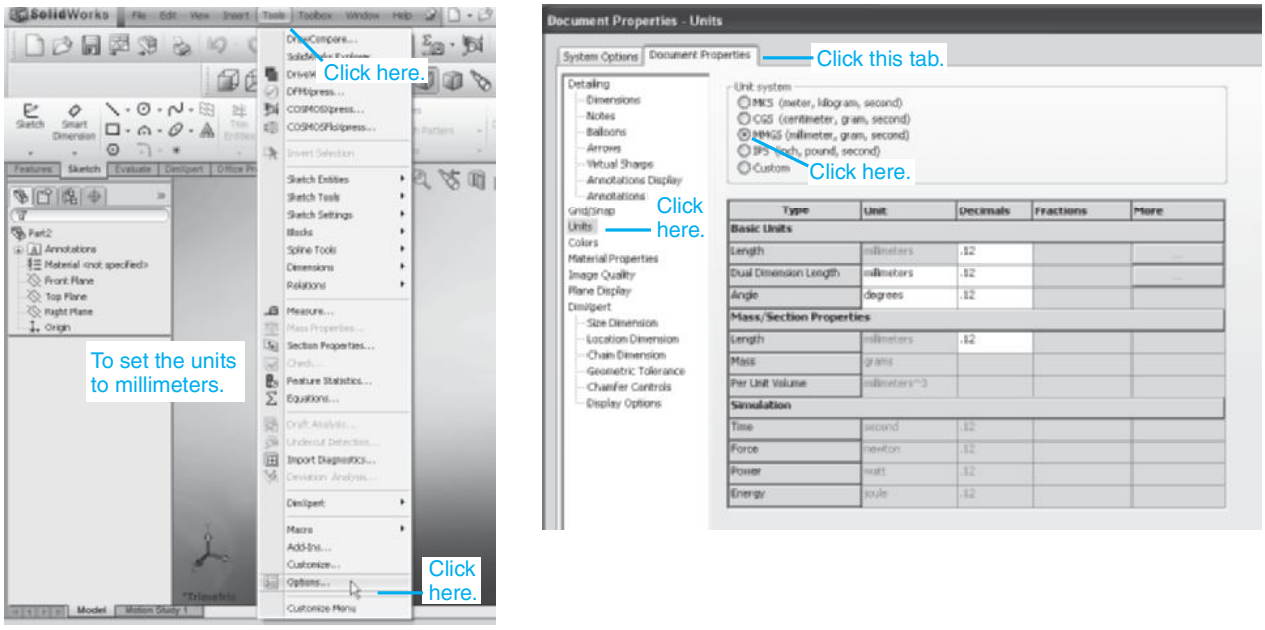


Figure 3-1

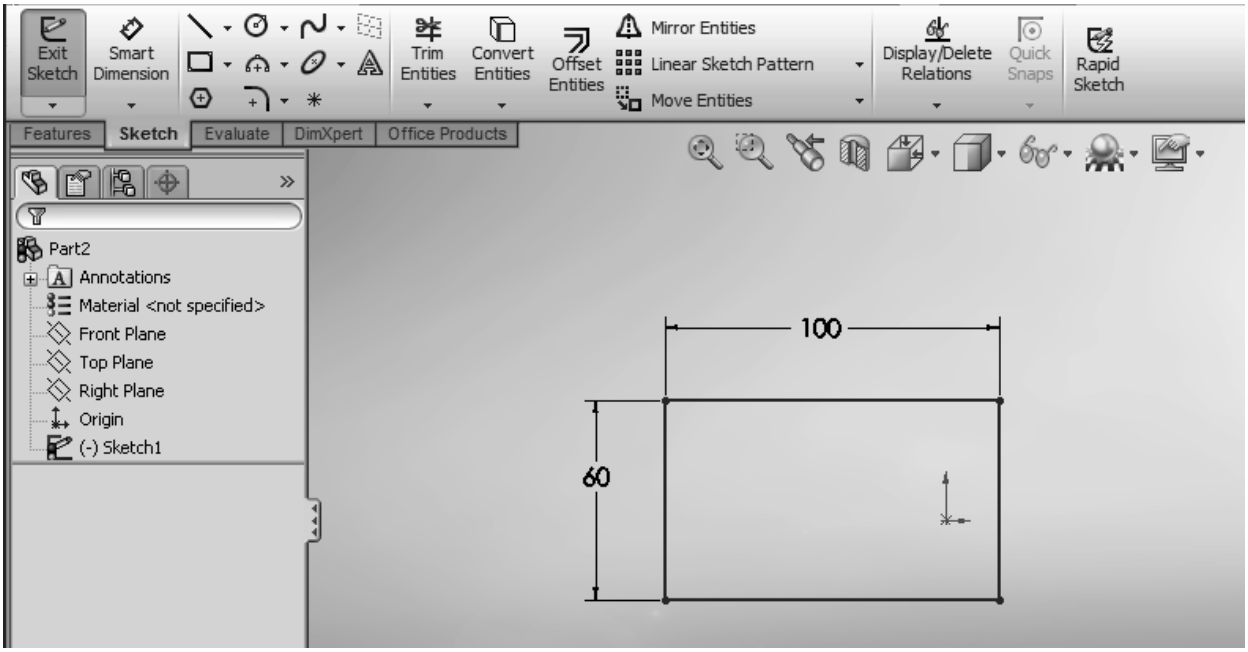
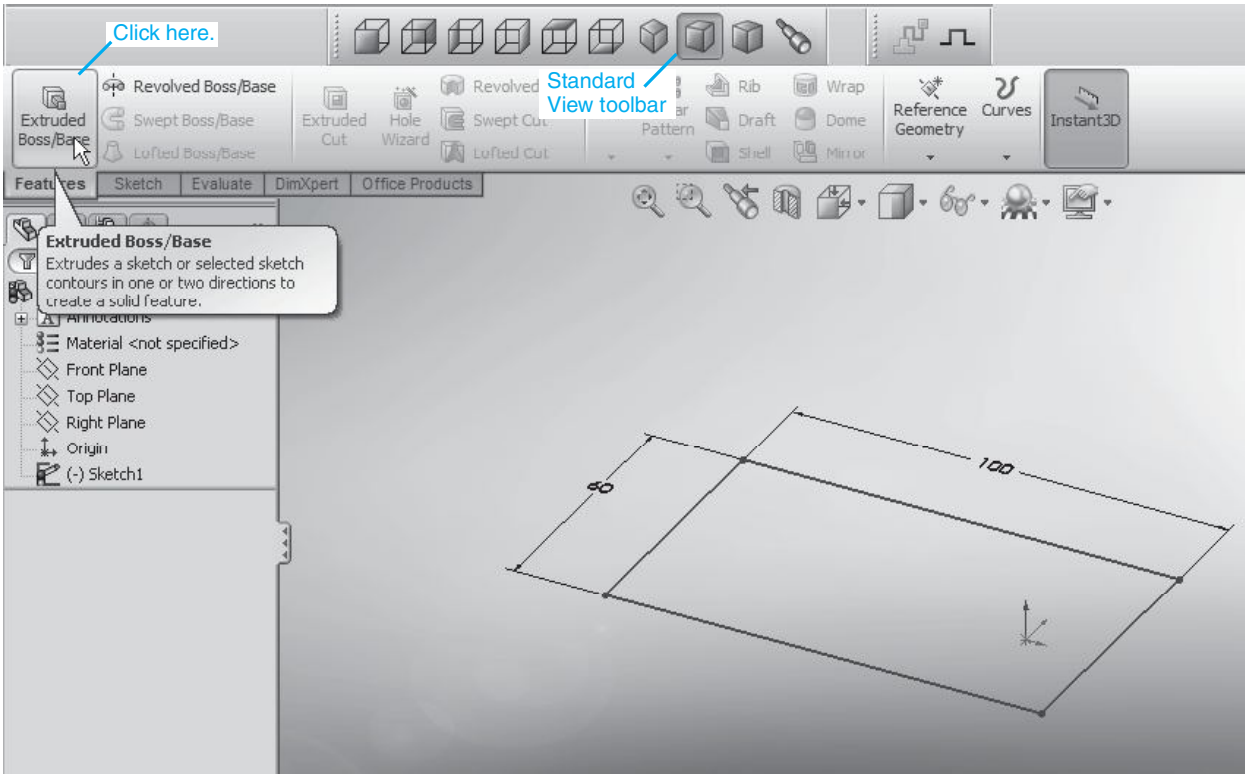


Figure 3-2



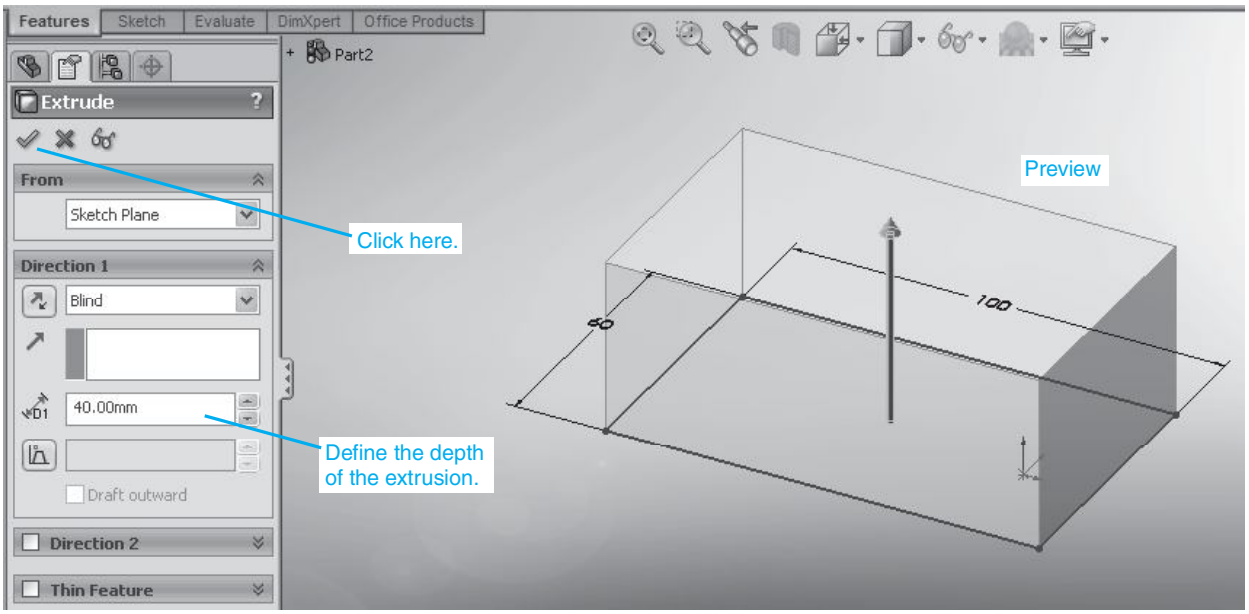


Figure 3-2 ***(continued)***

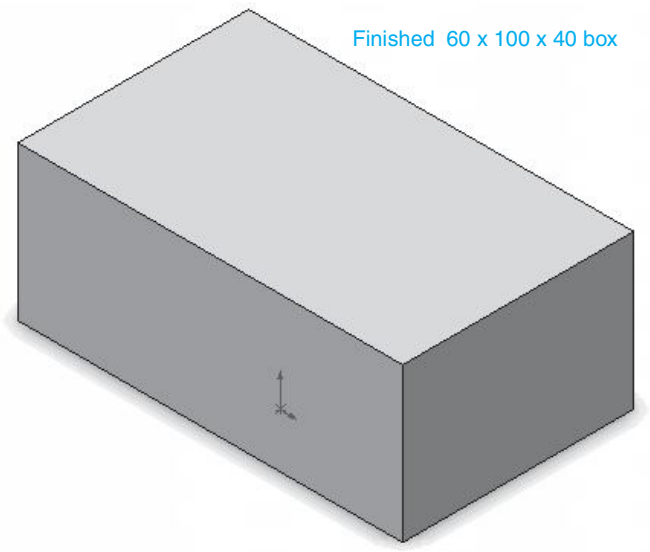


Figure 3-2 ***(continued)***

TIP

The extrusion depth may be defined by entering a value or by using the arrows at the right of the **Depth** box.

6. Click the OK check mark at the top of the **Extrude Properties Manager.**

The preceding example has perpendicular sides. The **Extrude** tool may also be used to create tapered sides. See Figure 3-3 and Figure 3-4. Tapered sides are called ***draft sides.***

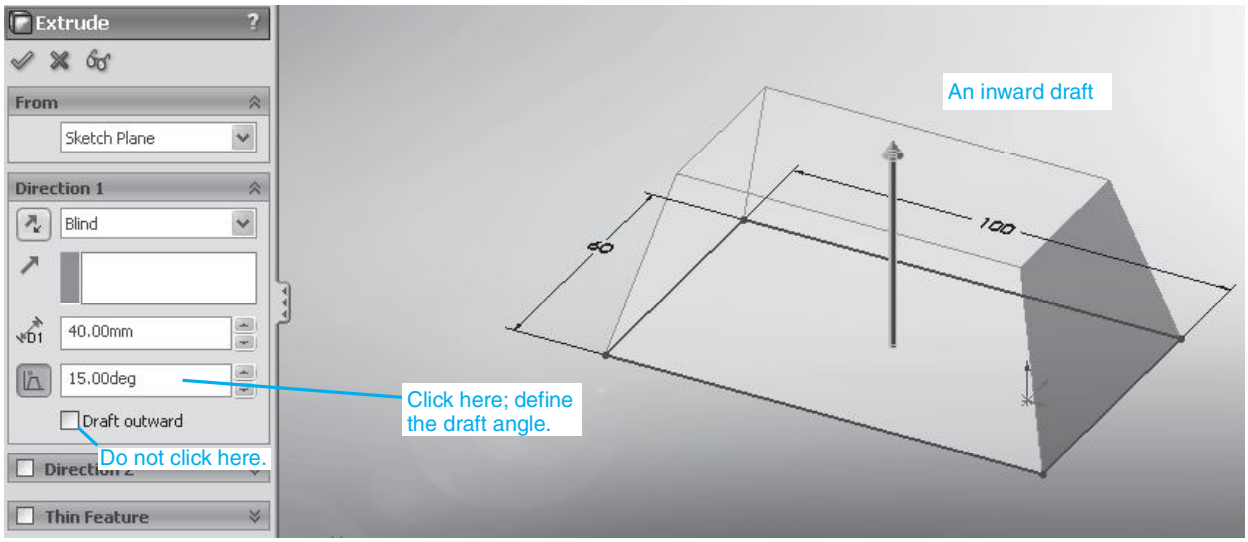


Figure 3-3

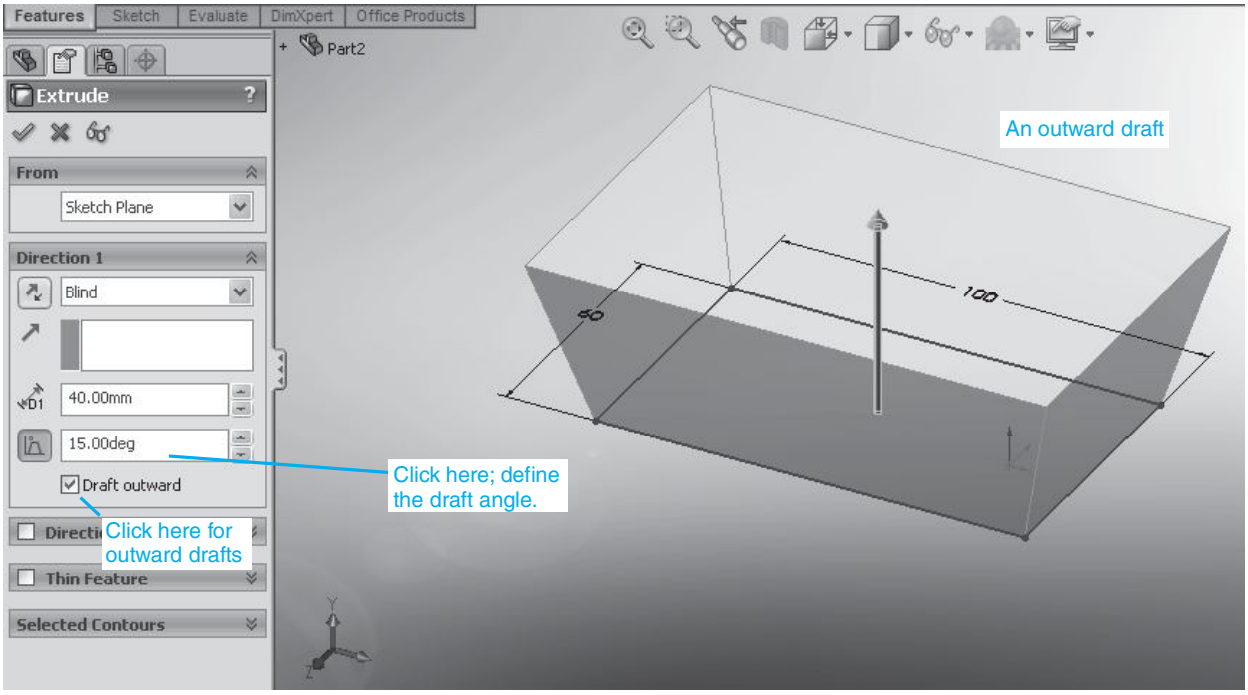


Figure 3-4

To Create Inward Draft Sides

1. Draw a **60 × 100** rectangle, click the **Features** tool, and click the **Extruded Boss/Base** tool.

   The **Extrude Properties Manager** will appear.

2. Click the **Draft On/Off** box.
3. Enter the draft angle value.

   In this example a **15°** value was entered. See Figure 3–3.

4. Click the OK check mark at the top of the **Extrude Properties Manager** to complete the object.

   The draft shown in Figure 3-3 is an inward draft.

To Create an Outward Draft

1. Repeat the same procedure, but this time check the **Draft outward** box.

   See Figure 3-4.

3-3 SAMPLE PROBLEM SP3-1

This section shows how to draw a solid 3D model of an L-bracket using the **Extrude** tool.

1. Draw a **60 × 100** rectangle and extrude it to a depth of **20mm.**

   See Figure 3-5.

2. Locate the cursor on the top surface of the box and right-click the mouse. Select the **Sketch** tool.

   The 2D sketch tools will return to the top of the screen.

3. Use the **Rectangle** tool to draw a rectangle on the top surface of the box. Use the upper left corner of the box as one corner of the rectangle.

Note:
The corner points and edge lines will change colors when they are activated.

4. Select the **Smart Dimension** tool and size the rectangle to **20 × 100.**
5. Click the **Features** tool, then select the **Extruded Boss/Base** tool.
6. Select the **20 × 100** rectangle to extrude to a depth of **40.00mm.**
7. Click the OK check mark in the **Extrude Properties Manager.**

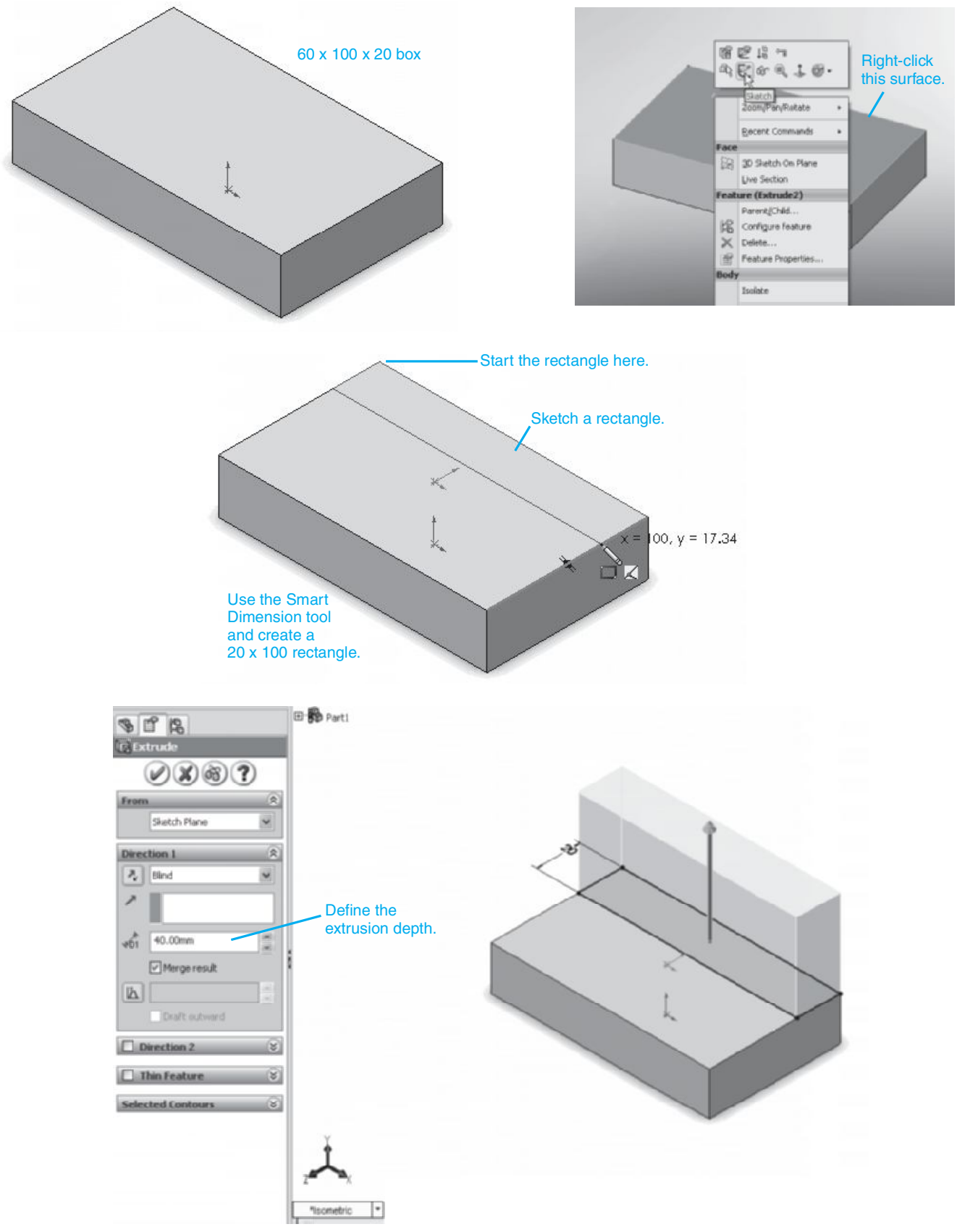
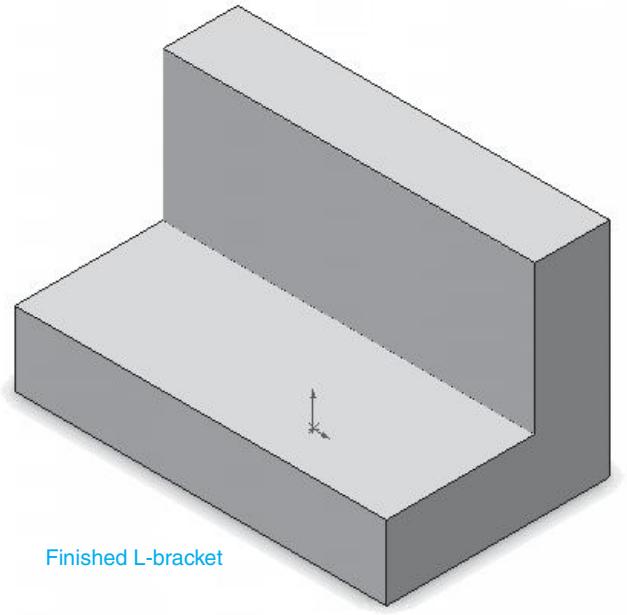

60 x 100 x 20 box
Right-click this surface.
Sketch
Zoom/Pan/Rotate
Recent Commands
Face
3D Sketch On Plane
Live Section
Feature (Extrude2)
Parent/Child...
Configure feature
Delete...
Feature Properties...
Body
Isolate
Start the rectangle here.
Sketch a rectangle.
x = 100, y = 17.34
Use the Smart Dimension tool and create a 20 x 100 rectangle.
Part1
Extrude
From
Sketch Plane
Direction 1
Blind
40.00mm
Merge result
Draft outward
Direction 2
Thin Feature
Selected Contours
Define the extrusion depth.
*Isometric

Figure 3-5



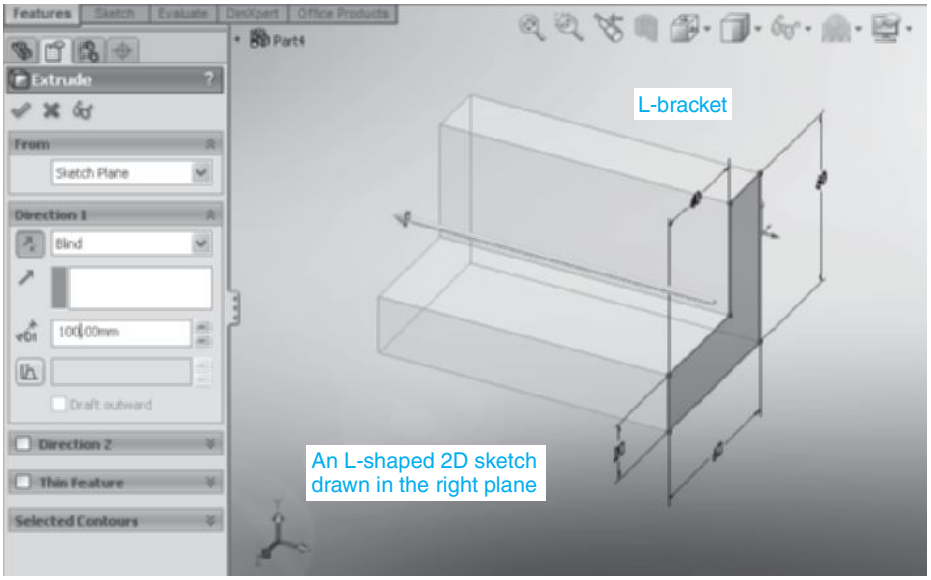


Figure 3-5 ***(continued)***

3-4 EXTRUDED CUT

This section will add a cutout to the L-bracket using the **Extruded Cut** tool. See Figure 3-6.

1. Locate the cursor on the lower front horizontal surface and right-click the mouse.
2. Click the **Sketch** option.
3. Use the **Rectangle** and **Smart Dimension** tools to draw a rectangle as shown.
4. Click the **Features** tool, then click the **Extruded Cut** tool.
5. Click the OK check mark in the **Cut-Extrude Properties Manager.**

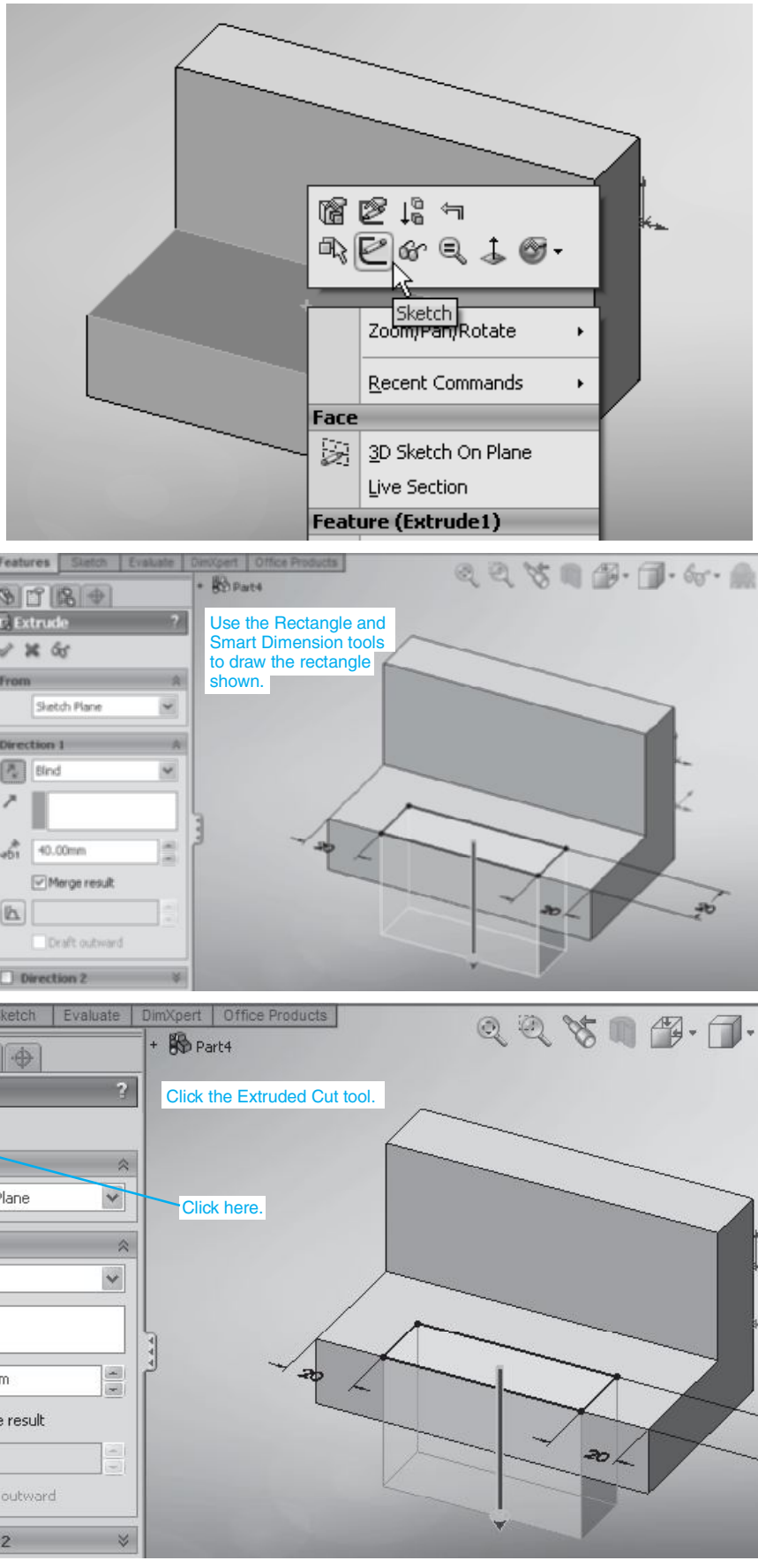

Sketch
Zoom/Pan/Rotate
Recent Commands
Face
3D Sketch On Plane
Live Section
Feature (Extrude1)
Features
Sketch
Evaluate
DimXpert
Office Products
Part4
Extrude
From
Sketch Plane
Direction 1
Blind
40.00mm
Merge result
Draft outward
Direction 2
Use the Rectangle and Smart Dimension tools to draw the rectangle shown.
Click the Extruded Cut tool.
Click here.

Figure 3-6

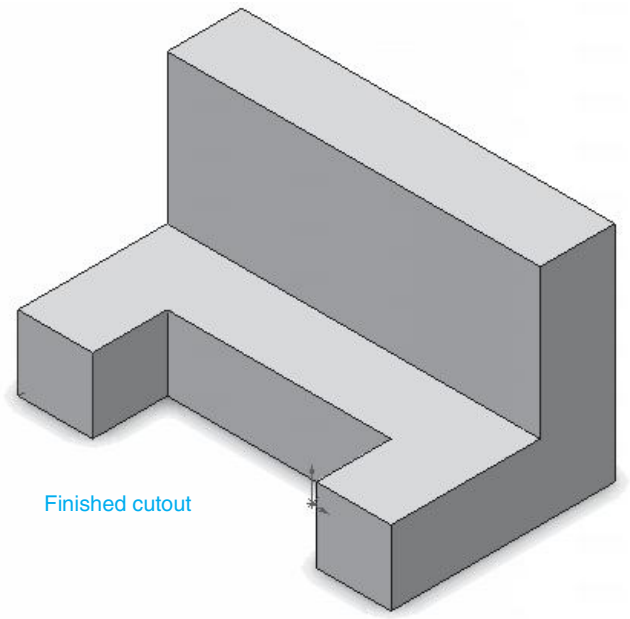


Figure 3-6 ***(continued)***

3-5 HOLE WIZARD

This section will add a hole to the L-bracket using the **Hole Wizard.** See Figure 3-7.

1. Click the **Features** tool, then click the **Hole Wizard** tool.
2. Click the front face as shown.
3. Select the type of hole, for example, clear, counterbore, or threaded.

In this example a clear hole was selected. SolidWorks calls this option **Hole.**

4. Select the **Standard** units.

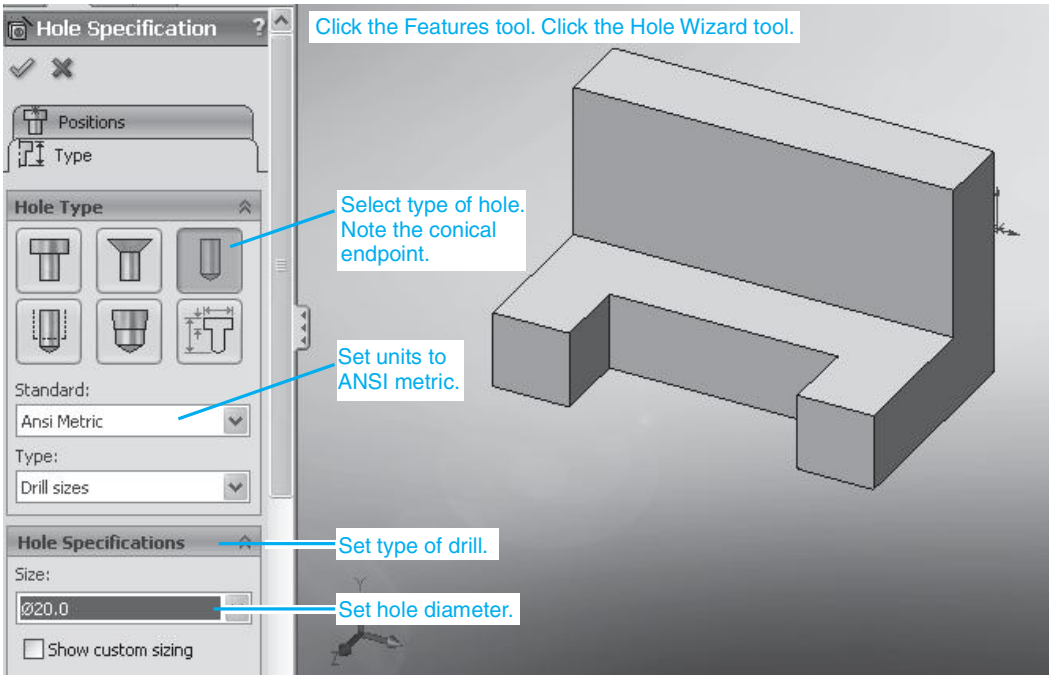
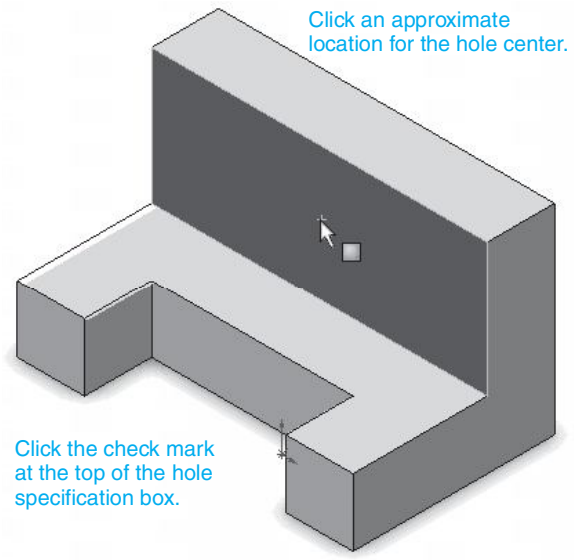
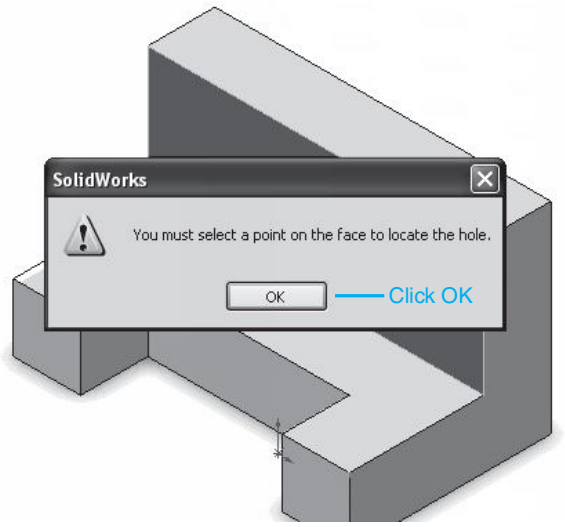
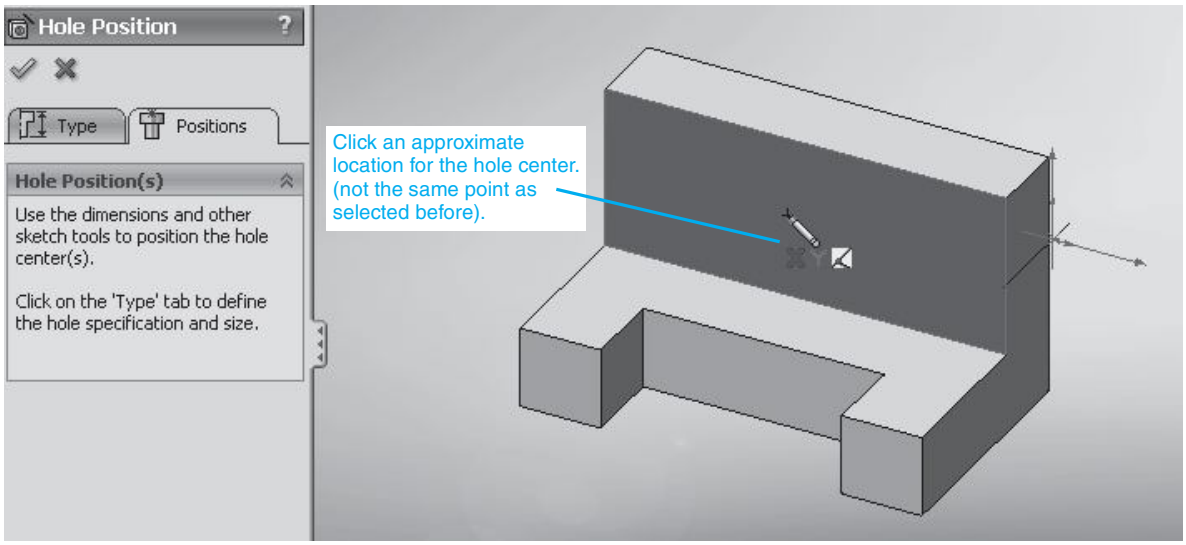
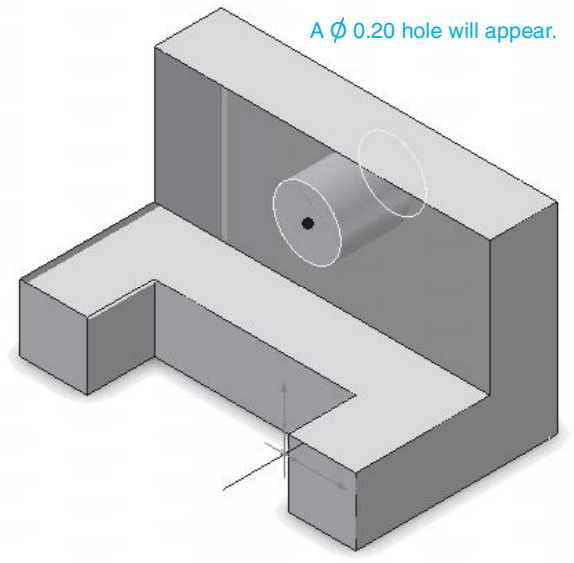


Figure 3-7









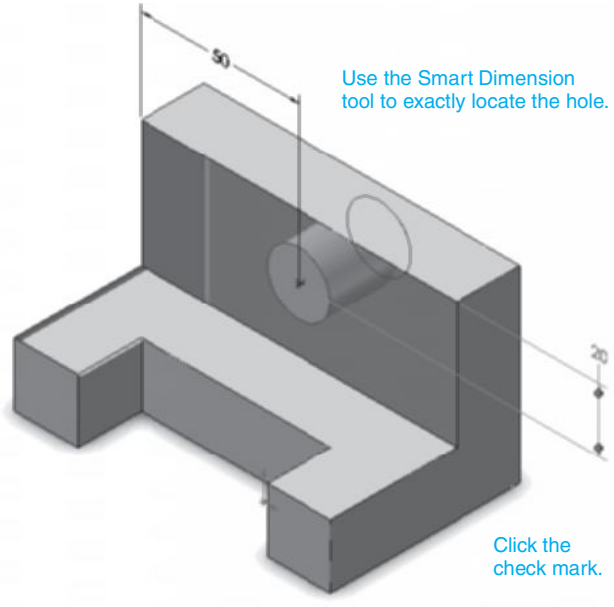


Figure 3-7 ***(continued)***

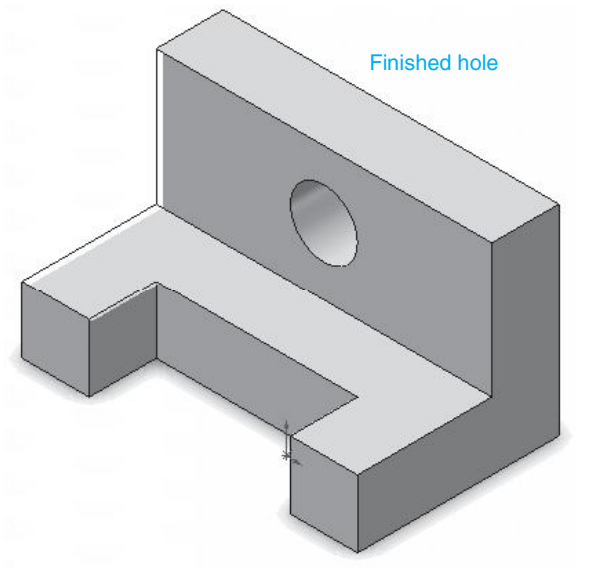


Figure 3-7 *(continued)*

In this example **Ansi Metric** was selected. ANSI is the American National Standards Institute, which publishes standards that will be covered in detail in the chapter on orthographic views and dimensions and tolerances.

5. Define the diameter of the hole.

   In this example a diameter of **20.0mm** was selected.

6. Click an approximate center point location for the hole.
7. Click the OK check mark.

   A dialog box will appear.

8. Click **OK** in the dialog box.
9. Again click an approximate location for the hole's center point.

TIP

Do not click the same point as was defined in step 6.

A preview of the hole will appear.

10. Use the **Smart Dimension** tool to locate the hole's center point.
11. Click the OK check mark.

    The hole will be added to the L-bracket.

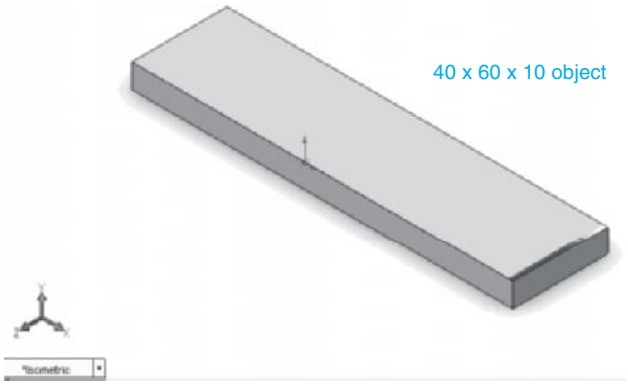
12. Save the L-bracket, as it will be used in later sections.

The hole created in Figure 3-7 is a ***through hole,*** that is, it goes completely through the object. Holes that do not go completely though are called ***blind holes.*** Note that the **Hole Specification Properties Manager** shows a conical point at the bottom of the hole. Holes created using an extruded cut circle will not have this conical endpoint. Blind holes created using a drill should include the conical point. For this reason, blind holes should, with a few exceptions, be created using the **Hole Wizard** tool.

3-6 A SECOND METHOD OF CREATING A HOLE

Holes may also be created using the **Circle** tool and then applying the **Extruded Cut** tool. Figure 3-8 shows a 40 × 160 × 10 object. Add four **Ø20** holes.

1. Draw a **40 × 160 × 10** object.
2. Right-click the mouse and select the **Sketch** tool. Use the **Circle** tool and draw a circle. Use the **Smart Dimension** tool to size and locate the circle.
3. Add three more **Ø20** circles.



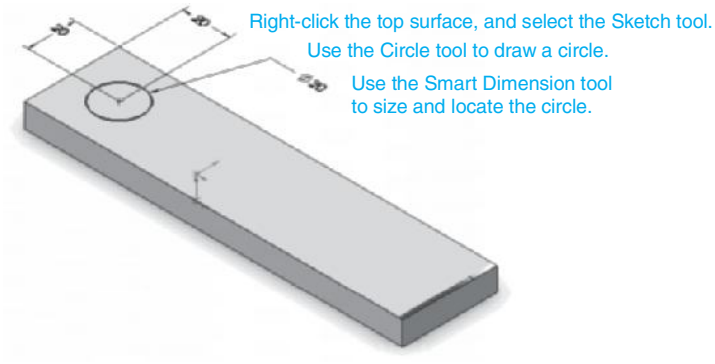


Figure 3-8

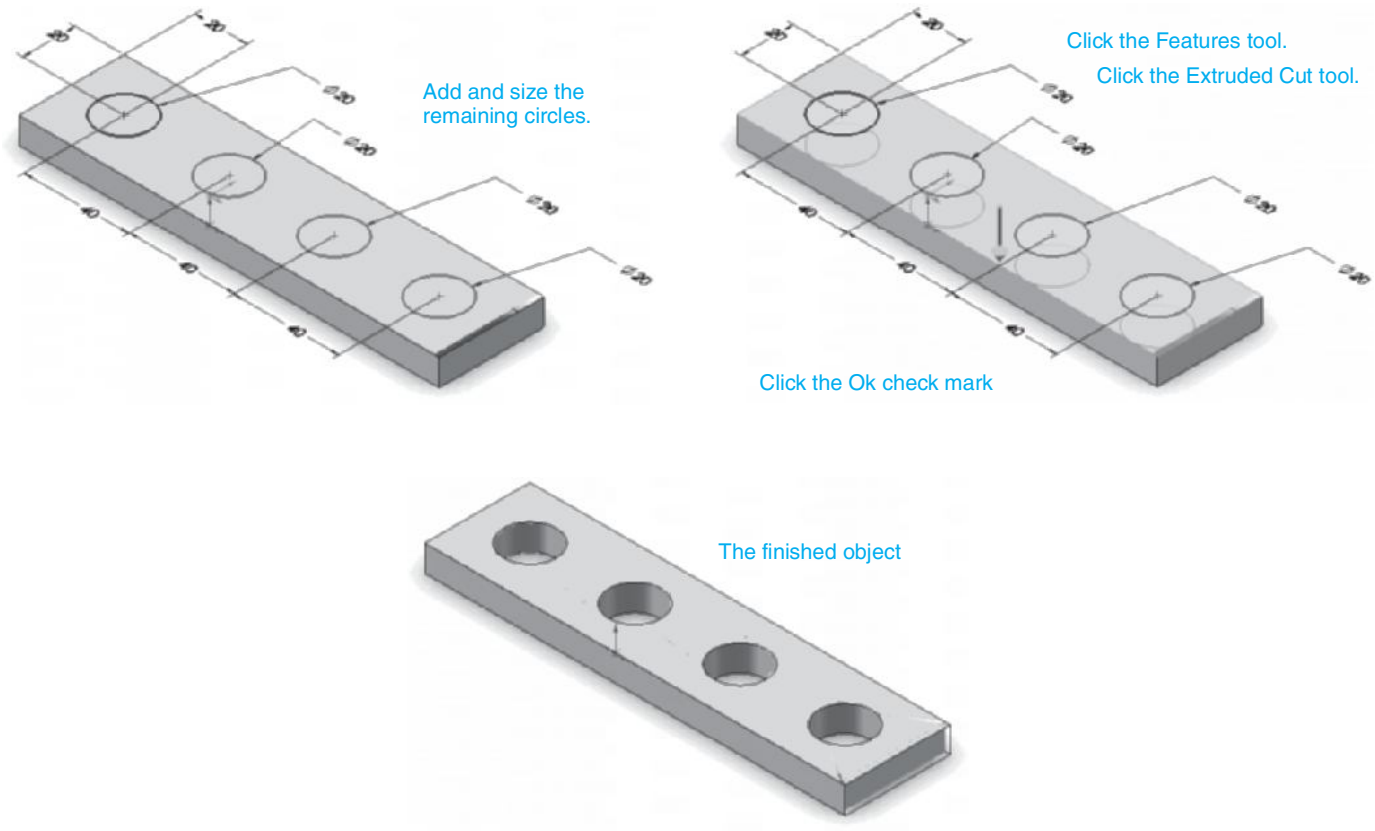


Figure 3-8 ***(continued)***

4. Click the **Features** tool, then click the **Extruded Cut** tool.

   A preview of the hole will appear.

5. Set the cut depth for **10** or greater.
6. Click the OK check mark.

TIP

The same four-hole pattern can be created using the **Linear Sketch Pattern** tool.

3-7 FILLET

A ***fillet*** is a rounded corner. Specifically, convex corners are called ***rounds,*** and concave corners are called *fillets,* but in general, all rounded corners are called *fillets*.

SolidWorks can draw four types of fillets: constant radius, variable radius, face fillets, and full round fillets. Figure 3-9 shows the L-bracket used to demonstrate the previous **Features** tools. It will be used in this section to demonstrate fillet tools.

1. Open the L-bracket drawing.

   If the L-bracket was not saved after the previous section, use the dimensions and procedures specified to re-create the object.

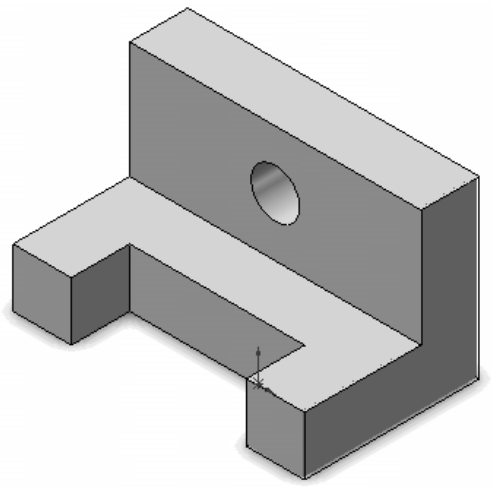


Figure 3-9

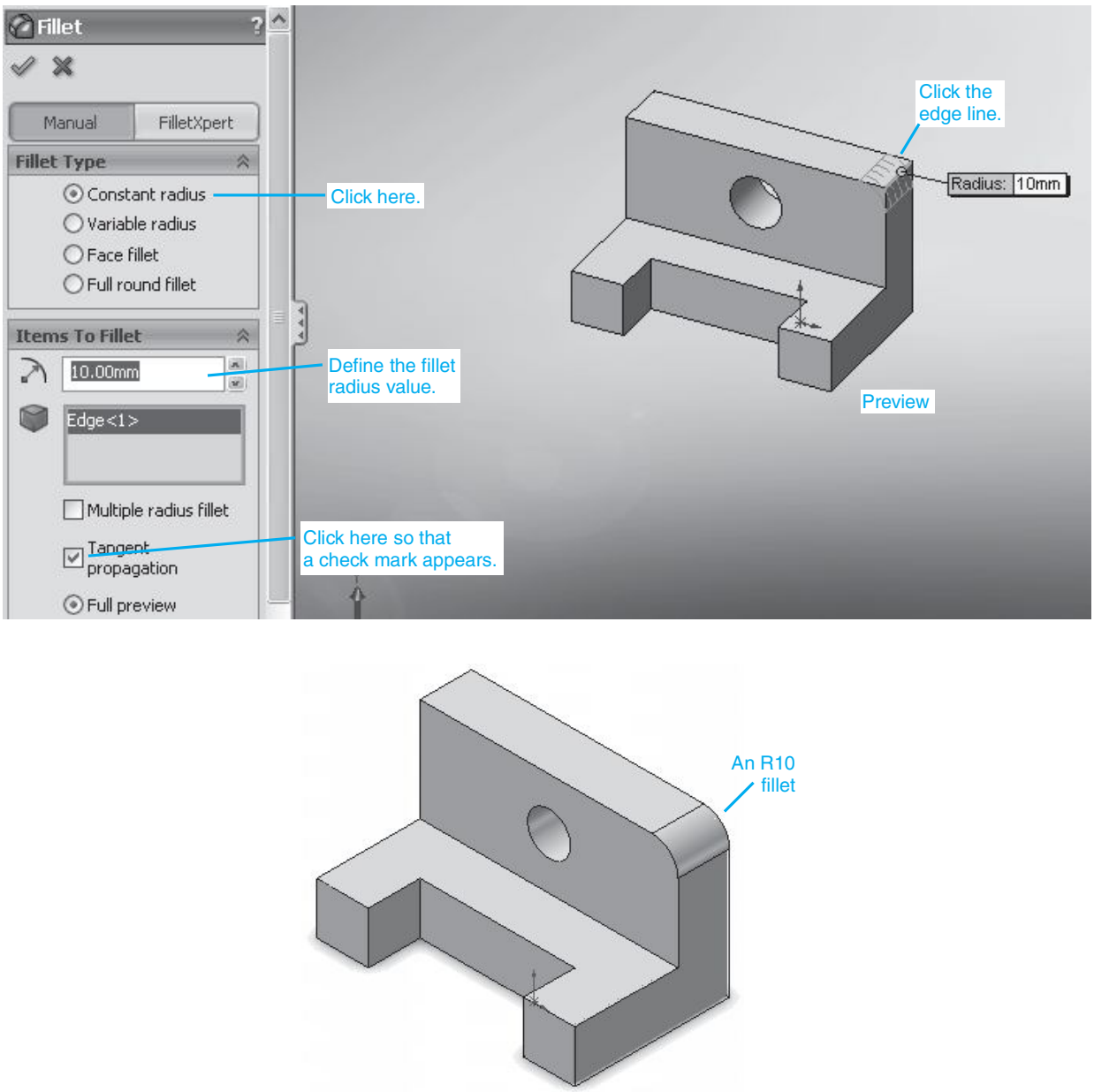


Figure 3-9 ***(continued)***

2. Click the **Fillet** tool.
3. Select **Constant radius** in the **Fillet Type** box and define the fillet's radius as **10.00mm.** Click the **Full preview** button.
4. Click the upper right edge line of the object,

   A preview of the fillet will appear.

5. Click the OK check mark.

To Create a Fillet with a Variable Radius

See Figure 3-10.

1. Click the **Fillet** tool.
2. Click the **Variable radius** button.
3. Click the edge line shown in Figure 3-10.

   Two boxes will appear on the screen, one at each end of the edge line.

4. Click the word **Unassigned** in the left box and enter a value of **15.**
5. Click the word **Unassigned** in the right box and enter a value of **5.**
6. Click the OK check mark.

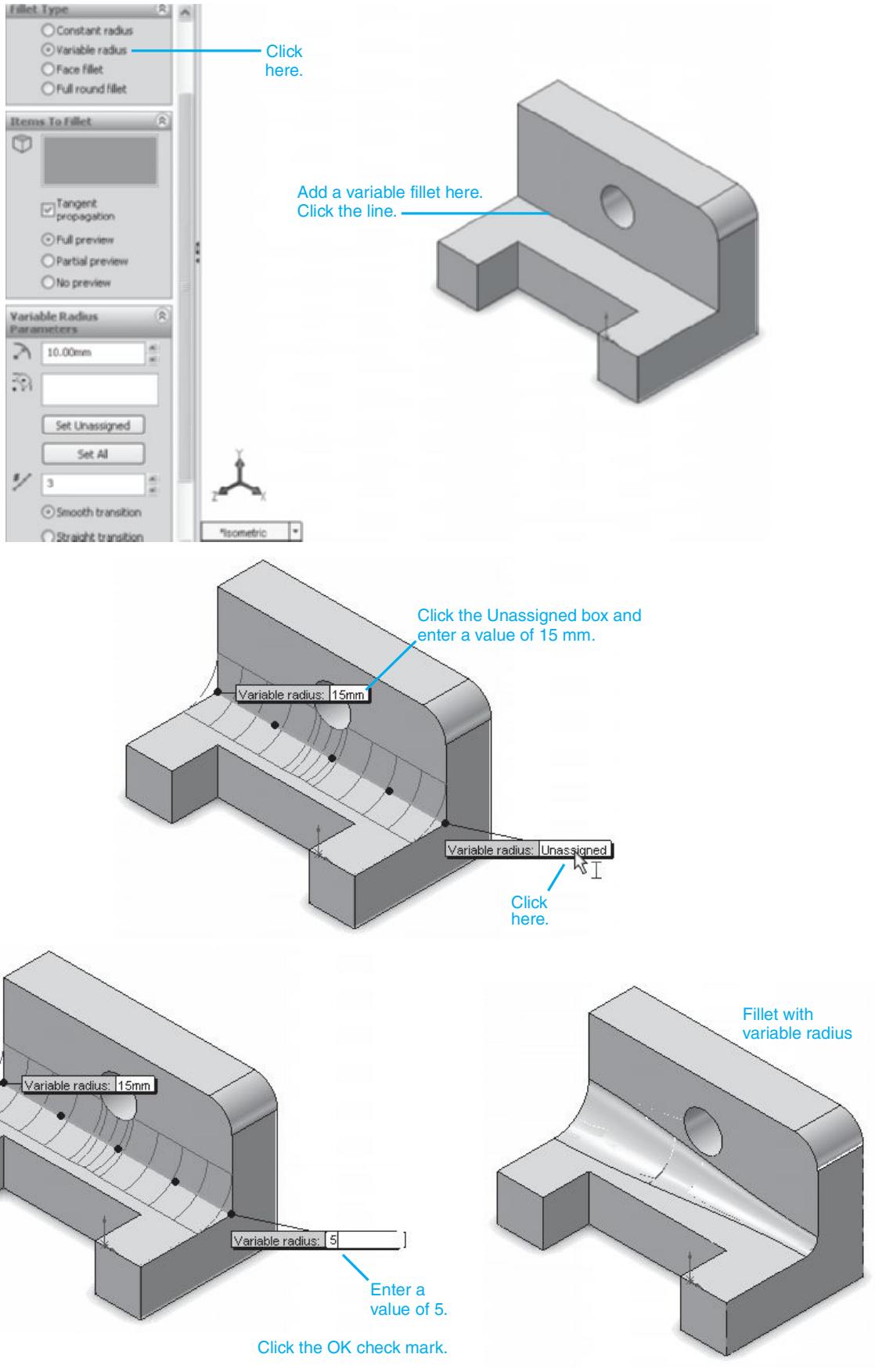

Fillet Type
Constant radius
Variable radius
Face fillet
Full round fillet
Items To Fillet
Tangent propagation
Full preview
Partial preview
No preview
Variable Radius Parameters
10.00mm
Set Unassigned
Set All
3
Smooth transition
Straight transition
*Isometric
Click here.
Add a variable fillet here.
Click the line.
Click the Unassigned box and enter a value of 15 mm.
Variable radius: 15mm
Variable radius: Unassigned
Click here.
Variable radius: 15mm
Variable radius: 5
Enter a value of 5.
Click the OK check mark.
Fillet with variable radius

Figure 3-10

To Create a Fillet Using the Face Fillet Option

The **Face fillet** option draws a fillet between two faces (surfaces), whereas **Fillet** uses an edge between two surfaces to draw a fillet.

See Figure 3-11.

1. Click the **Fillet** tool.
2. Click the **Face fillet** option.

Two boxes will appear in **Items to Fillet** box. They will be used to define the two faces of the fillet.

3. Define the fillet radius as **10.00mm.**
4. Define **Face 1** as shown.

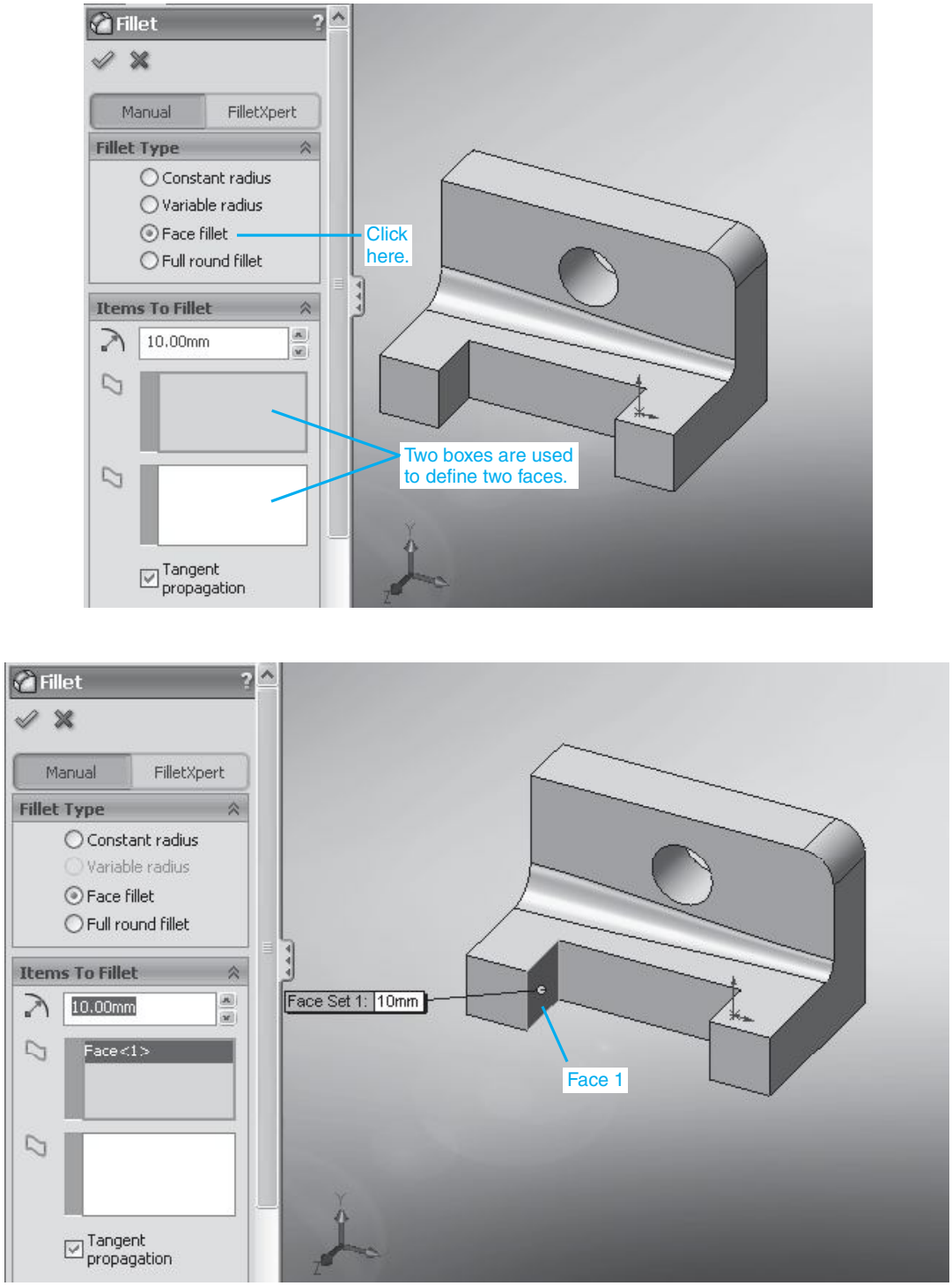


Figure 3-11

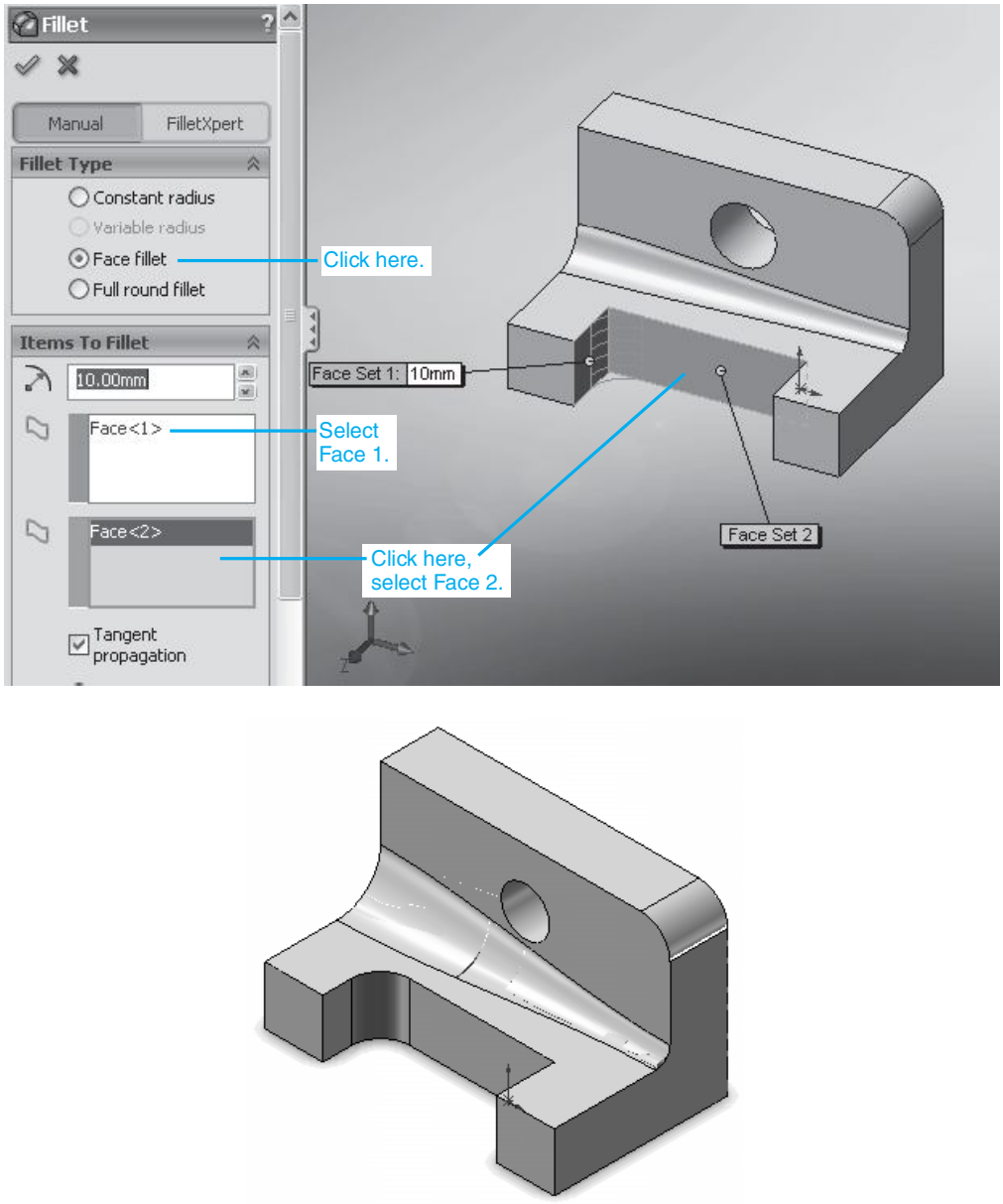


Figure 3-11 ***(continued)***

> ***Note:***
> The top box in the **Items to Fillet** box will be shaded, indicating that it is ready for an input.

5. Click the lower box in the **Items to Fillet** box (it will change colors) and define **Face 2** by clicking the surface as shown.
6. Click the OK check mark.

To Create a Fillet Using the Full Round Fillet Option

See Figure 3-12.

1. Use the **Undo** tool and remove the fillets created previously.

   This will return the original L-bracket shape.

2. Click the **Fillet** tool and click the **Full round fillet** button.

   Three boxes will appear. These boxes will be used to define Side Face 1, the Center Face, and Side Face 2.

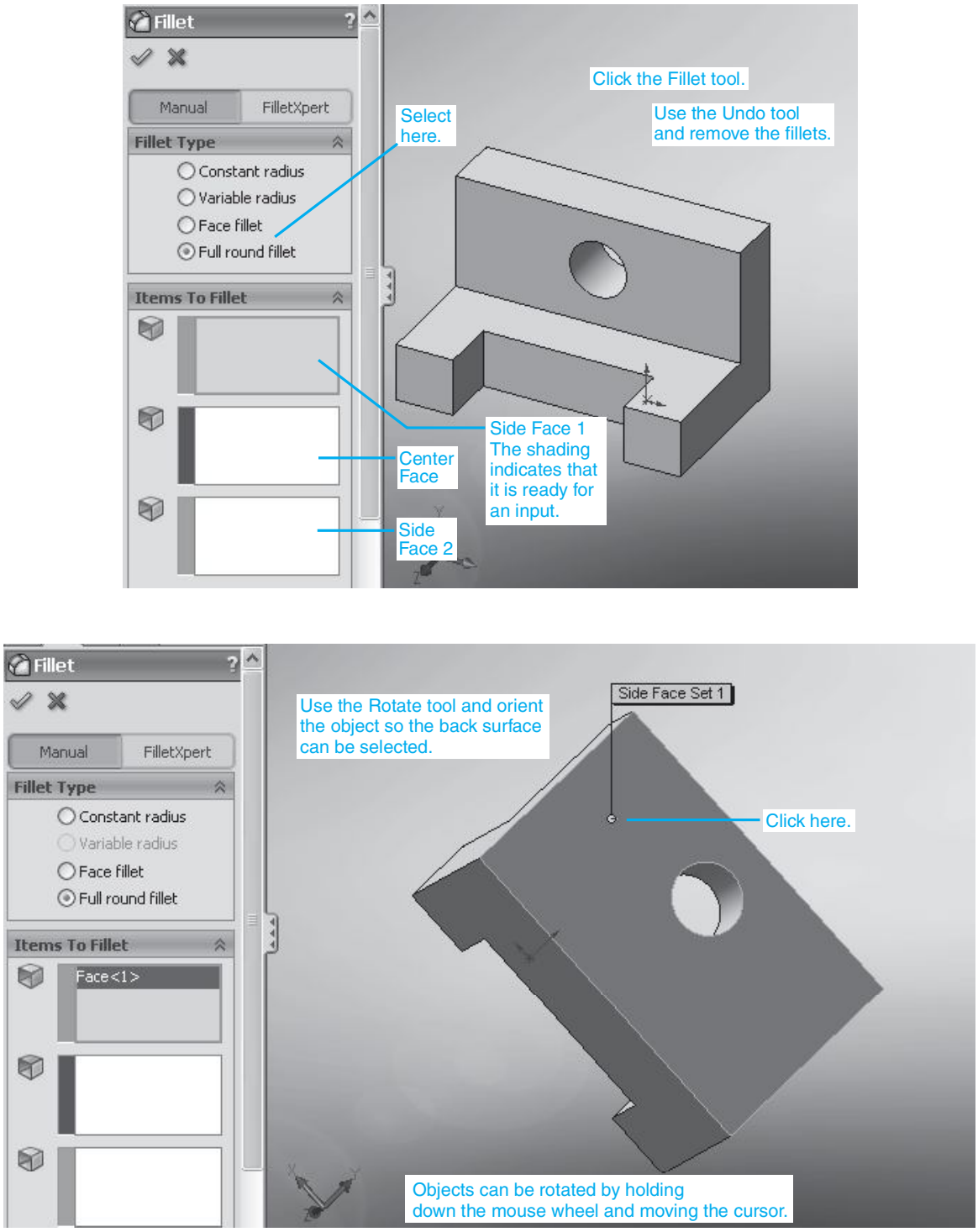


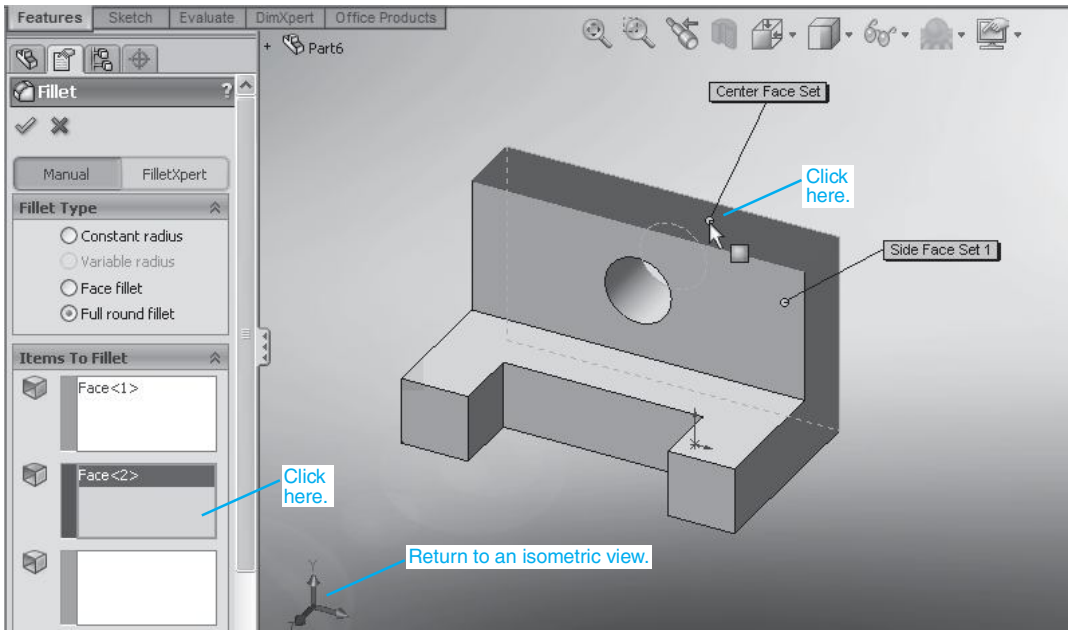
Figure 3-12

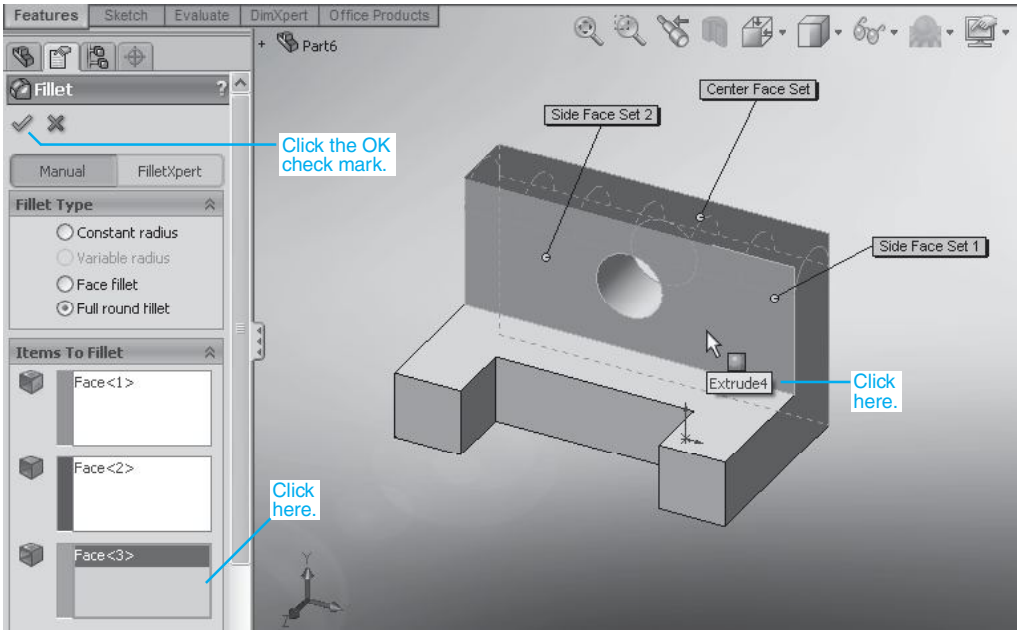
In this example, the default fillet radius value of **10** will be used.

3. Use the **Rotate** tool and orient the object so that the back surface can be selected.

TIP

Objects can be rotated by holding down the mouse wheel and moving the cursor.





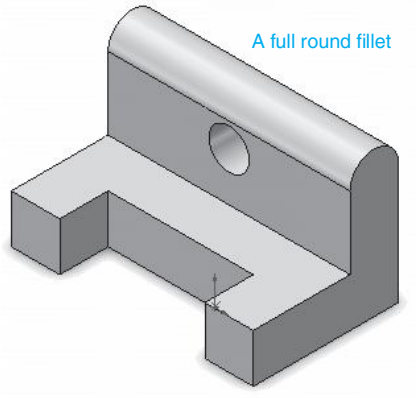


Figure 3-12 ***(continued)***

4. Click the back surface.

   The back surface is defined as **Side Face 1.**

5. Reorient the object to an isometric view.
6. Click the middle box in the **Items to Fillet** box and click the top surface of the object.

   The top surface is defined as the **Center Face.**

7. Click the lower of the three boxes in the **Items to Fillet** box and click the front surface of the object as shown.

   The front surface is defined as **Side Face 2.** A preview of the fillet will appear.

8. Click the OK check mark.

3-8 CHAMFER

A ***chamfer*** is a slanted surface added to a corner of an object. Chamfers are usually manufactured at 45° but may be made at any angle. Chamfers are defined using either an angle and a distance (5 × 45°) or by two distances (5 × 5). A vertex chamfer may also be defined.

To Define a Chamfer Using an Angle and a Distance

See Figure 3-13.

1. Use the **Undo** tool and remove the fillet created in the last section.
2. Click the **Chamfer** tool.

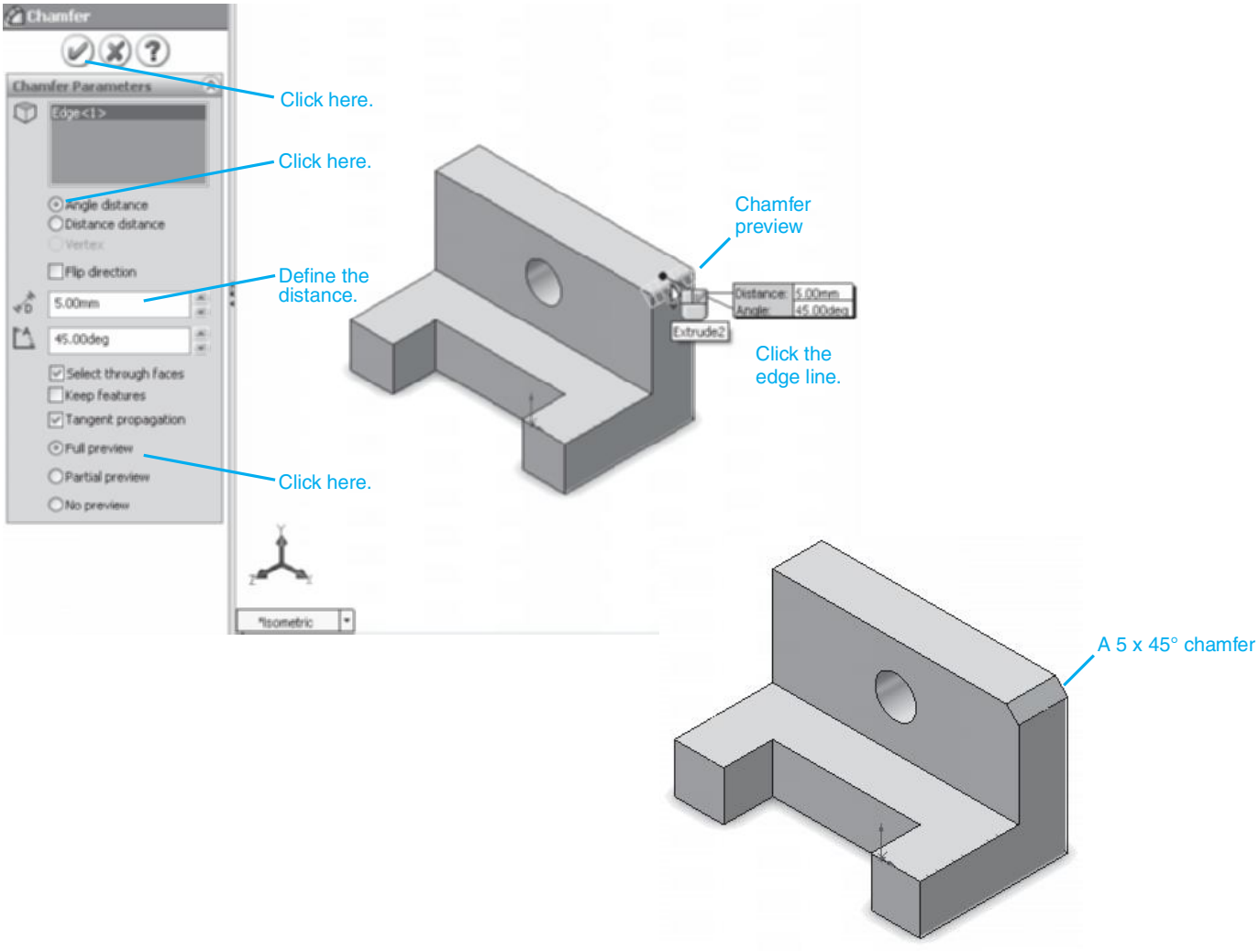


Figure 3-13

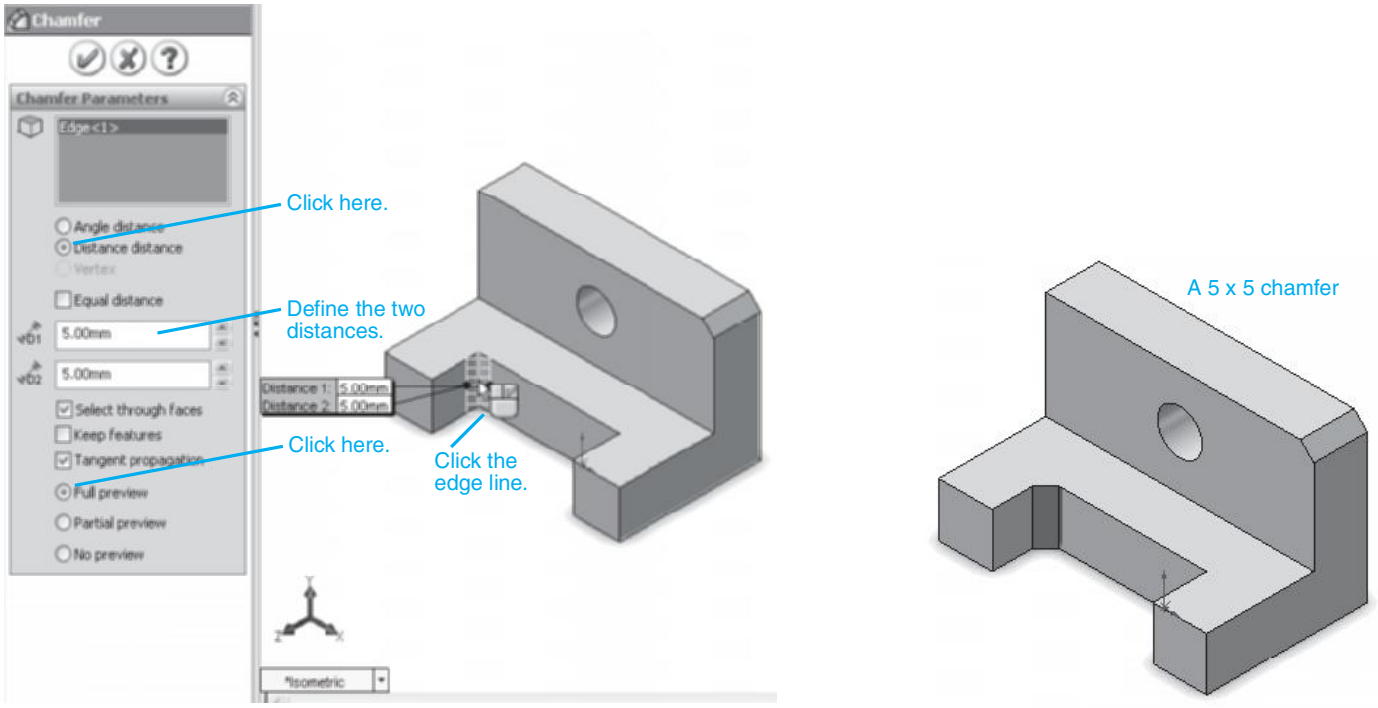


Figure 3-14

3. Click the **Angle distance** button.
4. Define the chamfer distance as **5** and accept the **45°** default value.
5. Click the top side edge line as shown.
6. Click the OK check mark.

To Define a Chamfer Using Two Distances

See Figure 3-14.

1. Click the **Chamfer** tool.
2. Click the **Distance distance** button.
3. Define the two distances as **5.00mm** each.

In this example the two distances are equal. Distances of different lengths may be used.

4. Click the inside vertical line as shown.
5. Click the OK check mark.

To Define a Vertex Chamfer

See Figure 3-15.

1. Click the **Chamfer** tool.
2. Click the **Vertex** button.

Three distance boxes will appear.

3. Define the three distances.

In this example three equal distances of **5.00mm** were used. The three distances need not be equal.

4. Click the lower top corner point as shown.
5. Click the OK check mark.

3-9 REVOLVED BOSS/BASE

The **Revolved Boss/Base** tool rotates a contour about an axis line. See Figure 3-16.

1. Start a new drawing, click the **Sketch** tool, and click the **Top Plane** option.
2. Use the **Sketch** tools to draw a line on the screen and then draw a 2D shape next to the line.
3. Change the drawing's orientation to **Isometric.**

Approximate the shape shown.

4. Click the **Features** tool, then click the **Revolved Boss/Base** tool.
5. Click the **Selected Contours** box, then click the contour on the screen.

> ***Note:***
> If the **Selected Contours** box is already shaded, it means that it has been activated automatically. Click the contour directly.

6. Click the axis box at the top of the **Revolve Parameters Properties Manager.**
7. Click the axis line on the screen

A preview of the revolved object will appear.

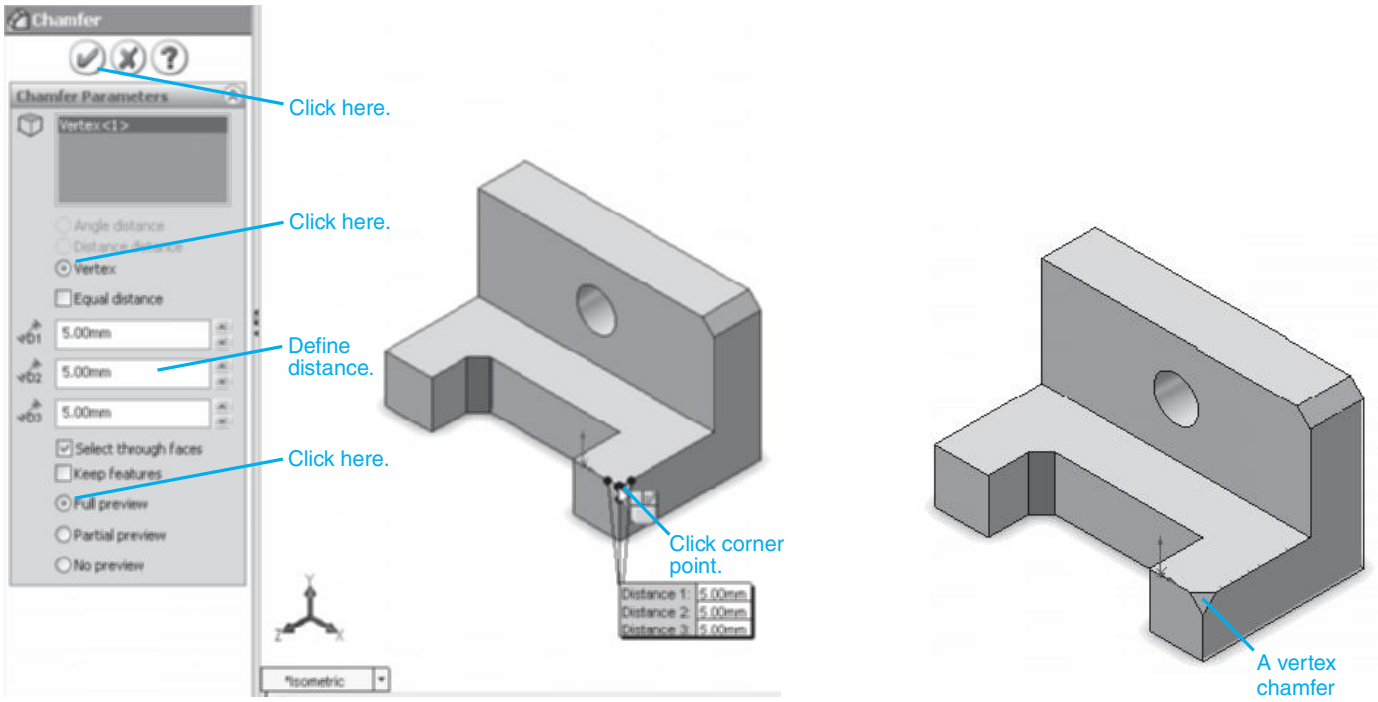


Figure 3-15

8. Click the OK check mark.

Figure 3-17 shows an example of a sphere created using the Revolve tool.

3-10 REVOLVED CUT

The Revolved Cut tool is used to cut revolved sections out of objects. Figure 3-18 shows a 60 × 100 × 40 box. Create a new sketch plane (**Sketch**) on the top surface of the box and use the **Centerpoint Arc** tool to draw an arc of radius **40** centered about the lower corner of the top surface as shown.

1. Click the **Revolved Cut** tool.
2. Click the **Selected Contours** box and select the area within the arc as the contour.
3. Click the **Axis of Revolution** box.
4. Click the top line as shown identifying the axis of revolution.

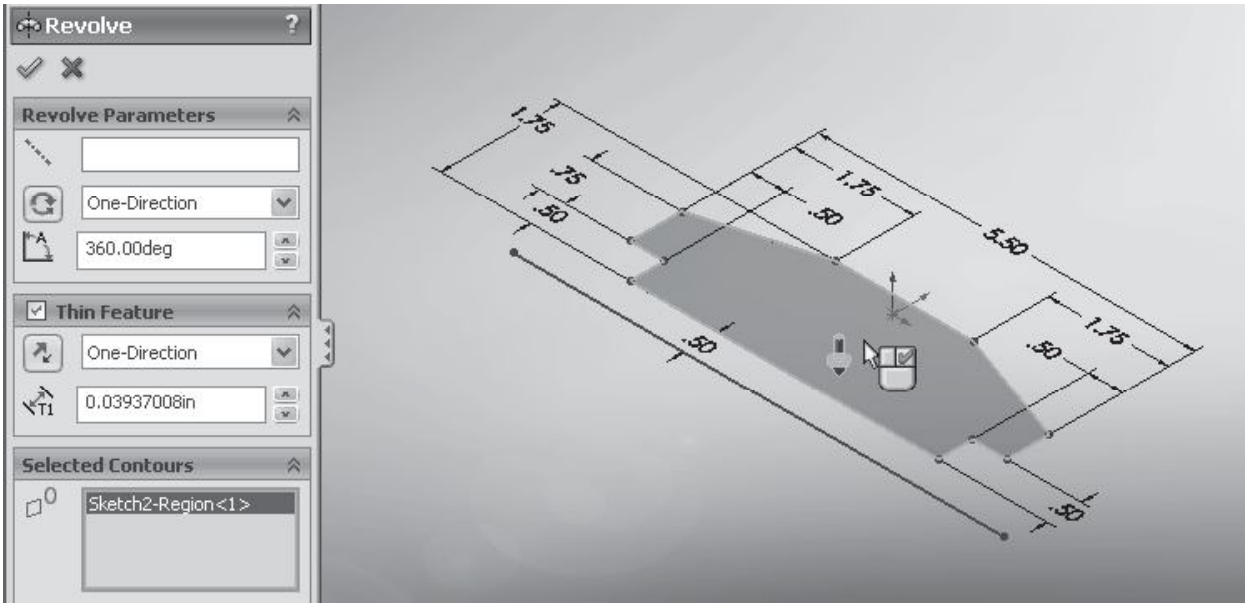
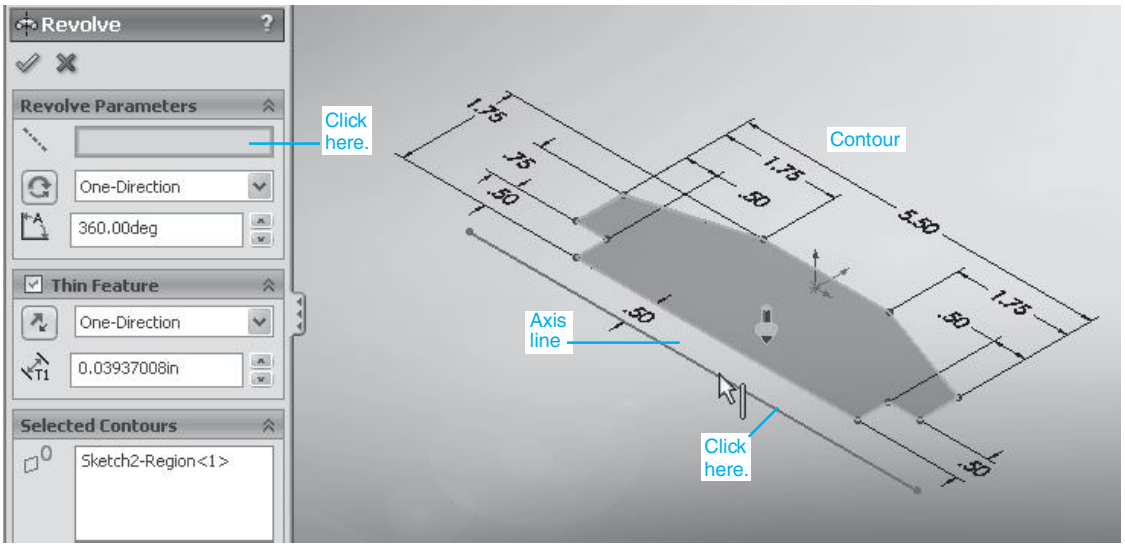
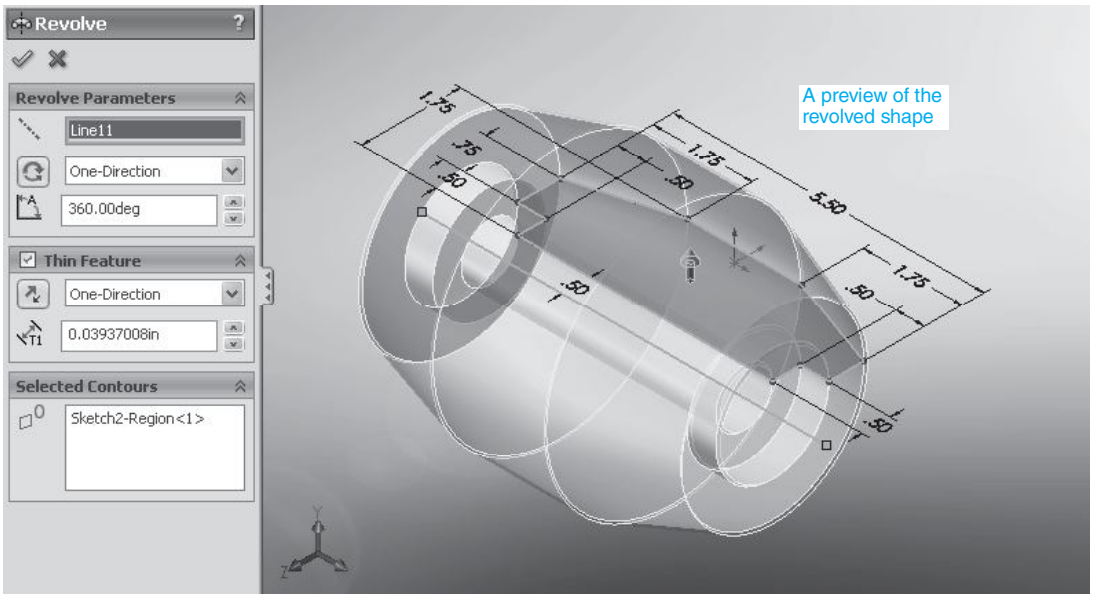


Figure 3-16





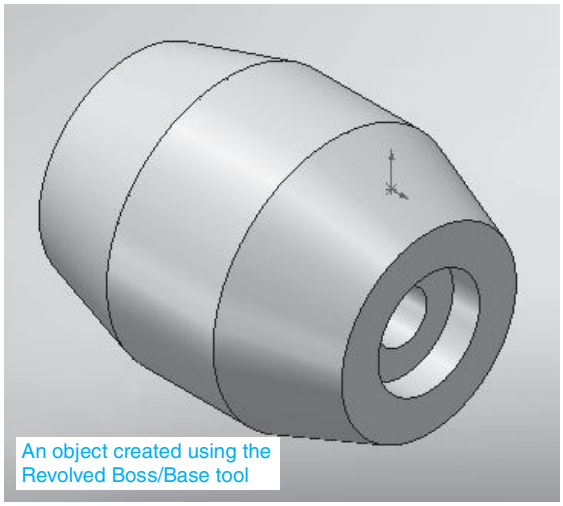
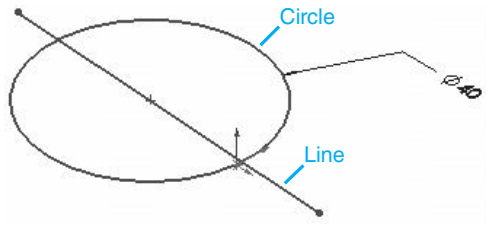
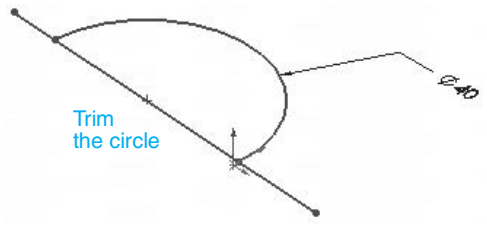
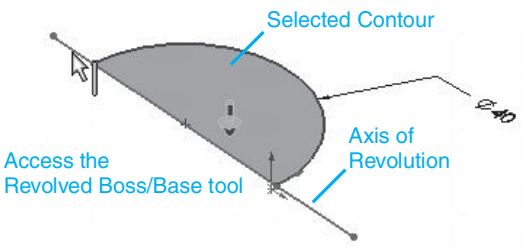
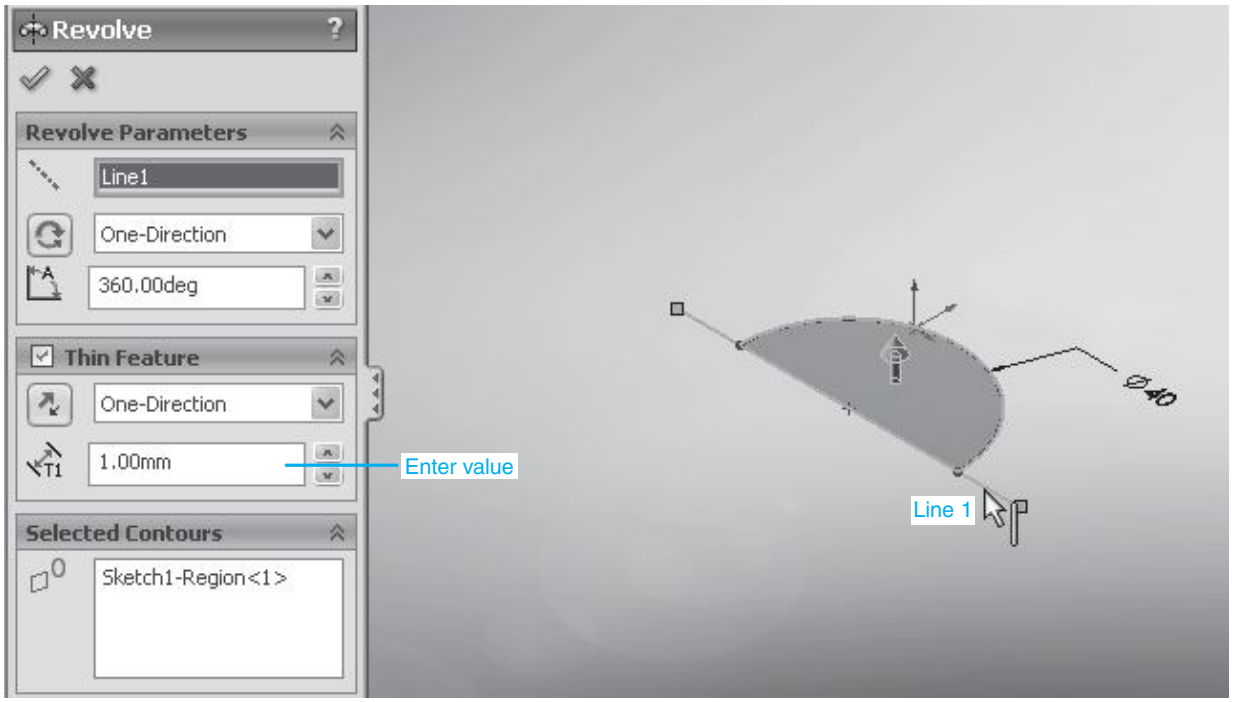


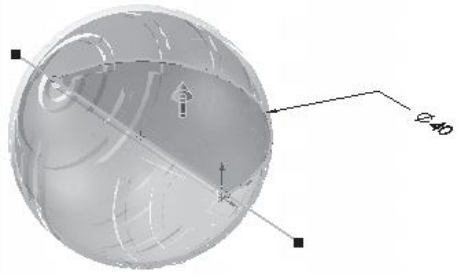
Figure 3-16 ***(continued)***











If necessary, click the Reverse Direction box in the Thin Feature box.

Click the OK check mark.

Figure 3-17

5. Set the **Direction Thickness** for **5.**
6. Click the **Reverse Direction** box.
7. Click the OK check mark.

   The **Bodies to keep** dialog box will appear.

8. Click **Select bodies** in the **Bodies to keep** dialog box.
9. Click **Body 1.**
10. Click **OK.**

3-11 REFERENCE PLANES

Reference planes are planes that are not part of an existing object. Up to now if we needed a new sketch plane, we selected an existing plane on the object. Consider the Ø3.0 × 3.50 cylinder shown in Figure 3-19. The cylinder was drawn with its base on the top plane. How do we create a hole through the rounded sides of the cylinder? If we right-click the rounded surface, no **Sketch** tool will appear.

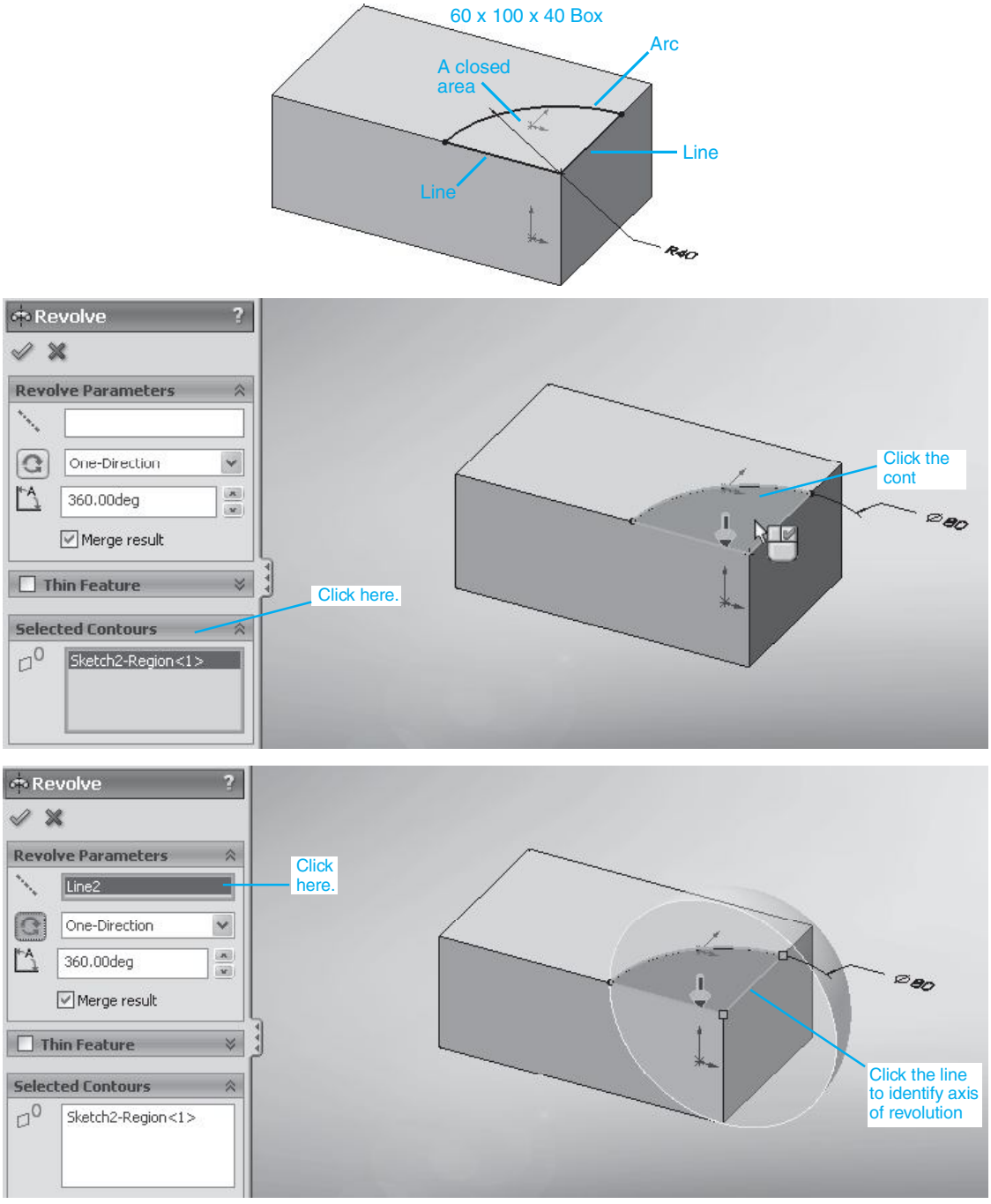


Figure 3-18

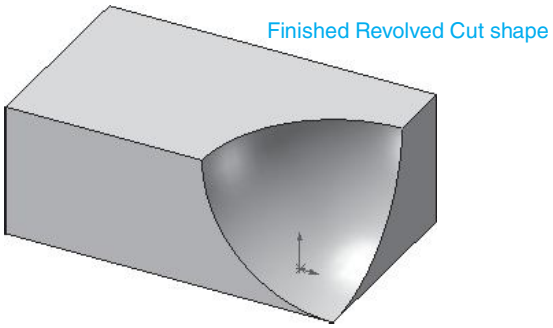
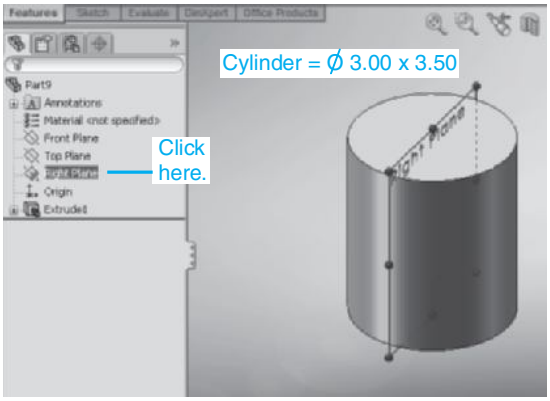


Figure 3-18 ***(continued)***



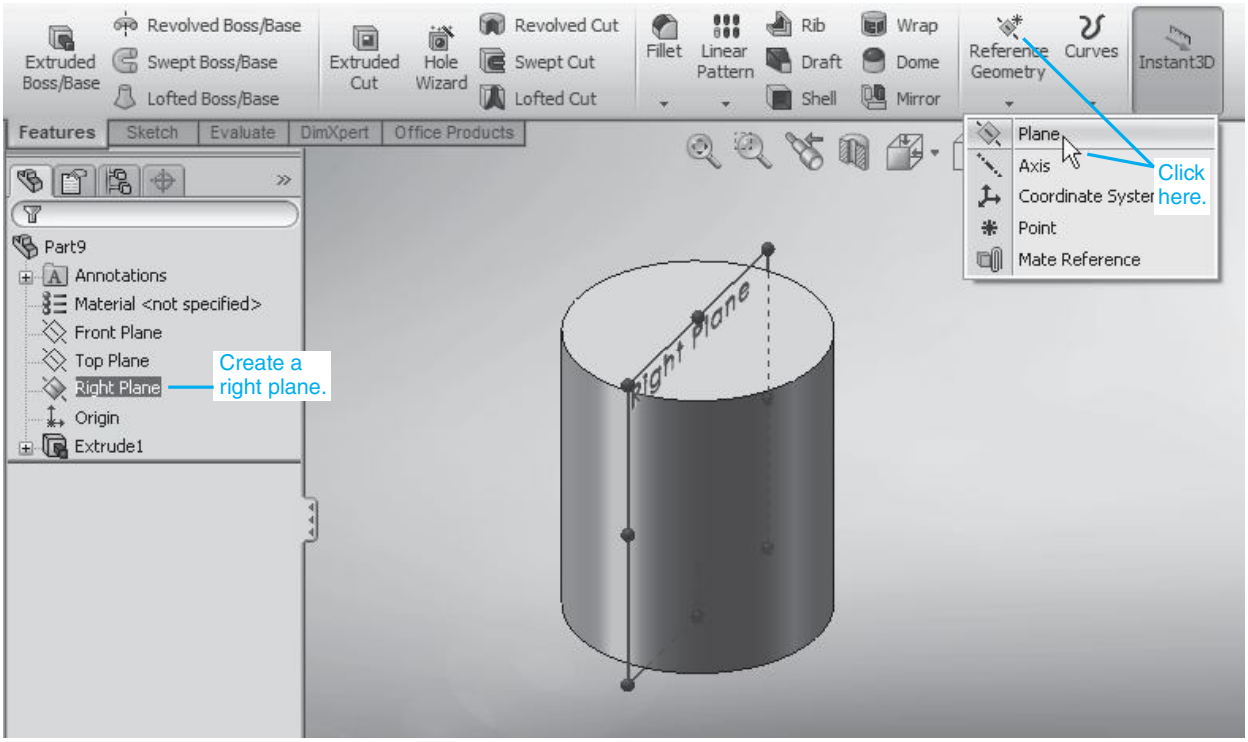
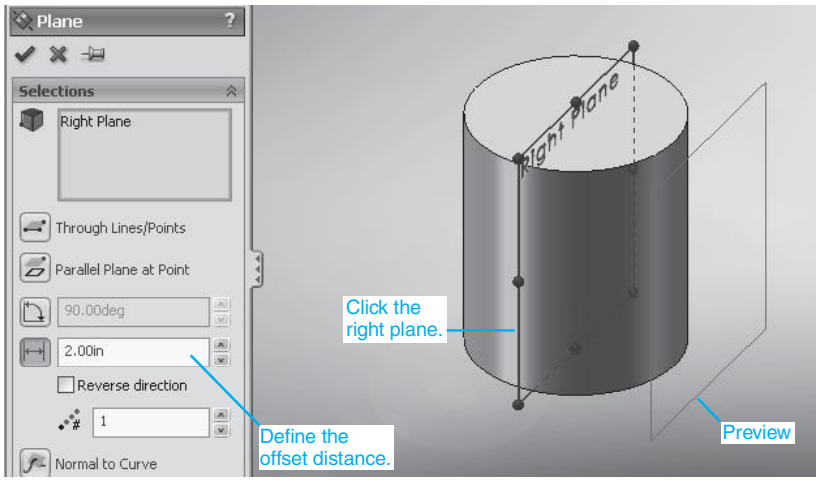
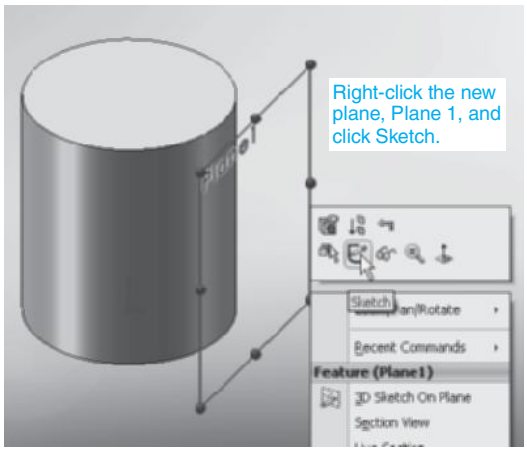
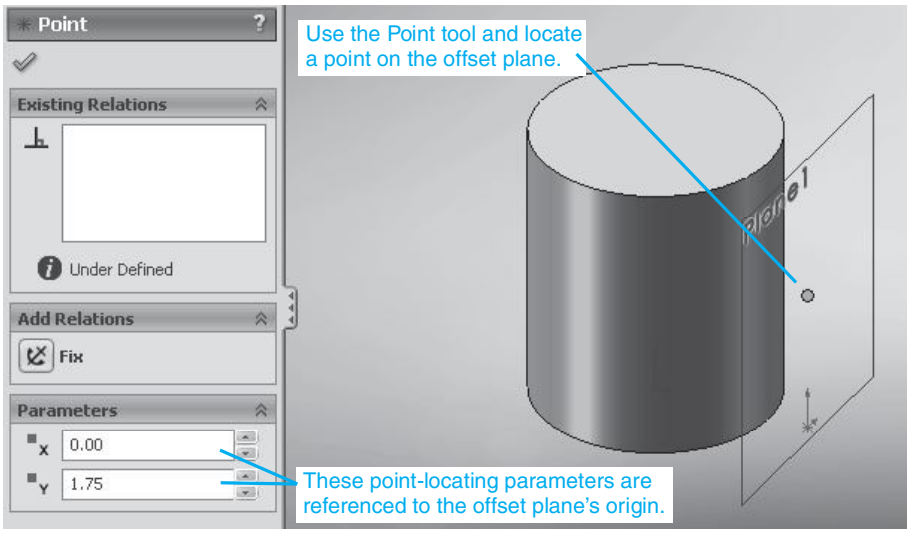


Figure 3-19







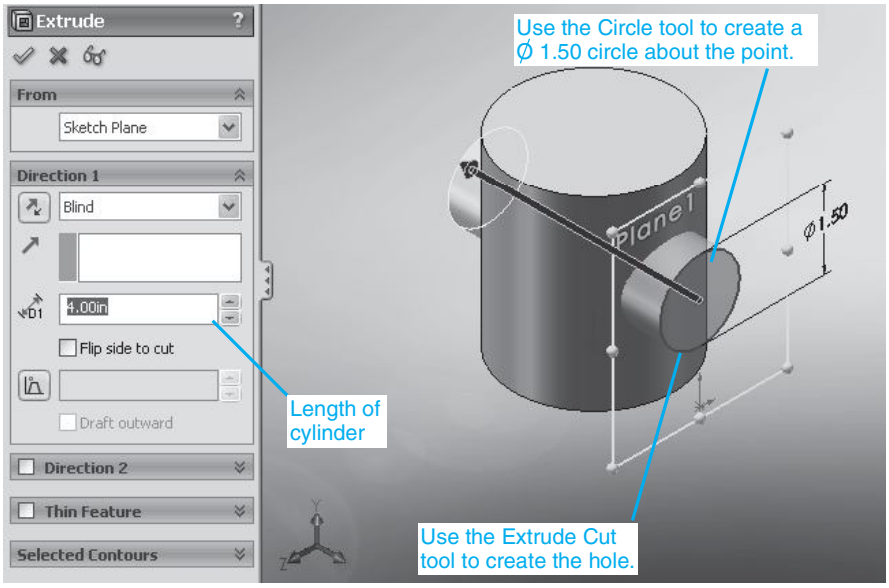


Figure 3-19 ***(continued)***

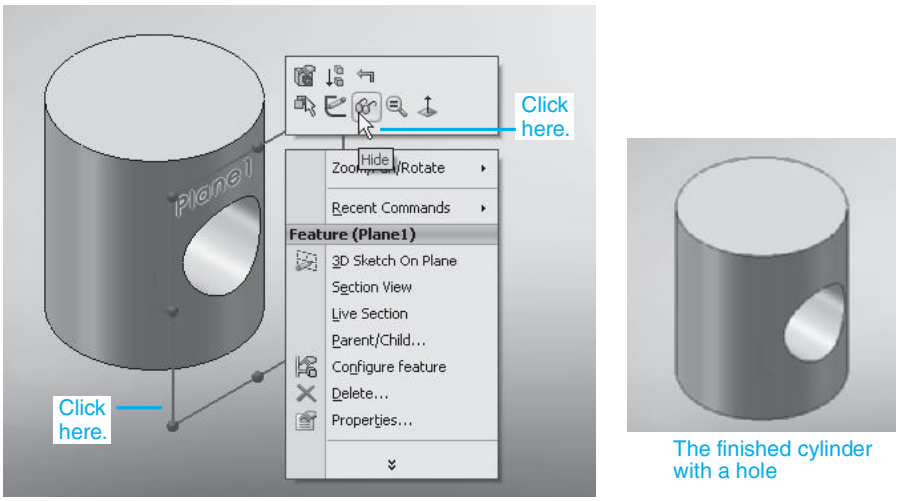


Figure 3-19 ***(continued)***

To Create a Reference Plane

1. Click the **Right Plane** tool.
2. Right-click the **Right Plane** tool and click the **Show** tool to assure that the right plane is visible.
3. Click the **Reference** tool located in the **Features** group and select the **Plane** option.

   The **Plane** box will appear. See Figure 3-19.

4. Set the offset distance for **2.00.**
5. Click one of the lines that show the right plane.
6. Click the OK check mark.

   The new reference plane is defined as **Plane 1.**

7. Right-click **Plane 1** and click the **Insert Sketch** tool.
8. Use the **Point** tool and locate a point at **X = 0.00, Y = 1.75** on Plane 1.
9. Click the **Circle** tool and draw a **Ø1.50** circle on Plane 1 using the defined point.
10. Click the **Features** tool and click the **Extruded Cut** tool.
11. Define the length of the cutting cylinder as **4.00** to assure that it passes completely through the Ø3.00 cylinder.
12. Click the OK check mark.
13. Hide Plane 1 and the right plane by right-clicking on the planes and selecting the **Hide** option.

3-12 LOFTED BOSS/BASE

The **Lofted Boss/Base** tool is used to create a shape between two planes, each of which contains a defined shape. Before drawing a lofted shape we must first draw two shapes on two different planes. In this example a square is lofted to a circle. See Figure 3-20.

1. Set the **Units** for millimeters and access a top plane.
2. Right-click the **Top Plane** heading and click the **Show** option.
3. Click the **Features** tool and click the **Reference** tool. Select the **Plane** option. (See Figure 3-19.) The **Plane** box will appear.
4. Click the existing top plane.
5. Set the distance between the planes for **60mm.**
6. Click the OK check mark.

   A new plane, **Plane 1,** will appear.

7. Right-click one of the outline lines of the top plane and select the **Sketch** option.

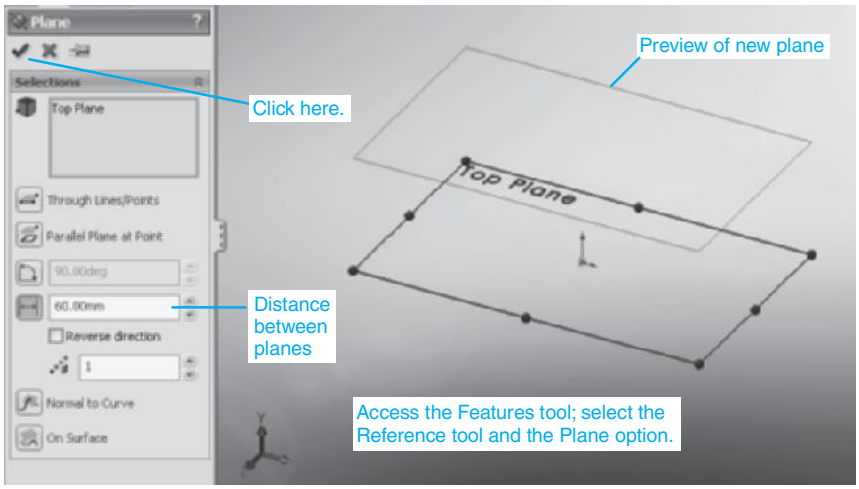
   The view orientation will automatically change to a top view.

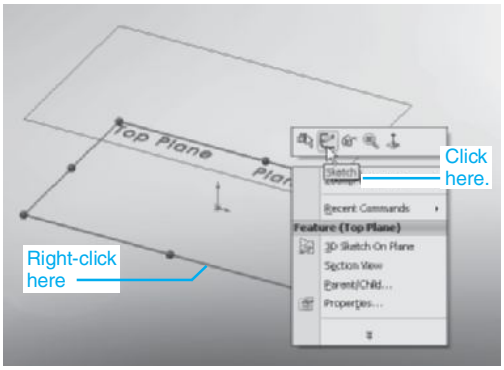
8. Sketch an approximate square about the origin. Right-click the mouse and click the **Select** option.

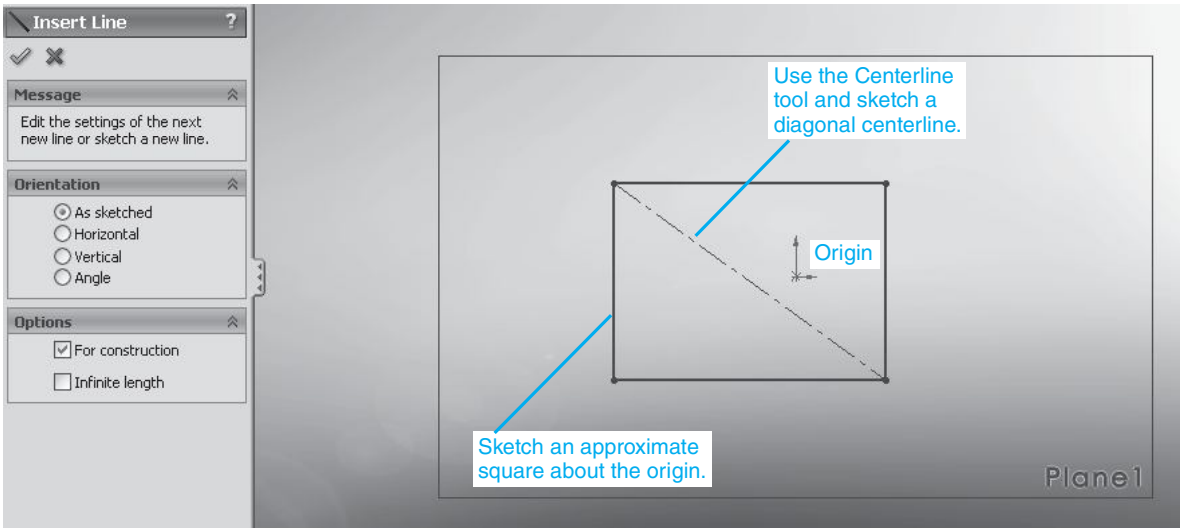
   The **Centerline** line and **Line Properties** tools will be used to center the square about the origin.

9. Access the **Centerline** tool (located with the **Sketch** tools) and draw a centerline diagonally across the approximate square. Right-click the mouse and click the **Select** option.
10. Click the origin.
11. Click and hold the **<Ctrl>** key, and click the diagonal centerline. Release the **<Ctrl>** key.
12. Click the **Midpoint** option in the **Add Relations** box.

    The midpoint of the diagonal centerline will align with the origin.


Plane
Selections
Top Plane
Through Lines/Points
Parallel Plane at Point
60.00mm
Reverse direction
Normal to Curve
On Surface
Click here.
Preview of new plane
Top Plane
Distance between planes
Access the Features tool; select the Reference tool and the Plane option.


Top Plane
Click here.
Recent Commands
Feature (Top Plane)
3D Sketch On Plane
Section View
Parent/Child...
Properties...
Right-click here


Insert Line
Message
Edit the settings of the next new line or sketch a new line.
Orientation
As sketched
Horizontal
Vertical
Angle
Options
For construction
Infinite length
Use the Centerline tool and sketch a diagonal centerline.
Origin
Sketch an approximate square about the origin.
Plane1

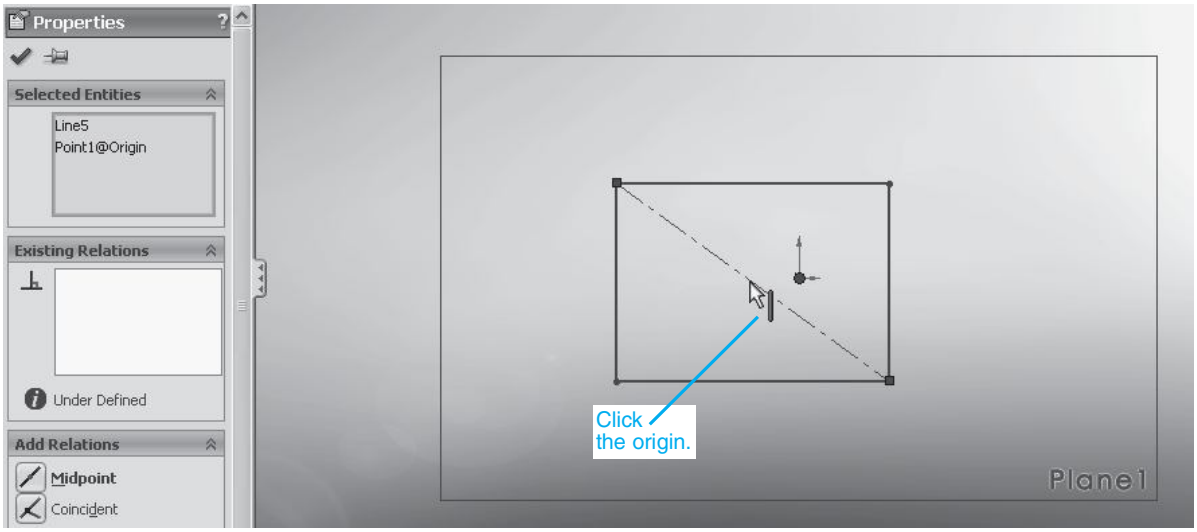
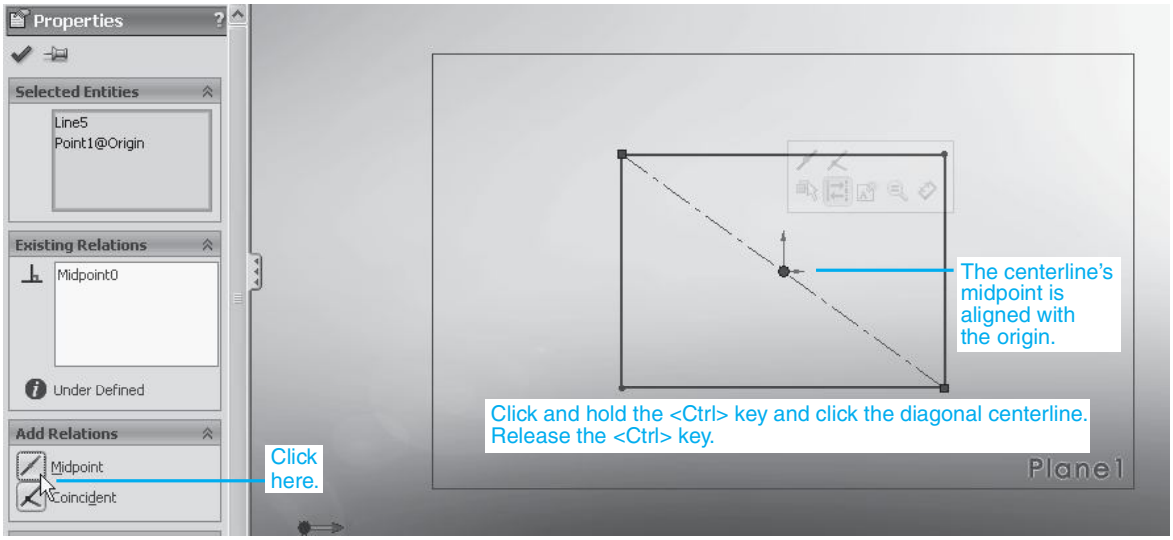
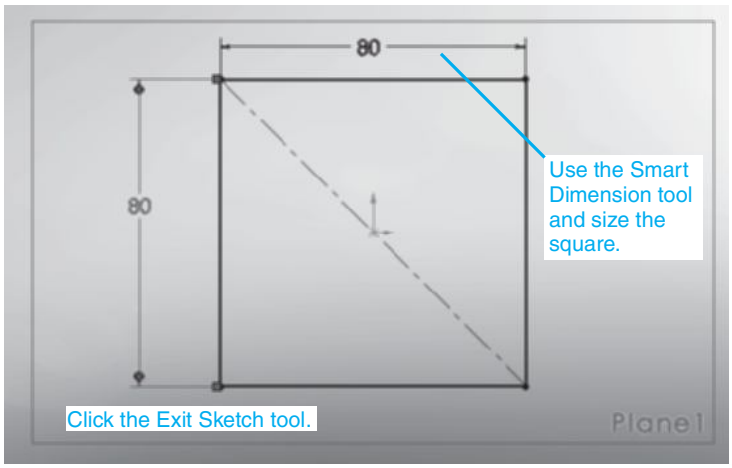
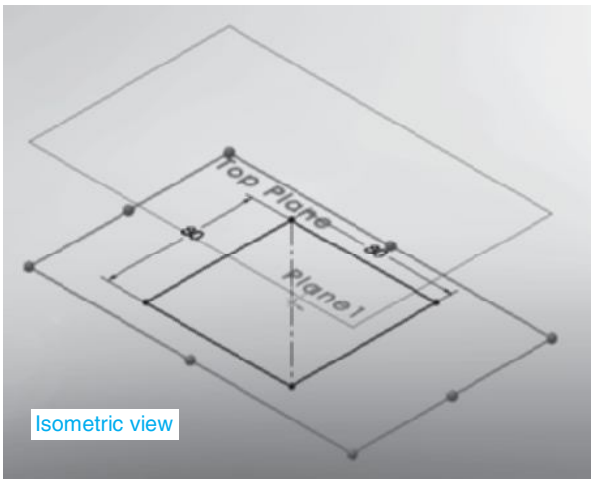

Properties
Selected Entities
Line5
Point1@Origin
Existing Relations
Under Defined
Add Relations
Midpoint
Coincident
Click the origin.
Plane1

Figure 3-20







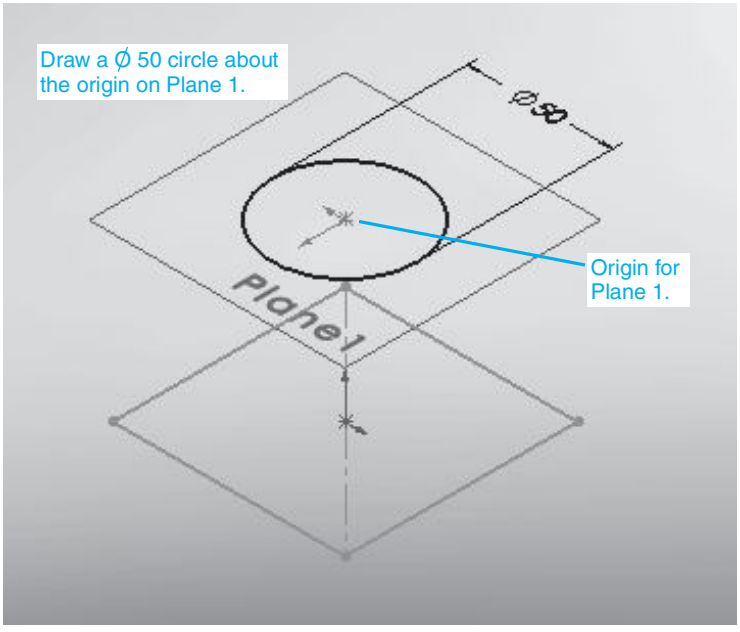
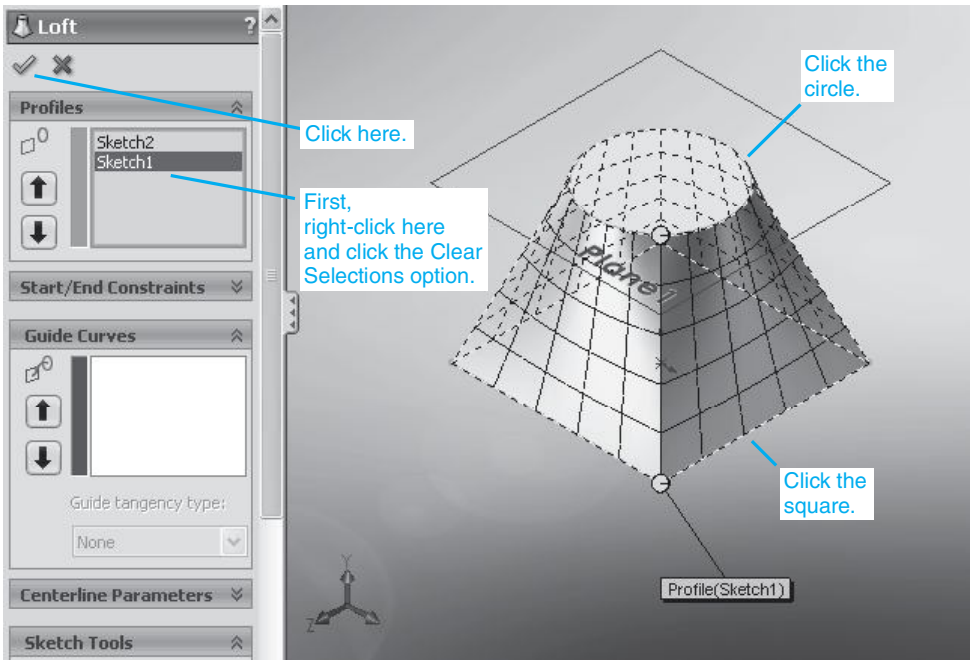


Figure 3-20 ***(continued)***



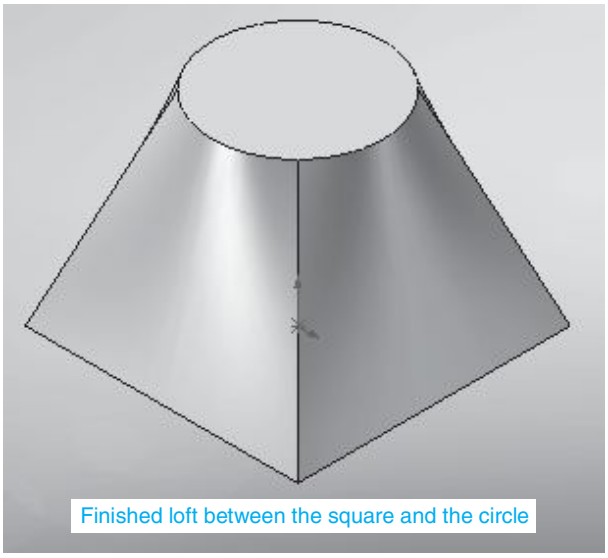


Figure 3-20 *(continued)*

13. Use the **Smart Dimension** tool and size the square.

    In this example, dimensions of **80** were used.

14. Change the view orientation to **Isometric.**
15. Exit the sketch.

TIP

You can exit a sketch either by clicking the **Exit Sketch** tool on the **Sketch** toolbar or by clicking the **Exit Sketch** tool in the upper right corner of the drawing screen.

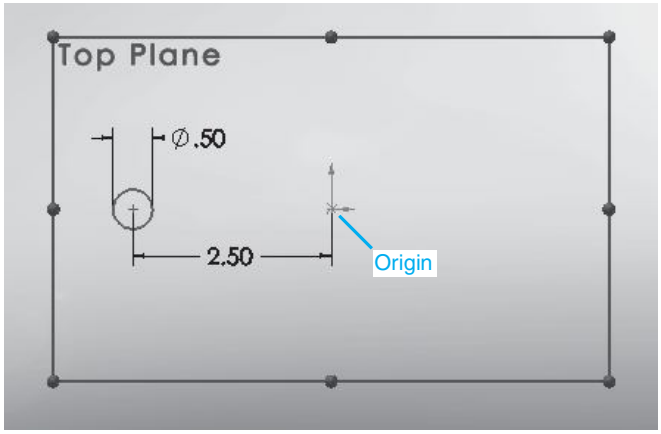
16. Right-click the **Plane 1** heading and click the **Sketch** option.

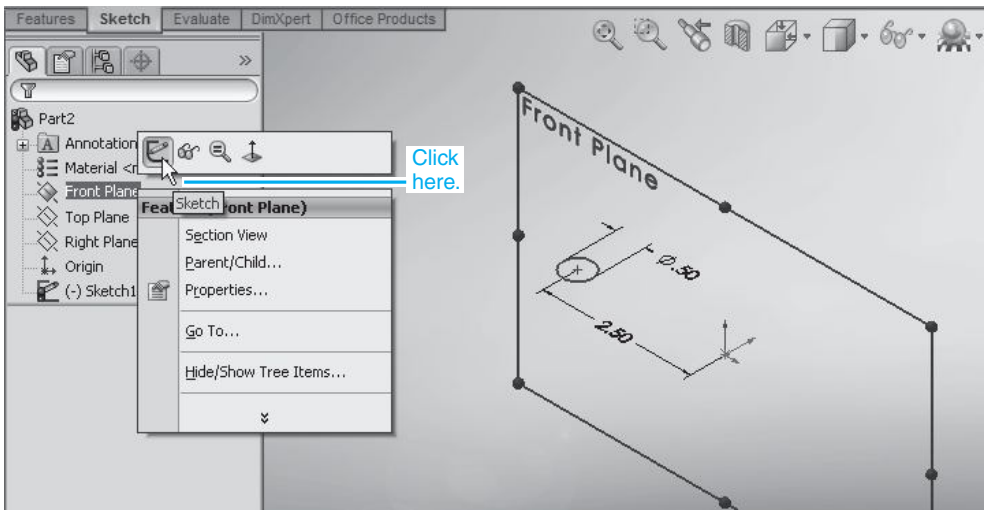
    A new origin will appear on Plane 1. We can now sketch on Plane 1.

17. Sketch a circle about the origin in Plane 1. Use the **Smart Dimension** tool and size the circle to **Ø50.**
18. Right-click the mouse and click the **Select** option. Click the **Exit Sketch** tool.

    The **Lofted Boss/Base** tool can now be applied.

19. Click the **Features** tool and click the **Lofted Boss/Base** tool.


Top Plane
Ø.50
2.50
Origin


Features
Sketch
Evaluate
DimXpert
Office Products
Part2
Annotation
Material <n
Front Plane
Top Plane
Right Plane
Origin
(-) Sketch1
Click here.
Sketch
Section View
Parent/Child...
Properties...
Go To...
Hide/Show Tree Items...
Front Plane
Ø.50
2.50

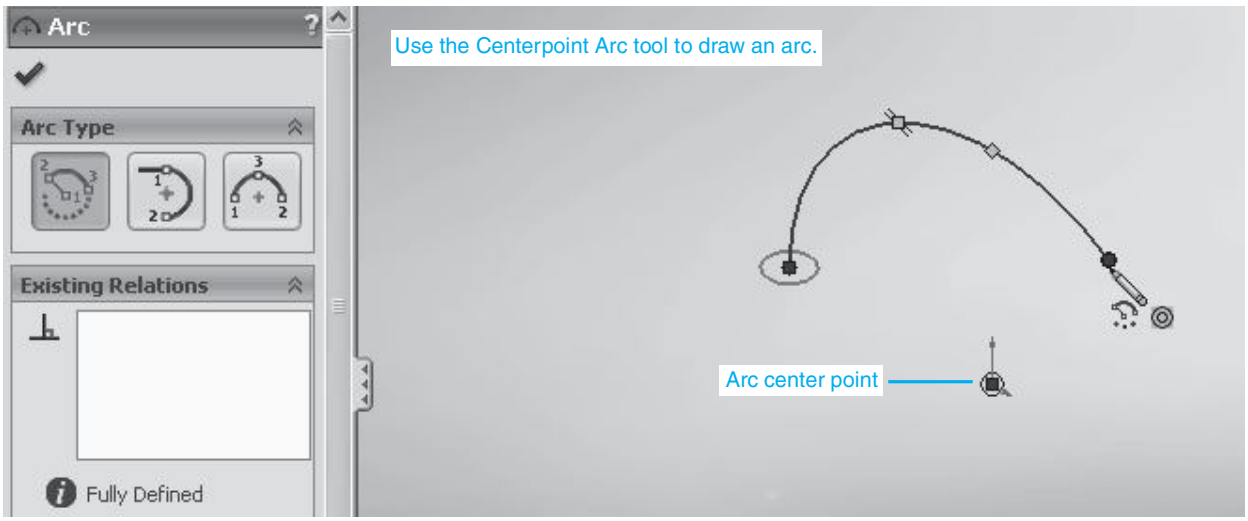

Arc
Arc Type
Existing Relations
Fully Defined
Use the Centerpoint Arc tool to draw an arc.
Arc center point

Figure 3-21

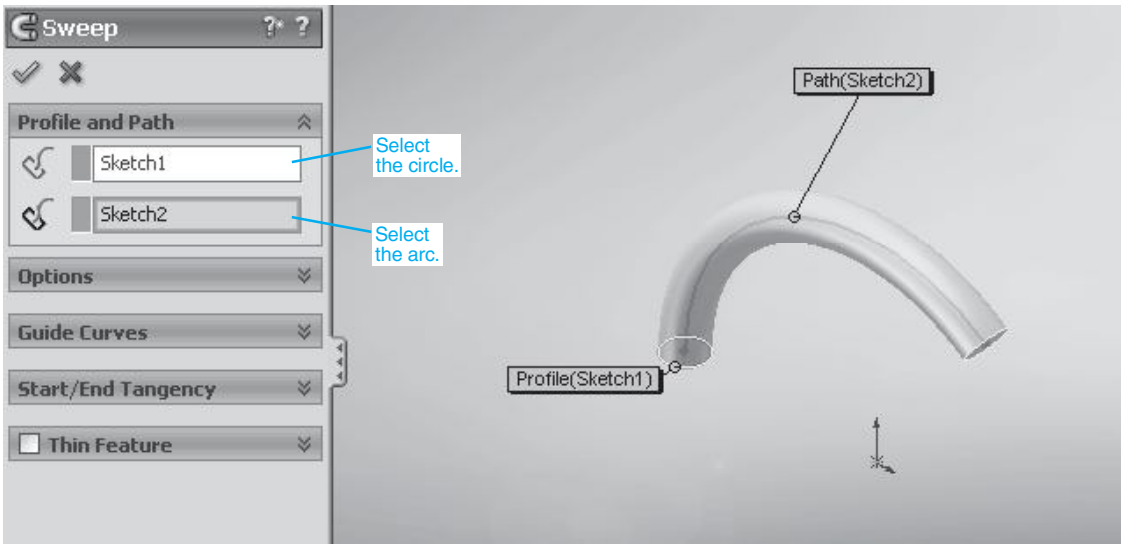


Figure 3-21 ***(continued)***

20. Right-click the **Profiles** box and click the **Clear Selections** option.
21. Click the square, then click the circle.
22. Click the OK check mark. Hide the planes.

3-13 SWEPT BOSS/BASE

The **Swept Boss/Base** tool is used to sweep a profile along a path line. As with the **Lofted Boss/Base** tool, existing shapes must be present before the **Swept Boss/Base** tool can be applied. In this example, a Ø0.50 circle will be swept along an arc with a 2.50 radius for 120°. See Figure 3-21.

1. Start a new drawing and click the **Top Plane** tool. Use the **Circle** and **Smart Dimension** tools to draw a **Ø0.50** circle **2.50** from the origin.
2. Change the drawing screen to an isometric orientation.
3. Right-click the mouse and click the **Select** option.
4. Click the **Exit Sketch** tool.

We are now going to create a new sketch on a different sketch plane, so we must exit the top plane.

5. Right-click **Front Plane** and click the **Sketch** option.
6. Use the **Centerpoint Arc** tool to draw an arc with a **2.50** radius with the origin as its center point through 120°. Click the origin, expand the arc until it intersects with the circle's center point, click the center point, and move the cursor until the arc is approximately 120°. Define the arc's length as **120°** in the **Parameters** box.
7. Click the OK check mark.
8. Click the **Exit Sketch** tool.

This completes the second sketch. The **Swept Boss/Base** tool can now be applied.

9. Click the **Features** tool, then click the **Swept Boss/Base** tool.
10. Select the circle as **Sketch 1** and the arc as **Sketch 2.**
11. Click the OK check mark.

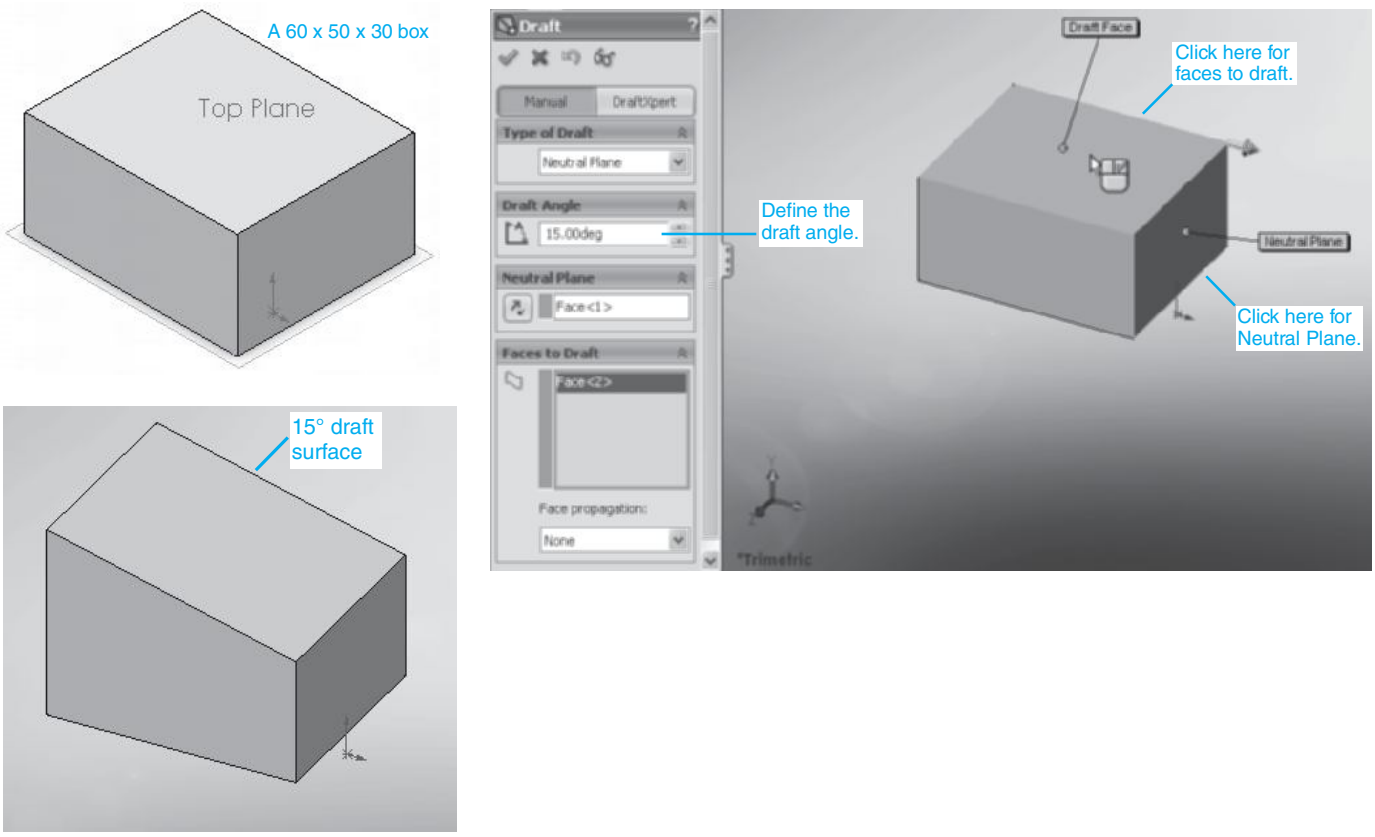


Figure 3-22

3-14 DRAFT

The **Draft** tool is used to create slanted surfaces. See Figure 3-22, which shows a 60 × 50 × 30 box. In this example a 15° slanted surface will be added to the top surface.

1. Click the **Draft** tool.
2. Select the **Neutral Plane** by clicking the right vertical face of the box.
3. Define the **Draft Angle** as **15°.**
4. Select the draft face by clicking the top surface of the box.

The draft angle will be applied to the draft face relative to the 90° angle between the two faces.

5. Click the OK check mark.

Figure 3-23 shows a slanted surface created by making the top surface the neutral plane and the front surface the draft plane.

3-15 LINEAR SKETCH PATTERN

The **Linear Sketch Pattern** tool is used to create rectangular patterns based on a given object.

Figure 3-24 shows a 15 × 20 × 10 box located on an 80 × 170 × 5 base. The box is located 10 from each edge of the base as shown.

1. Click the **Linear Sketch Pattern** tool.

> ***Note:***
> The box should be selected automatically, but if is not, use the **Features to Pattern** tool to select the box.

2. Define **Direction 1** by clicking the back top line as shown.
3. Define the spacing as **30.00.**

Spacing is the distance between two of the objects in the pattern as measured from the same point on each object, for example, the distance from the lower front corner on one object to the lower front corner of the next object.

4. Define the number of columns in the pattern.
5. Define **Direction 2,** the spacing for **Direction 2,** and the number of rows in the pattern.

A preview of the pattern will appear.

6. Click the OK check mark.

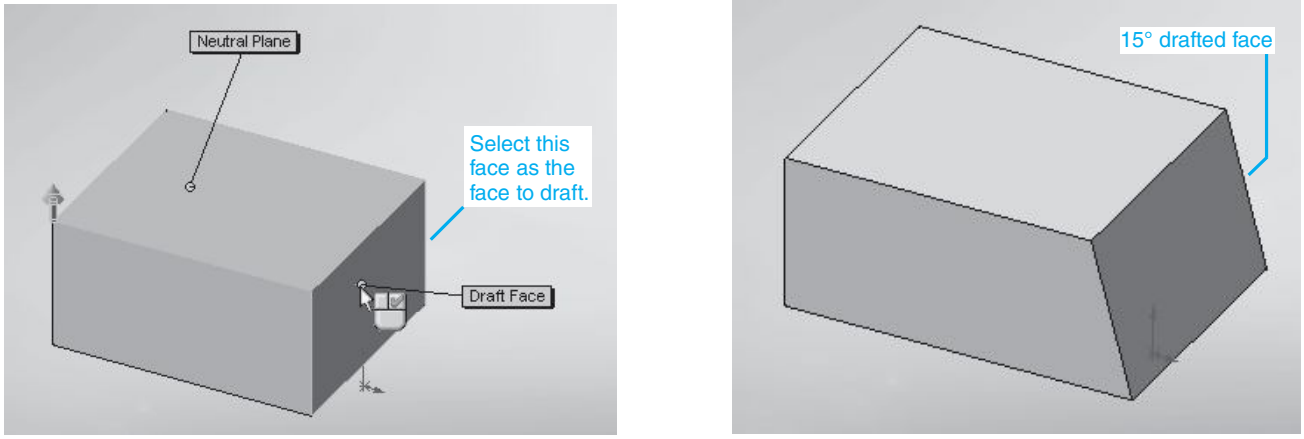


Figure 3-23

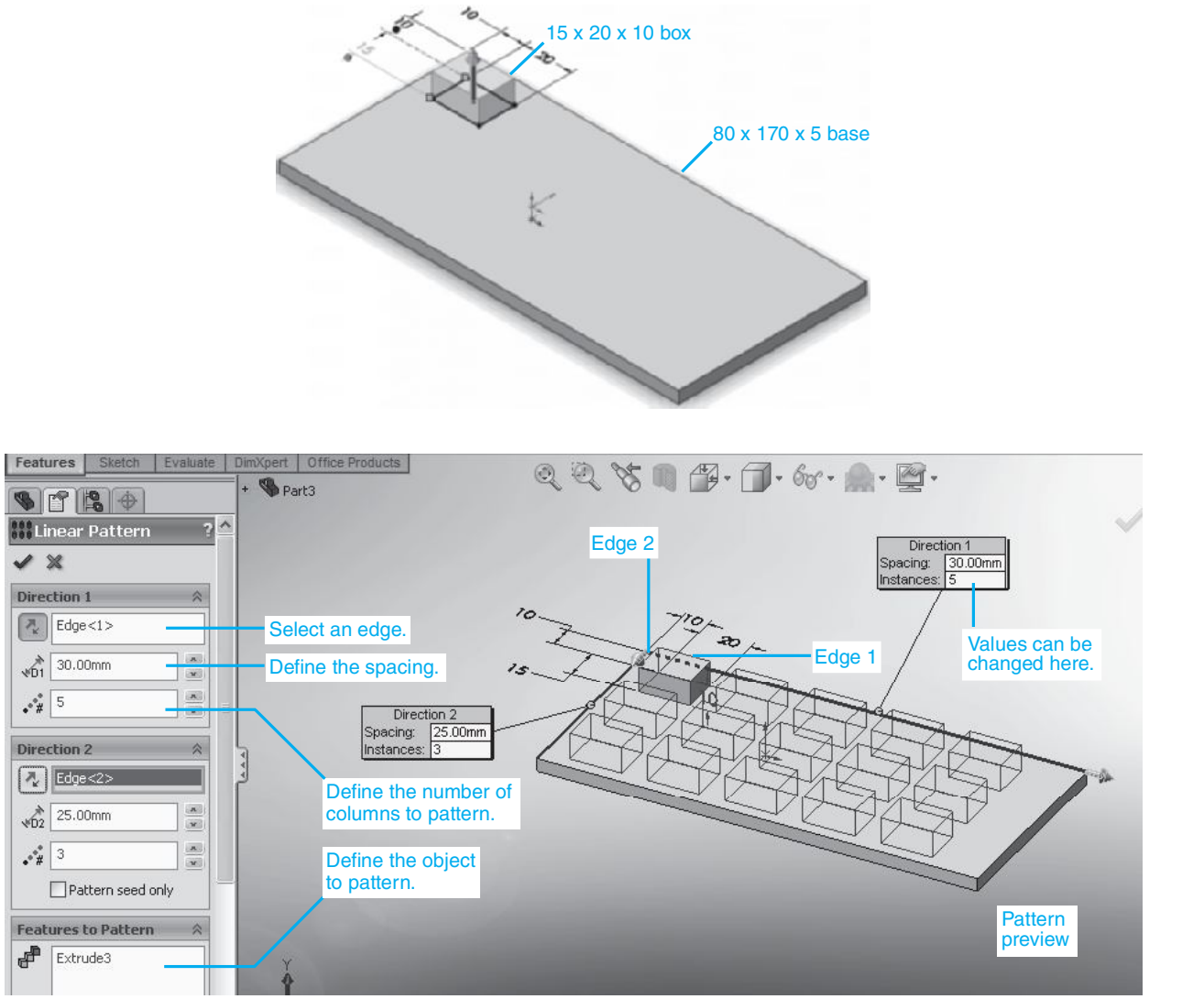


Figure 3-24

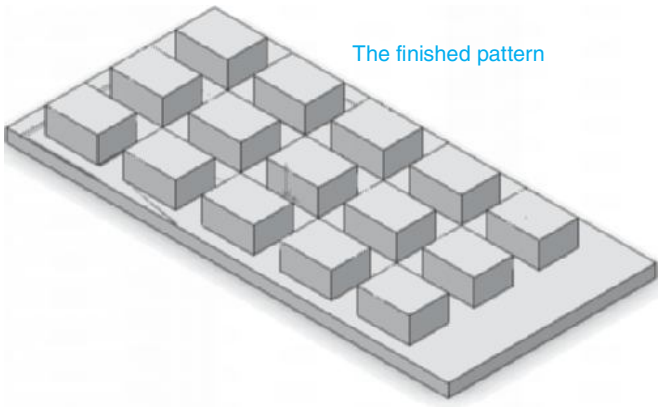


Figure 3-24 ***(continued)***

3-16 CIRCULAR SKETCH PATTERN

The **Circular Sketch Pattern** tool is used to create circular patterns about an origin. See Figure 3-25.

1. Create a **Ø160 × 10** cylinder.
2. Create a **Ø30** hole centered about the Ø160 cylinder's origin.
3. Define an axis for the Ø30 hole by accessing the **Reference** tool in the **Features** tools and then clicking the **Axis** option.
4. Click the **Cylindrical/Conical Face** option in the **Axis Properties Manager** and click the inside surface of the Ø30 hole.

   An axis should appear in the center of the hole.
5. Right-click the top surface of the cylinder and click the **Sketch** option.
6. Sketch a **Ø20** circle **60** from the surface's center point.

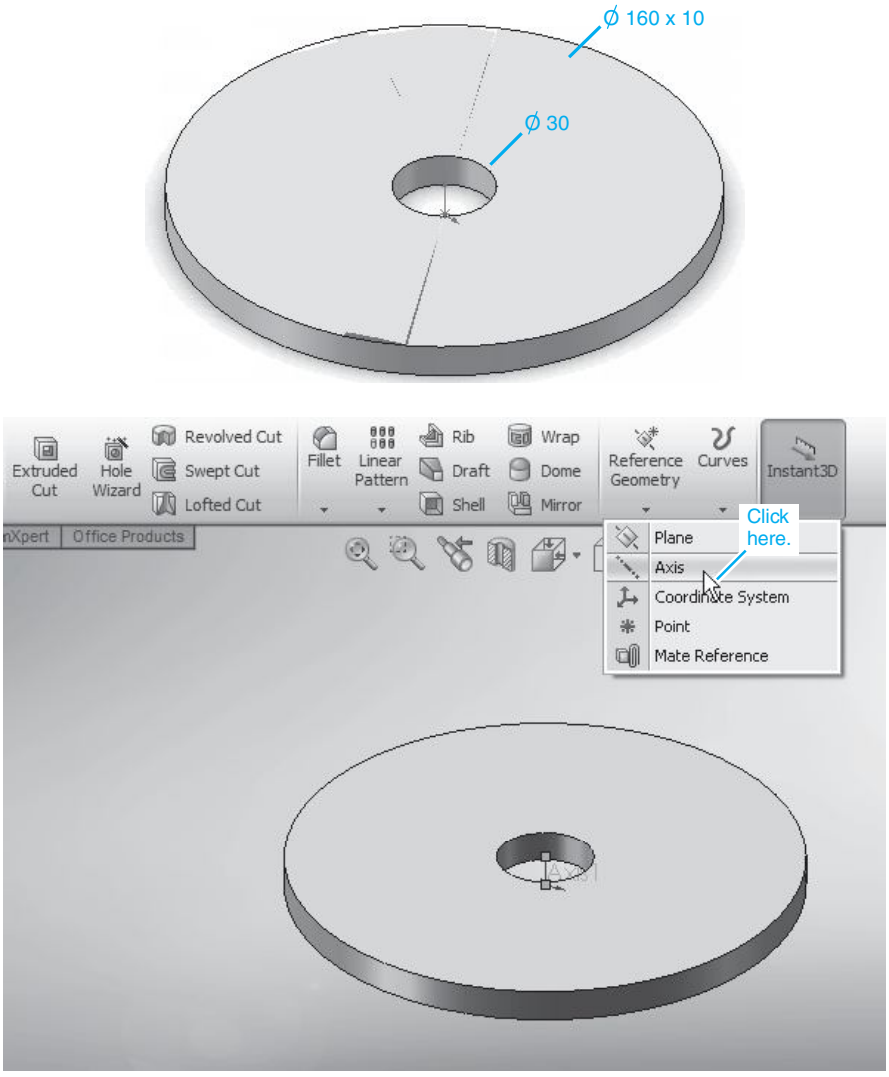
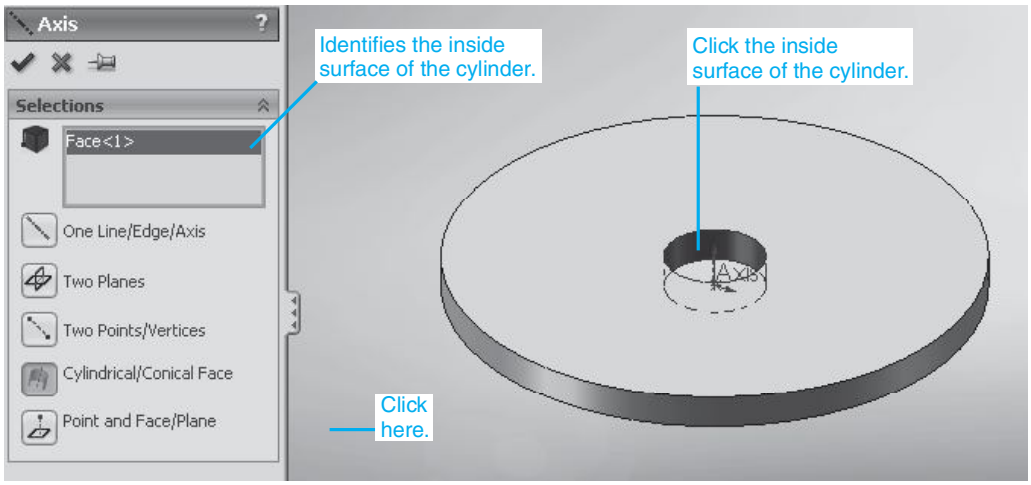
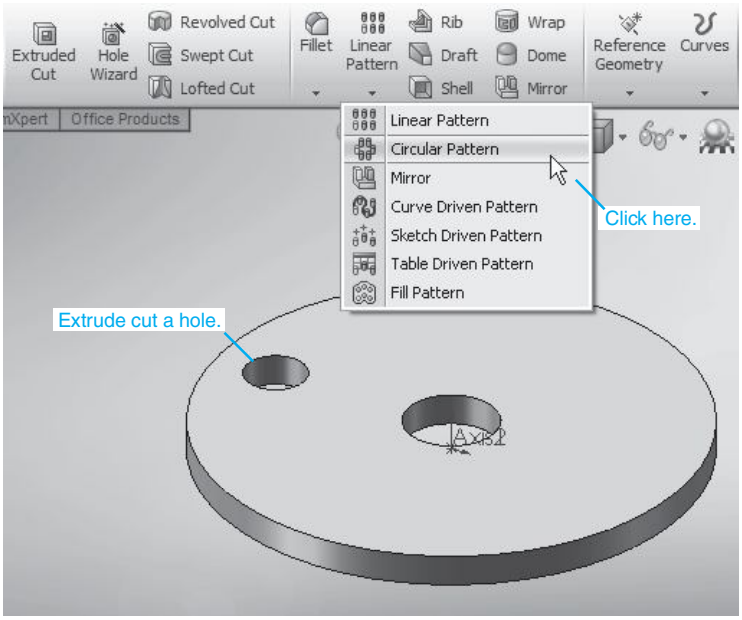
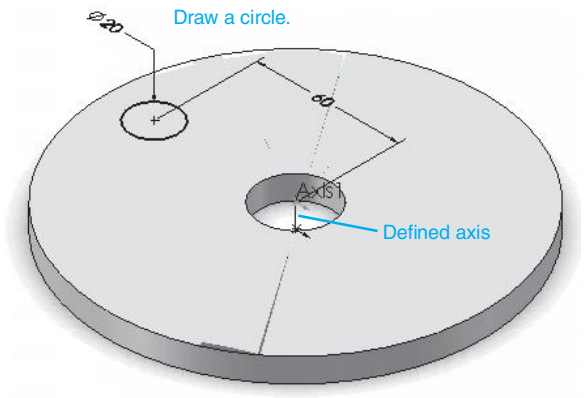
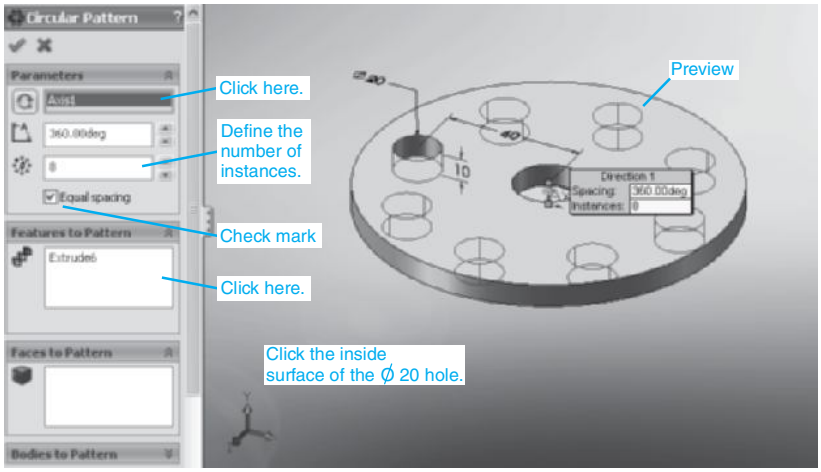


Figure 3-25









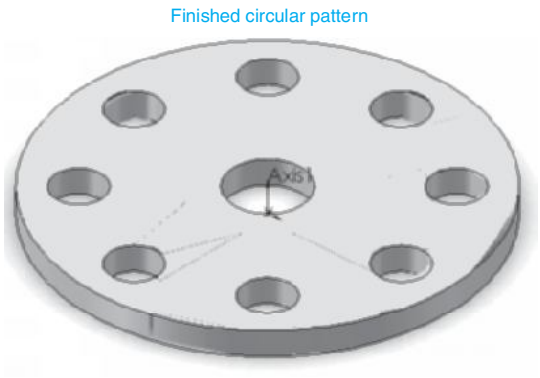


Figure 3-25 ***(continued)***

7. Access the **Features** tools and select the **Circular Pattern** tool.
8. Define the **Number of features** in the pattern for **8**, click the **Equal spacing** option, select the axis of the Ø30 hole as the axis and the Ø20 hole as the **Features to Pattern.**

   A preview will appear.
9. Click the OK check mark.

3-17 MIRROR

The **Mirror** tool is used to create mirror images of features. A mirror image is not the same as a copy. In this section we will mirror the object shown in Figure 3-26.

1. Access the **Features** tools, and click the **Mirror** tool.
2. Click the **Features to Mirror** box, then window the entire object.

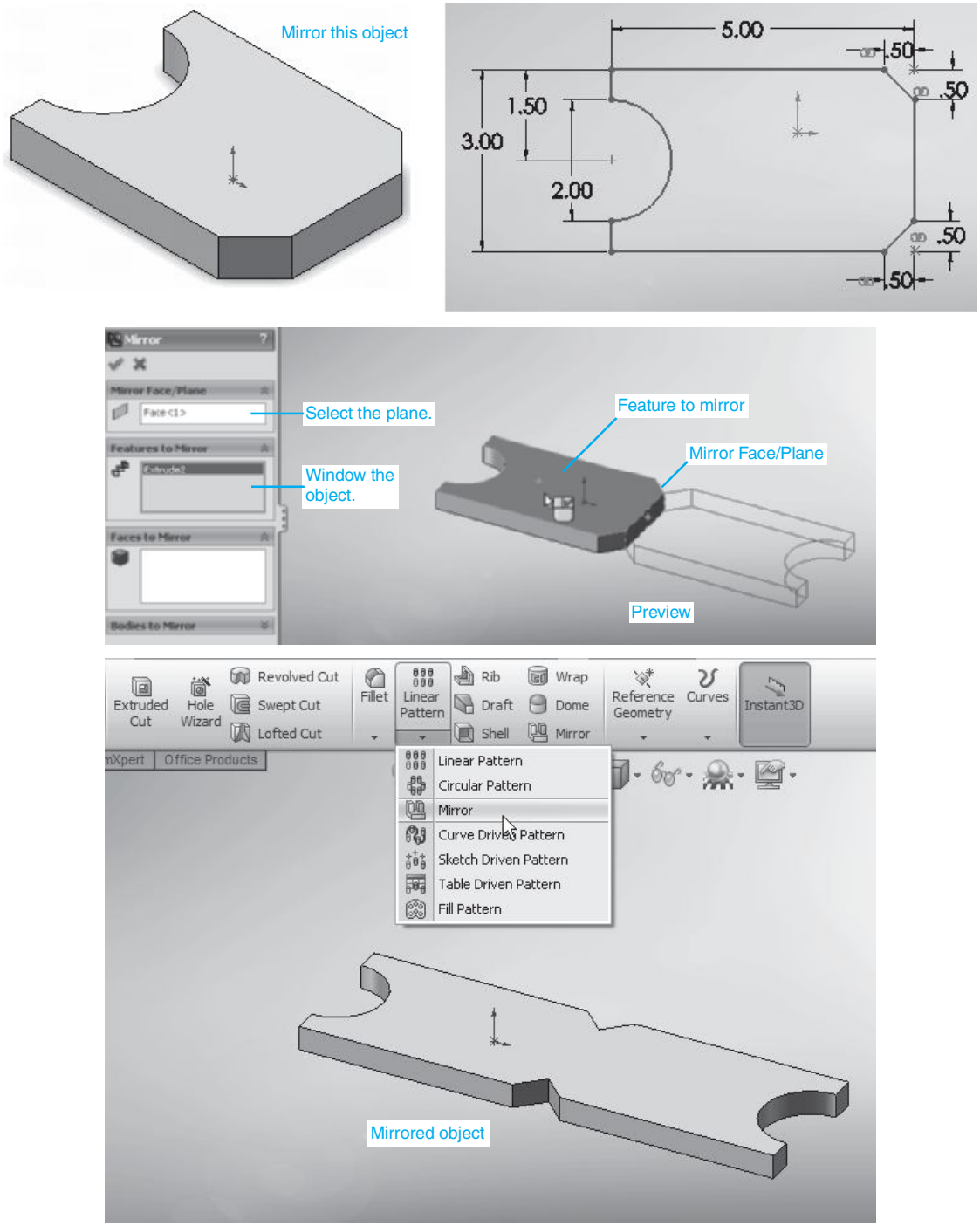


Figure 3-26

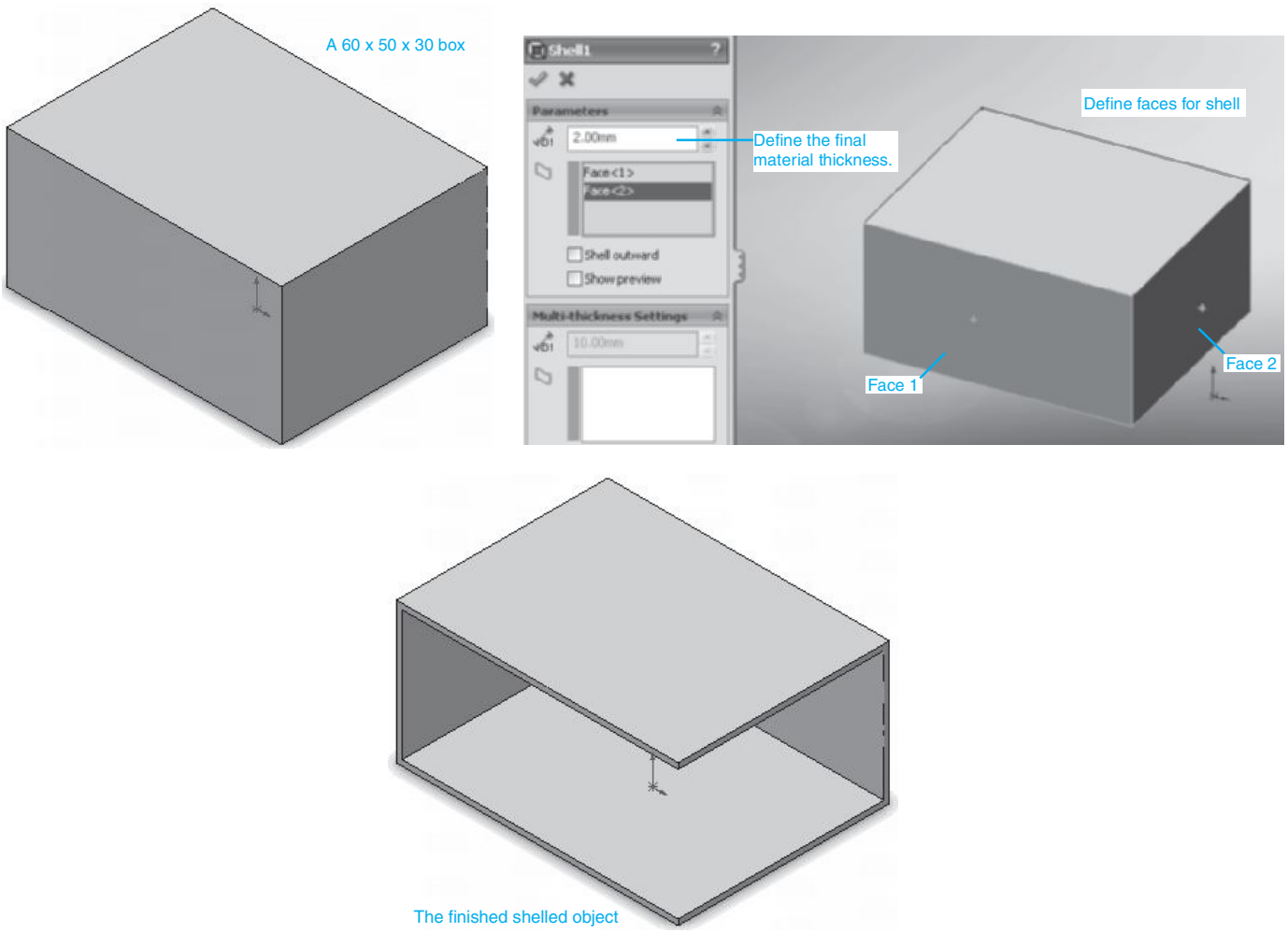


Figure 3-27

3. Click the **Mirror Face/Plane** box, then click the right edge plane as shown.

   A preview will appear.

4. Click the OK check mark.

3-18 SHELL

The **Shell** tool is used to hollow out an existing object, making it into a thin-walled object. See Figure 3-27.

1. Draw a 60 × 50 × 30 box. Use the top plane and an isometric orientation.
2. Access the **Features** tools, and click the **Shell** tool.
3. Define the final material thickness.

   In this example a value of **2.00** was entered.

4. Define the faces to be shelled. Click the **Faces to Remove** box, click the left front face and then the right front face as shown.
5. Click the OK check mark.

3-19 EDITING FEATURES

SolidWorks allows you to edit existing models. This is a very powerful feature in that you can easily make changes to a completed model without having to redraw the entire model.

TIP

The **Edit Sketch** tool is used to edit shapes created using the **Sketch** tools such as holes. The **Edit Features** tool is used to edit shapes created using the **Features** tools such as a cut or extrusion.

Figure 3-28 shows the L-bracket originally created in Sections 3-4 through 3-7. The finished object may be edited. In this example, the hole's diameter and the size of the cutout will be changed.

To Edit the Hole

See Figure 3-29.

1. Right-click the **Ø20.0** hole callout in the **Features Manager** on the left side of the drawing screen and select the **Edit Feature** option.

TIP

The hole will be highlighted when selected.

The **Hole Specification Properties Manager** will appear.

2. Select a new hole diameter.

   In this example **Ø24** was selected.

3. Click the OK check mark.

To Edit the Cutout

See Figure 3-30.

1. Right-click the **Cut-Extrude 1** callout in the **Features Manager** box on the left side of the drawing screen, and select the **Edit Sketch** option.

   The cutout will be highlighted when selected.
   The sketch used to define the cutout will appear.

2. Double-click the two **20** dimensions that define the length of the cutout and change their value to **30.**
3. Click the OK check mark.
4. Click the **Exit Sketch** tool.

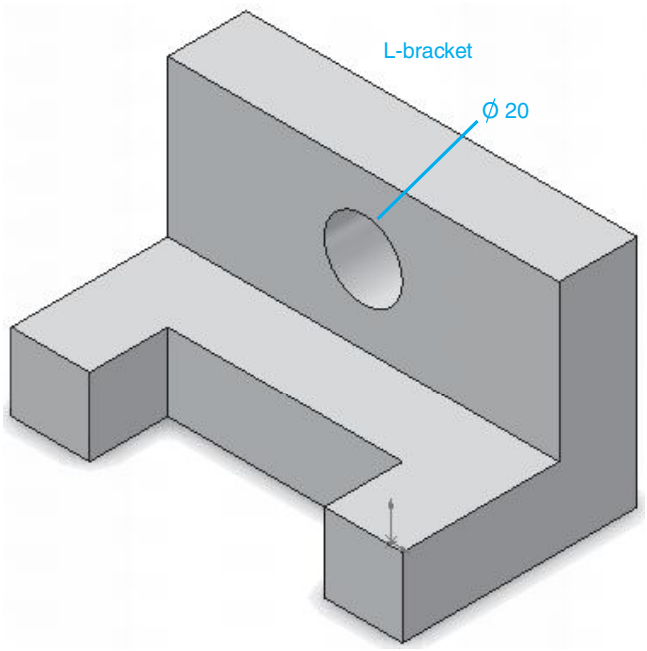


Figure 3-28

3-20 SAMPLE PROBLEMS SP3-1

Figure 3-31 shows a cylindrical object with a slanted surface, a cutout, and a blind hole. Figure 3-32 shows how to draw the object. The procedures presented in Figure 3-32 represent one of several possible ways to create the object.

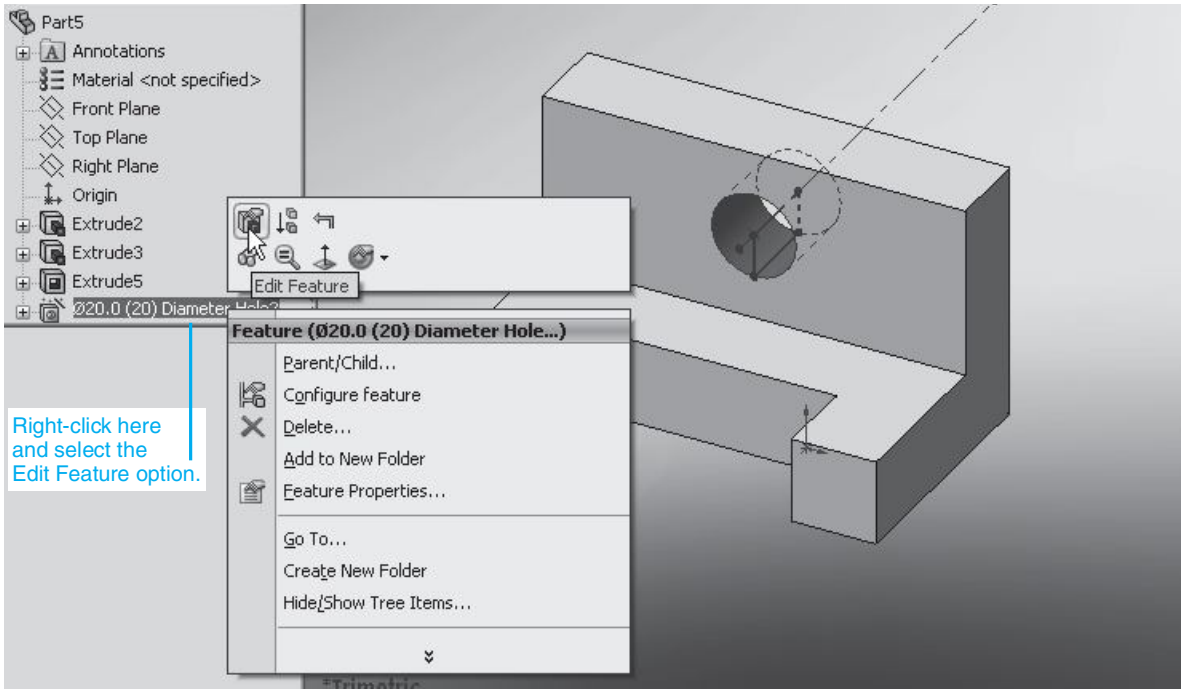


Figure 3-29

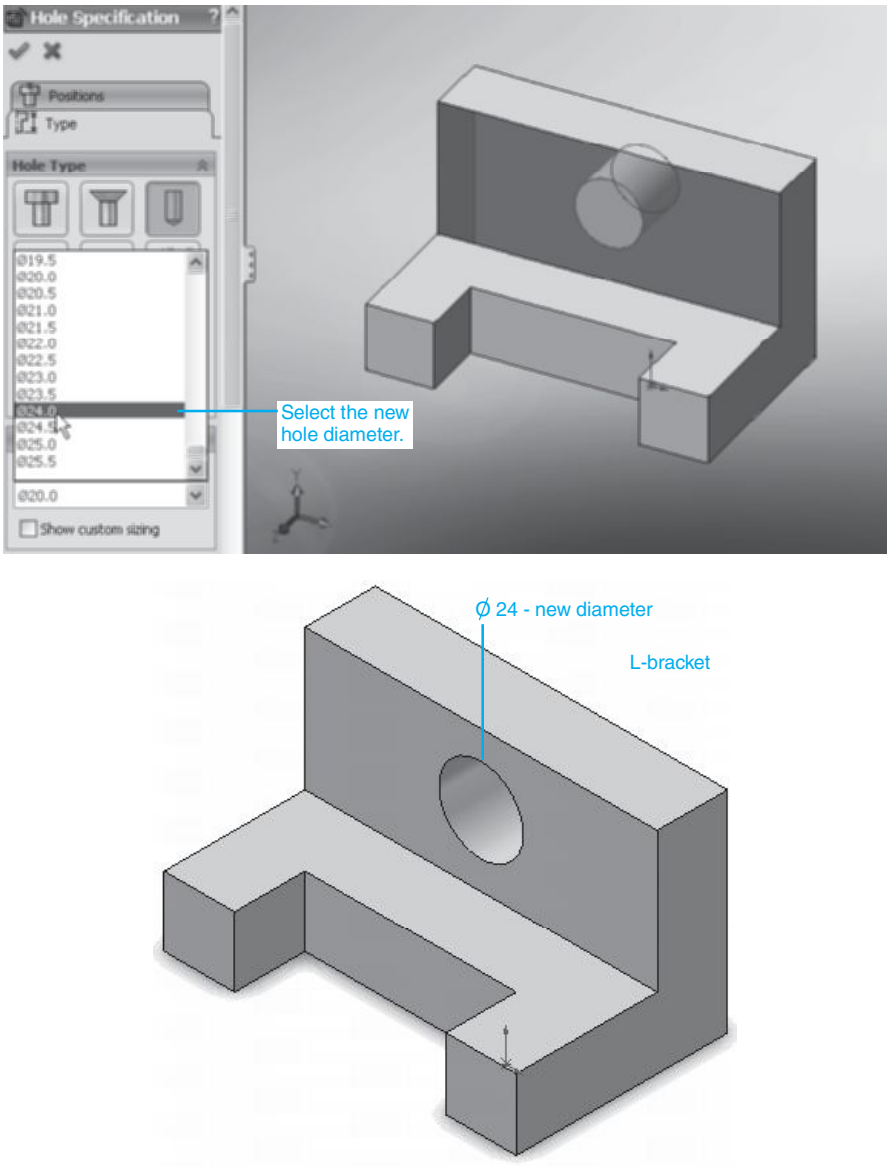


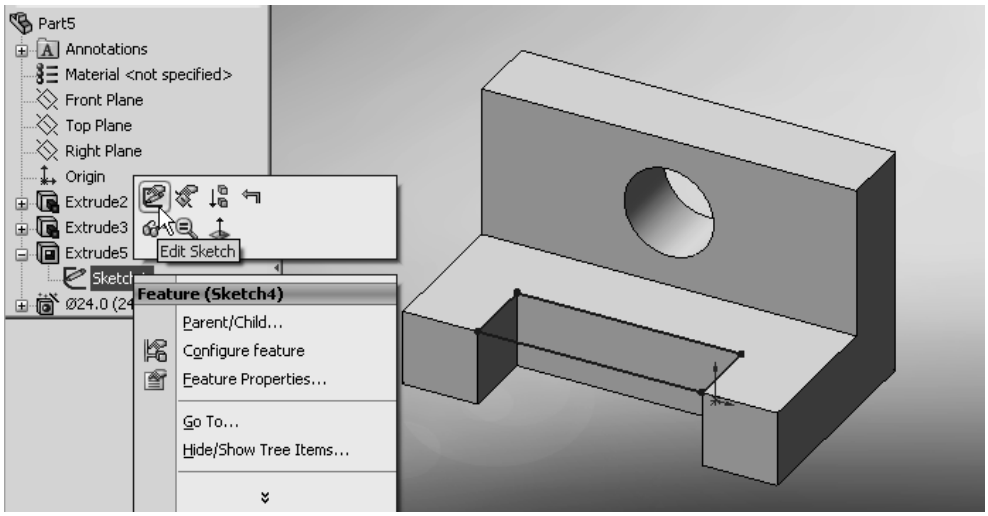
Figure 3-29 *(continued)*

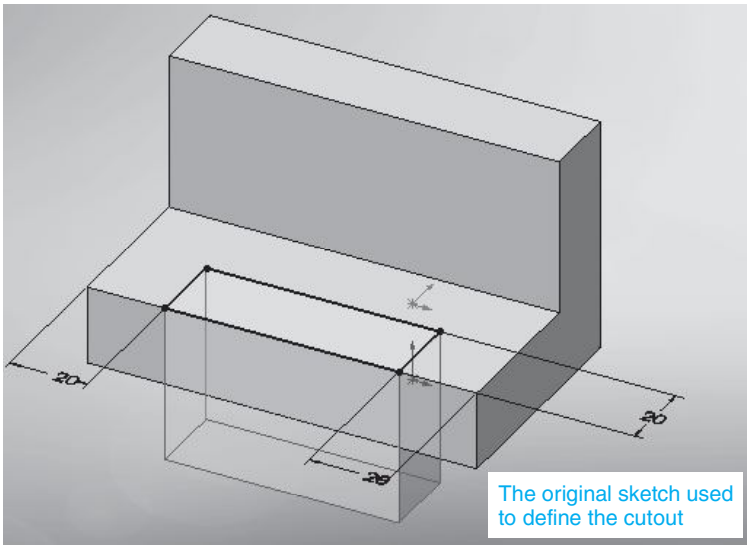
To Draw a Cylinder

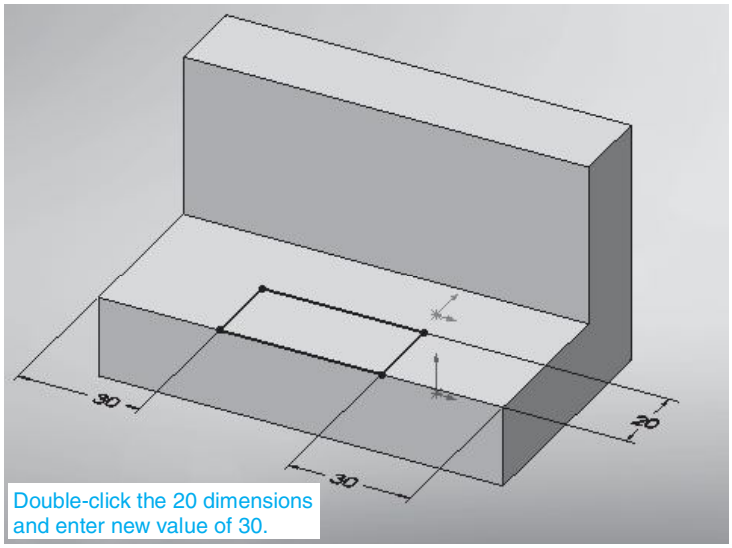
1. Start a new **Part** document.
2. Define the units as **millimeters** and access the top plane.
3. Draw a **Ø58** circle and extrude it to **60.**

To Create a Slanted Surface on the Cylinder

1. Click the **Right Plane** tool, access the **Reference** tool in the **Features** tools, and click the **Plane** option.
2. Define the offset plane distance in the **Plane Properties Manager** as **30,** and click the check mark.
3. Right-click the offset plane and select the **Sketch** option.
4. Select the **Right Plane** orientation.
5. Use the **Line** tool and draw an enclosed triangular shape.
6. Use the **Smart Dimension** tool to define the size and location of the triangle.
7. Change the drawing's orientation to **Dimetric** and click the **Extrude Cut** tool in the **Features** tools.
8. Set the length of the cut to **60.00mm** and click the check mark.
9. Right-click the offset plane and click the **Hide** option.


Part5
Annotations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
Extrude2
Extrude3
Extrude5
Sketch
Ø24.0 (24
Edit Sketch
Feature (Sketch4)
Parent/Child...
Configure feature
Feature Properties...
Go To...
Hide/Show Tree Items...


20
20
20
The original sketch used to define the cutout


30
30
20
Double-click the 20 dimensions and enter new value of 30.

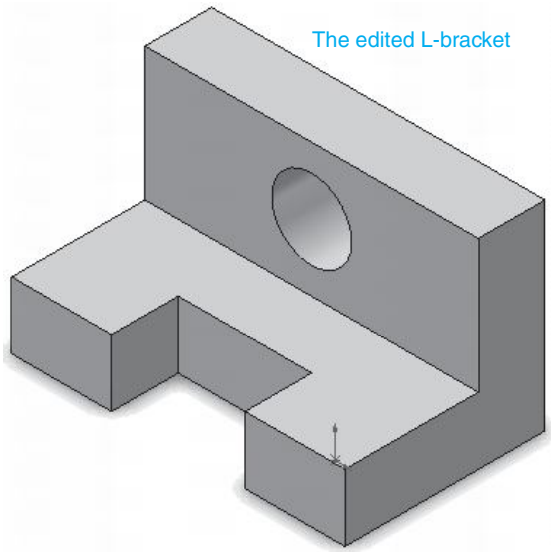

The edited L-bracket

Figure 3-30

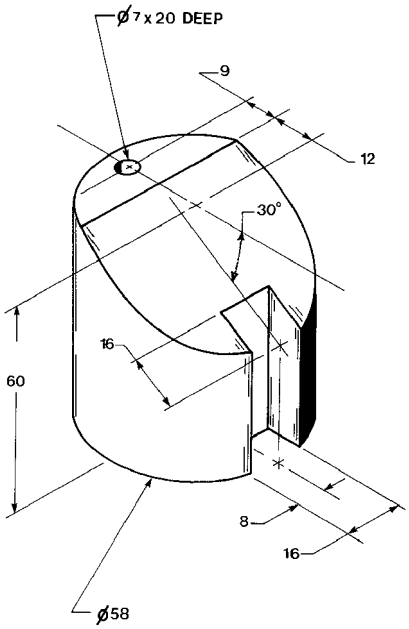


Figure 3-31

To Add the Vertical Slot

See Figure 3-33.`

1. Right-click the slanted surface and click the **Sketch** tool.
2. Click the **Custom** tool next to the axis orientation icon and click **Normal to view.**
3. Draw a vertical line through the origin. Start the line on the edge of the slanted surface.
4. Use the **Rectangle** tool on the **Sketch** toolbar and draw an **8 × 16** rectangle as shown. Use the **Smart Dimension** tool to size the rectangle.
5. Draw a second **8 × 16** rectangle as shown.
6. Change the drawing orientation to a dImetric view.
7. Exit the sketch.
8. Click the **Top Plane** tool and use the **Reference** tool on the **Features** toolbar and create an offset top plane **60** from the base of the cylinder.

Note:

The **Extruded Cut** tool will extrude a shape perpendicular to the plane of the shape. In this example the plane is slanted, so the extrusion would not be vertical, as required. The rectangle is projected into the top offset plane and the extrusion tool applied there.

9. Change the drawing orientation to the top view.
10. Sketch a rectangle on the offset plane over the projected view of the 16 × 16 rectangle on the slanted plane.
11. Change the drawing orientation to a dimetric view.
12. Use the **Extruded Cut** tool on the **Features** toolbar to cut out the slot.
13. Hide the 60 offset plane and hide the 16 × 16 rectangle on the slanted surface.
14. Click the check mark.

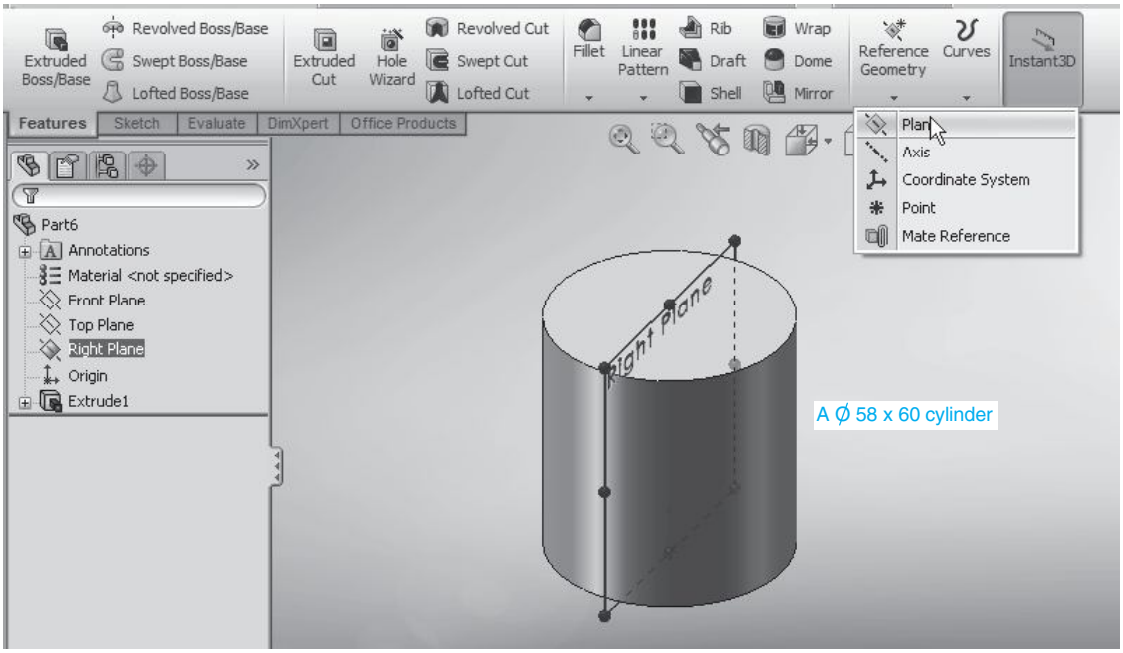
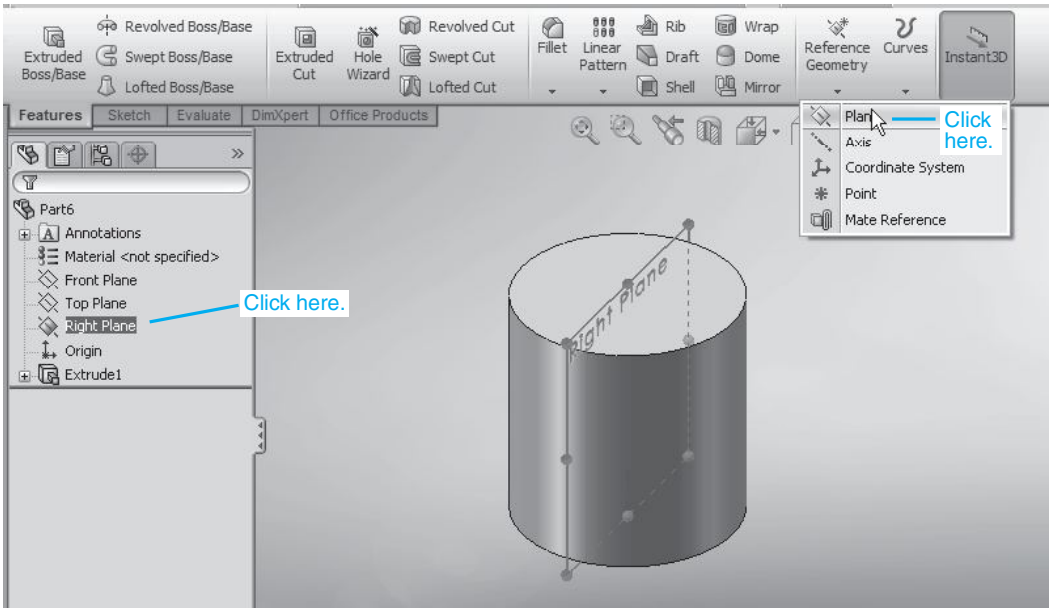
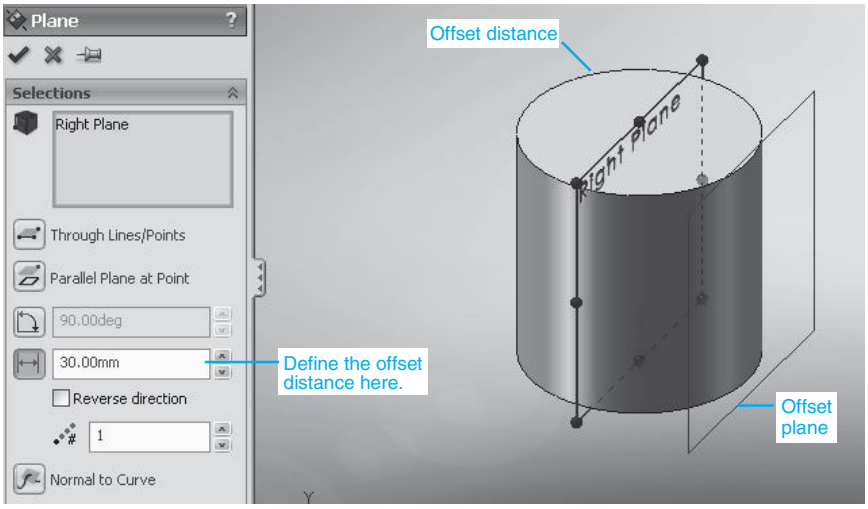
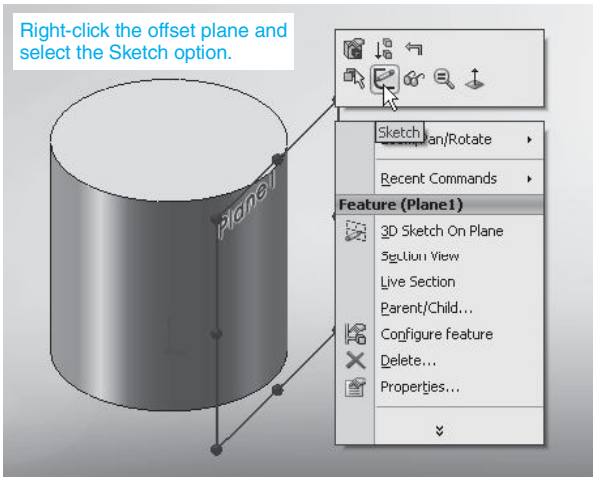


Figure 3-32







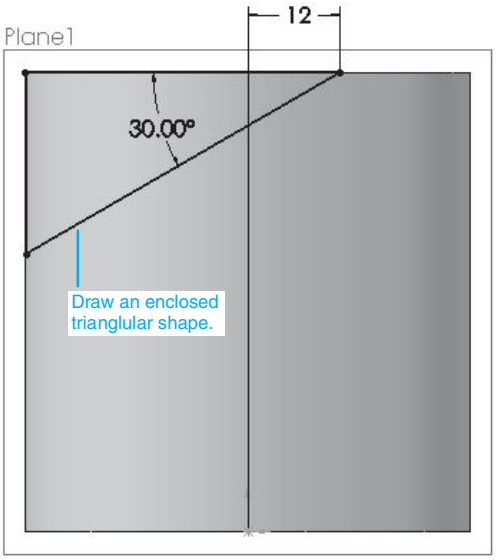


Figure 3-32 ***(continued)***

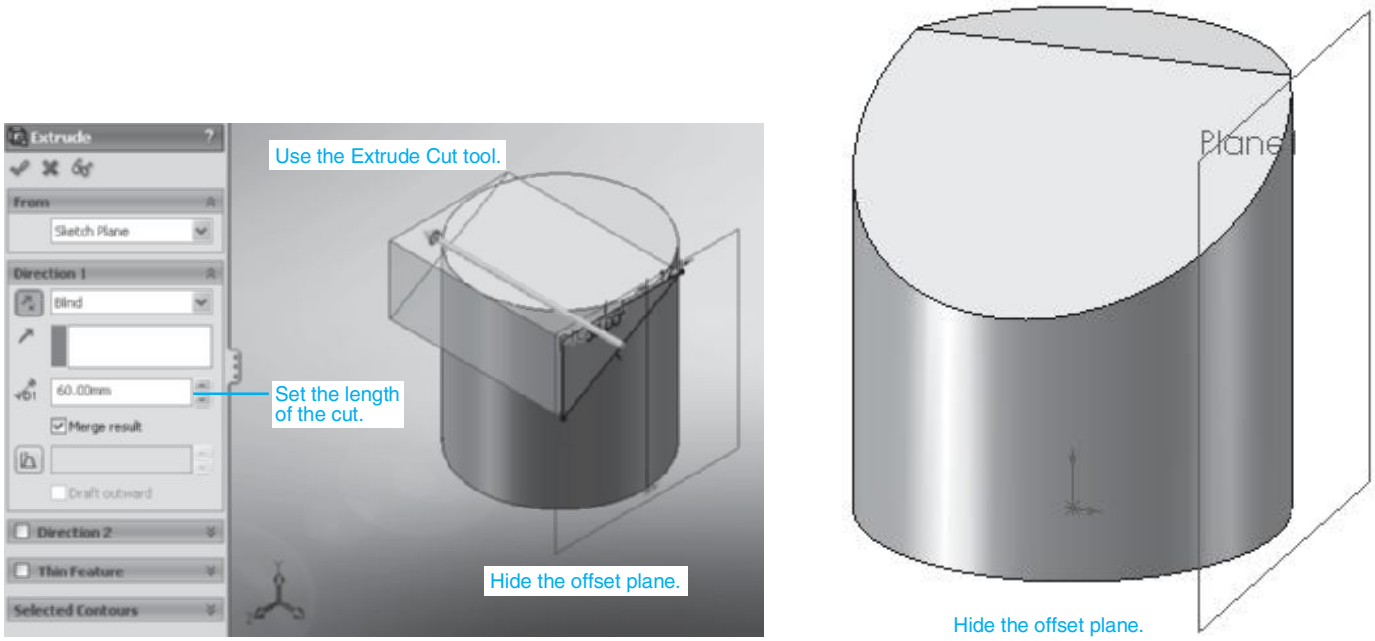


Figure 3-32 ***(continued)***

To Add the Ø8 Hole

See Figure 3-34.

1. Use the **Point** tool and sketch a point on the flat portion of the top surface. Use the origin to center the point.
2. Use the **Smart Dimension** tool and locate the point according to the given dimensions.
3. Exit the sketch.

Note:

There are two ways to draw blind holes (holes that do not go all the way through): draw a circle and use the **Extruded Cut** tool to remove material, or use the **Hole Wizard.** In this example the **Hole Wizard** tool is used because it will generate a conical-shaped bottom to the hole. Conical-shaped hole bottoms result from using a twist drill, which has a conical-shaped cutting end.

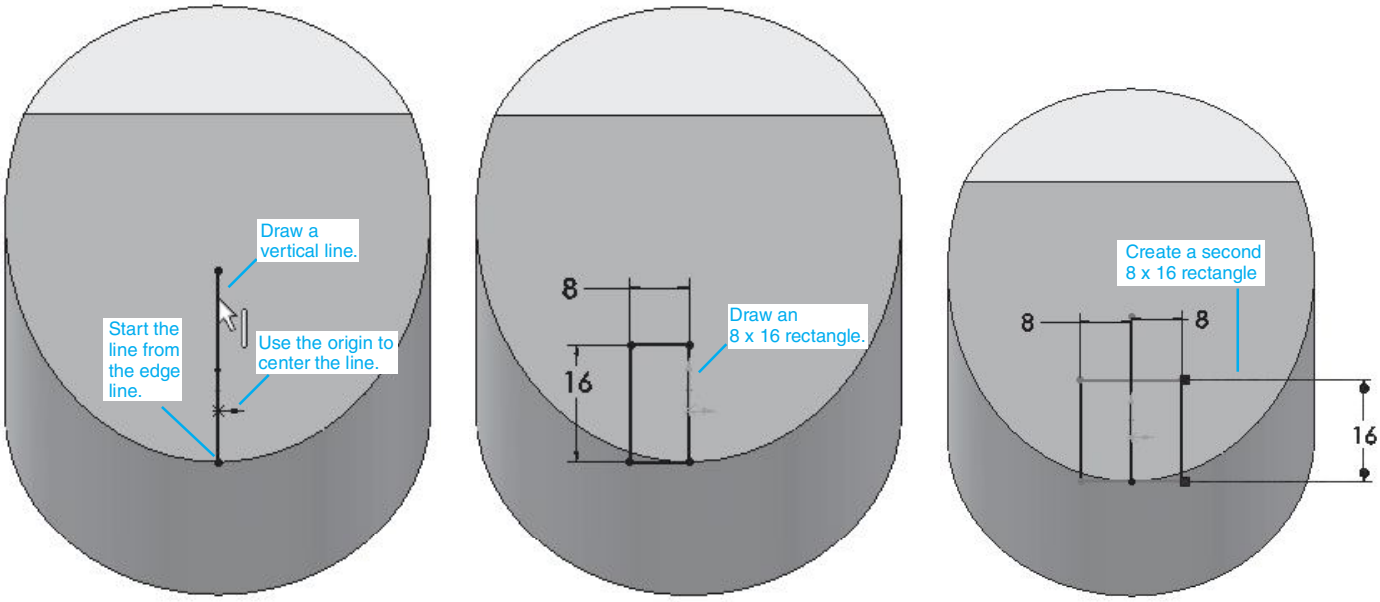
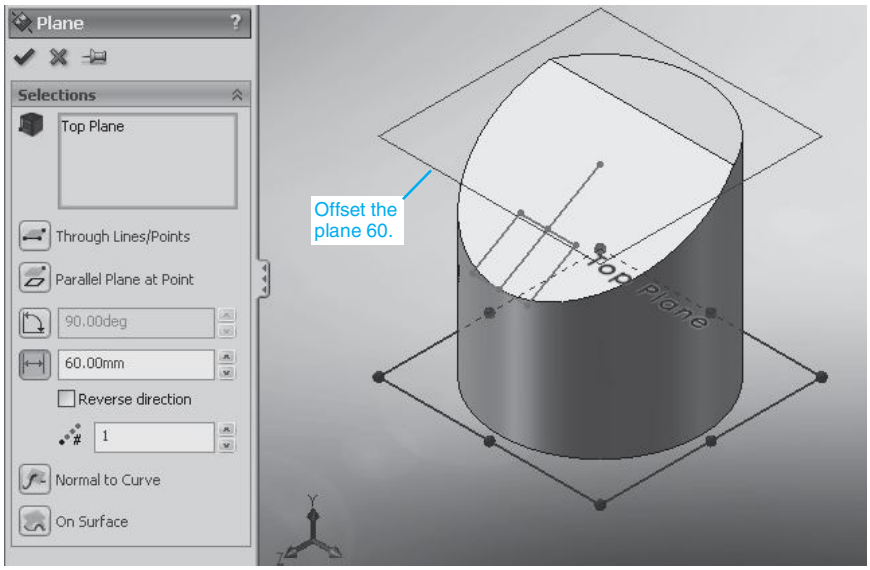
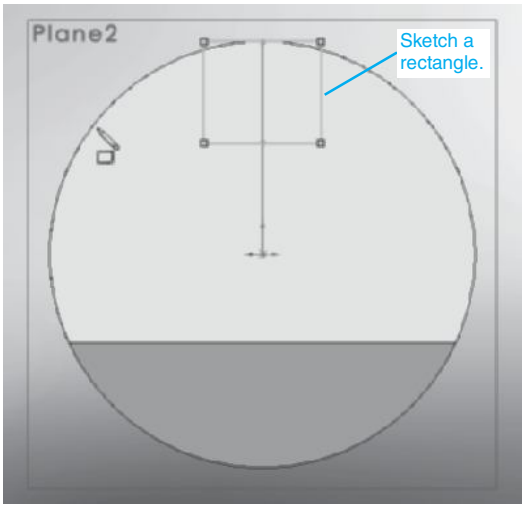
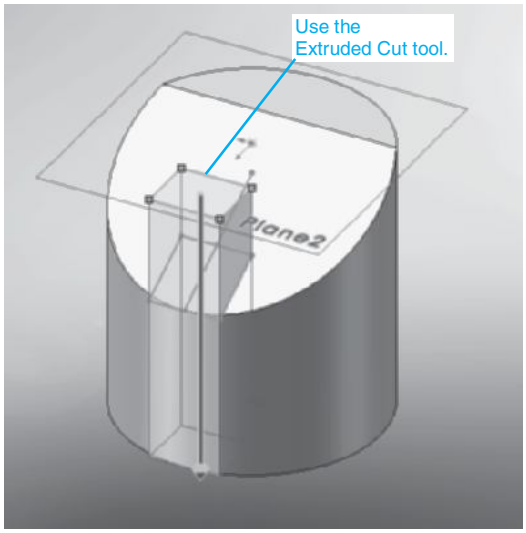
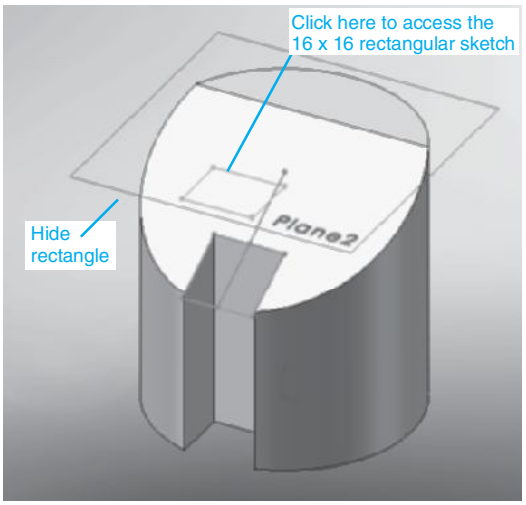


Figure 3-33









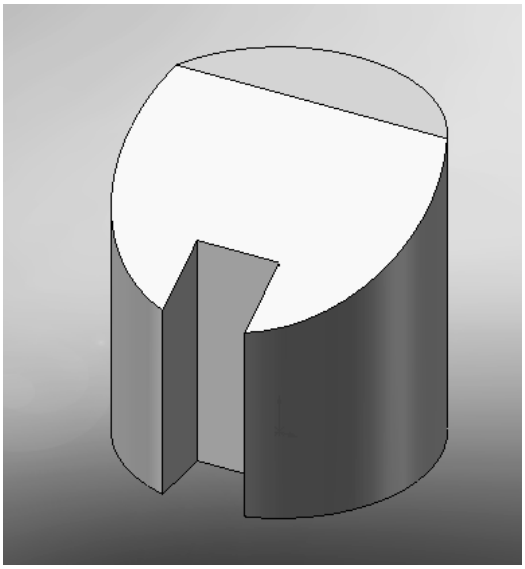


Figure 3-33 ***(continued)***

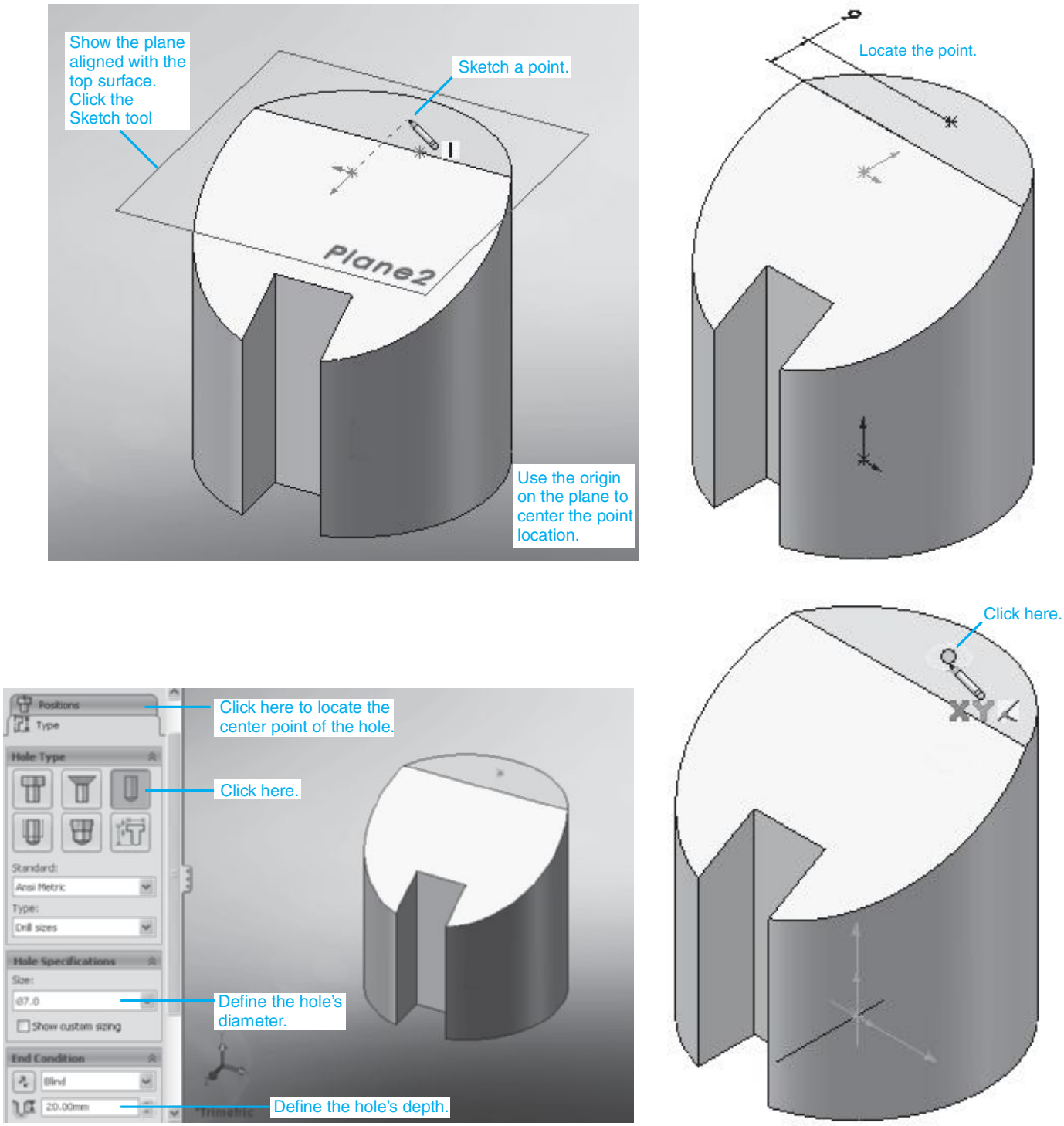


Figure 3-34

4. Click the **Hole Wizard** tool on the **Features** toolbar.
5. Click the **Hole** option and define the hole's diameter and depth.
6. Click the **Positions** tab in the **Hole Specification Properties Manager**.
7. Click the point.
8. Click the check mark.
9. Change the drawing orientation and verify that the hole has a conical-shaped bottom.

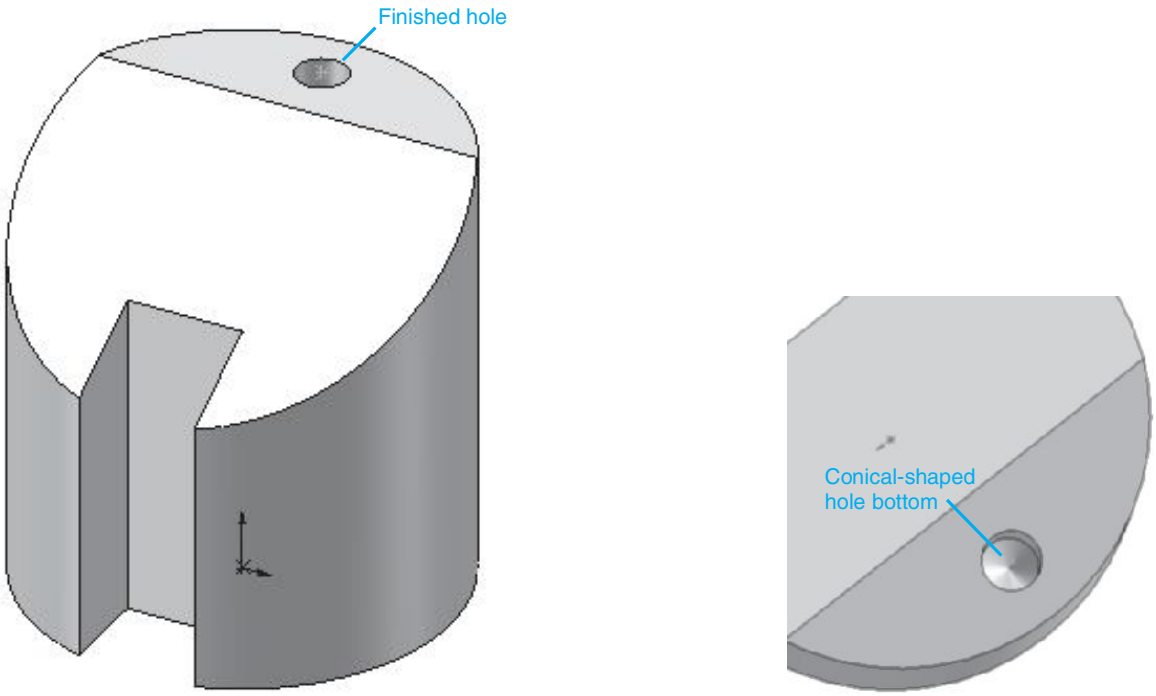


Figure 3-34 ***(continued)***

3-21 SAMPLE PROBLEM SP3-2

Figure 3-35 shows a dimensioned object. In this example we will start with the middle section of the object. See Figure 3-36.

1. Sketch a profile using the right plane based on the given dimensions.
2. Use the **Extruded Boss/Base** tool to add thickness to the profile.
3. Right-click the right surface of the object and select the **Sketch** option.
4. Use the **Rectangle** tool and draw a rectangle based on the given dimensions. Align the corners of the rectangle with the corners of the object.
5. Use the **Extruded Boss/Base** tool and extrude the rectangle **20** to the right.
6. Reorient the object and draw a rectangle on the left surface of the object.
7. Use the **Extruded Boss/Base** tool and extrude the rectangle **20** to the left.
8. Create a sketch plane on the right side of the object and draw a rectangle based on the given dimensions as shown.
9. Use the **Extruded Cut** tool on the **Features** toolbar and cut out the rectangle over the length of the object.

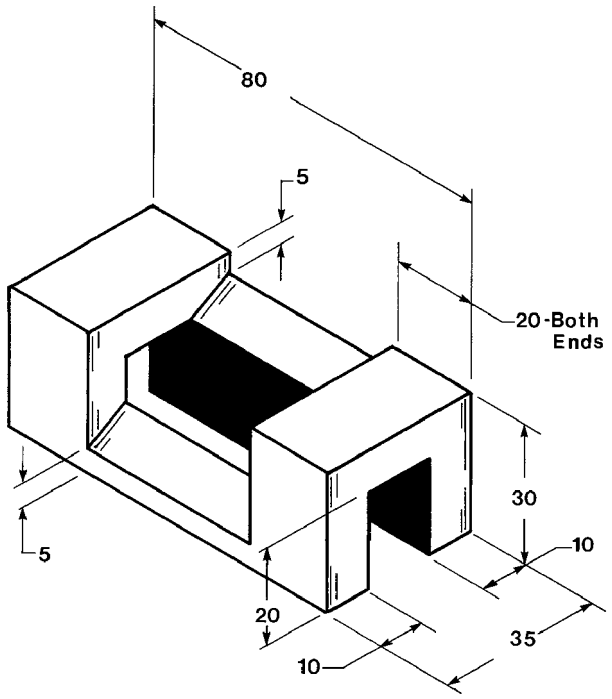
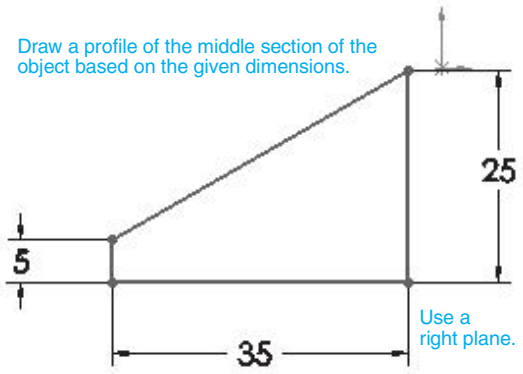
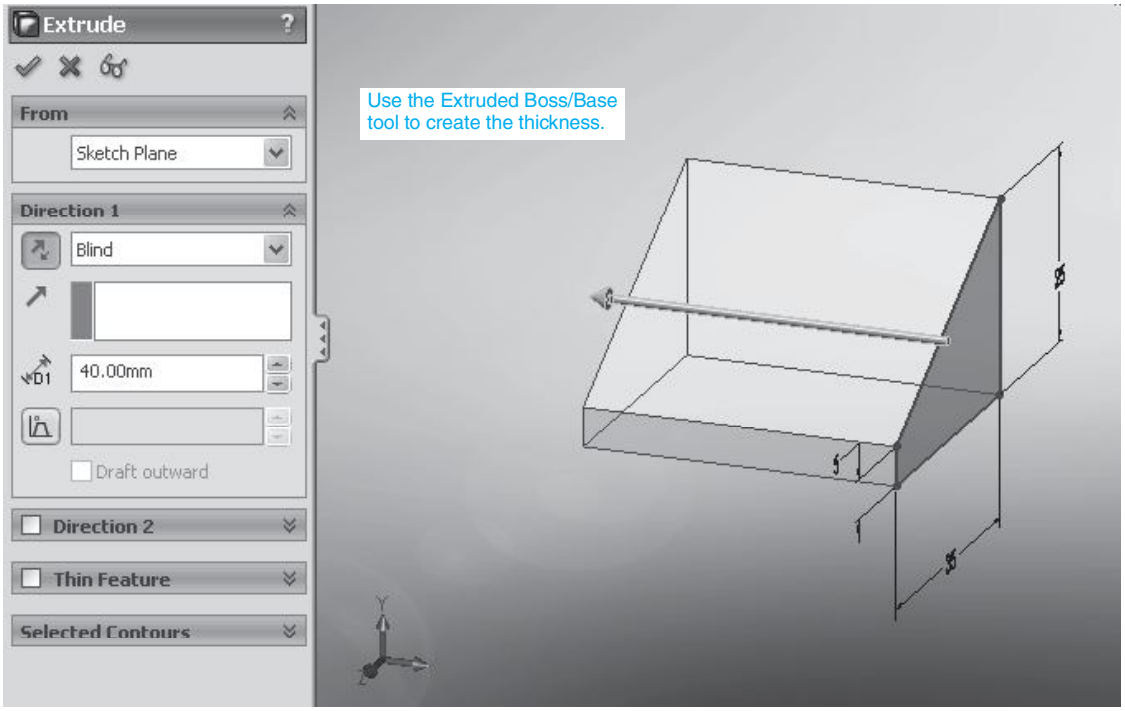


Figure 3-35


Draw a profile of the middle section of the object based on the given dimensions.
25
5
35
Use a right plane.


Extrude
From
Sketch Plane
Direction 1
Blind
40.00mm
Draft outward
Direction 2
Thin Feature
Selected Contours
Use the Extruded Boss/Base tool to create the thickness.

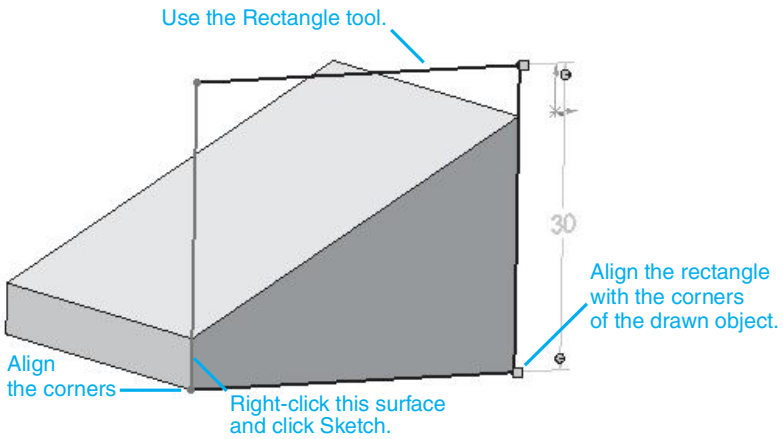

Use the Rectangle tool.
30
Align the rectangle with the corners of the drawn object.
Align the corners
Right-click this surface and click Sketch.

Figure 3-36

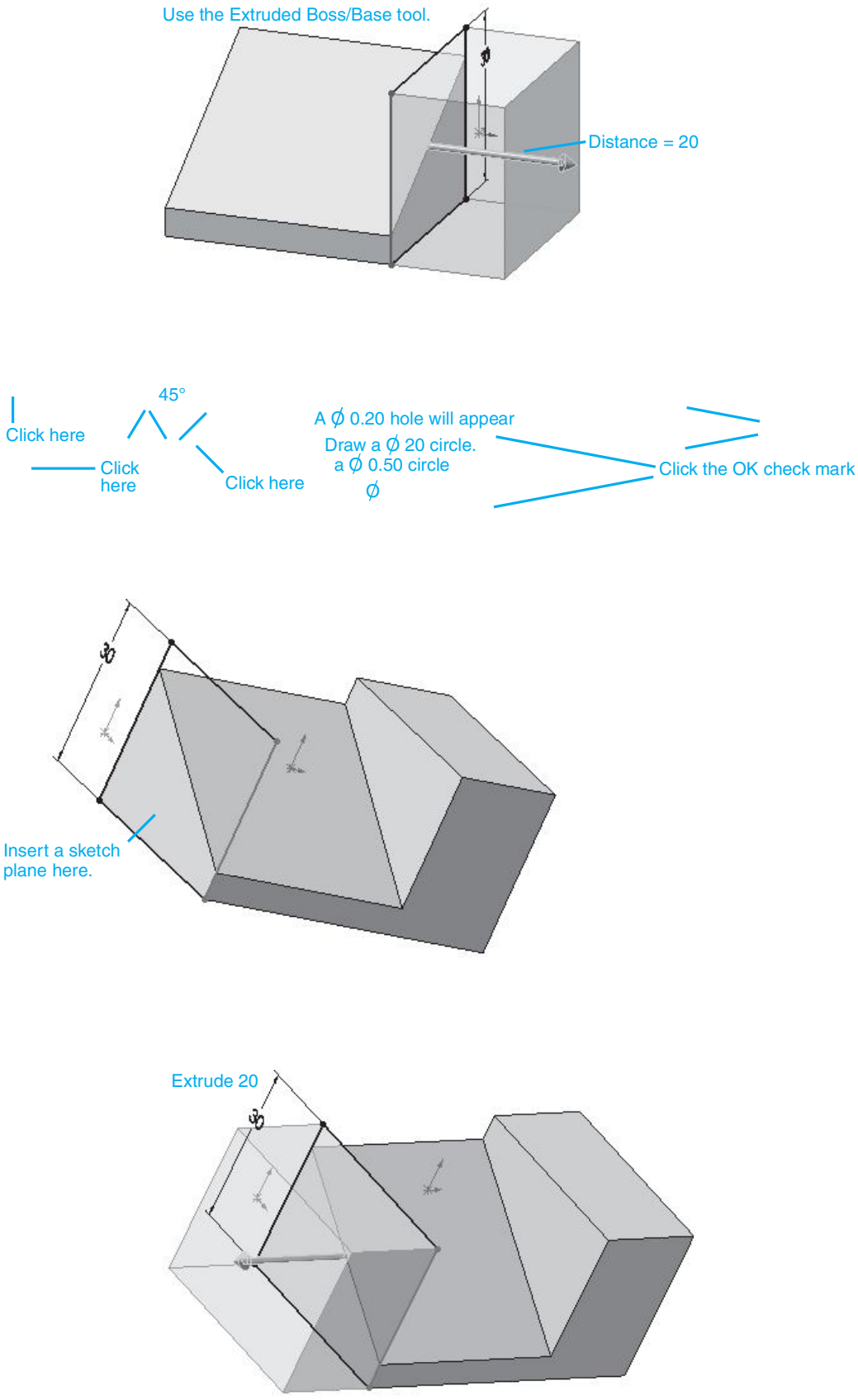
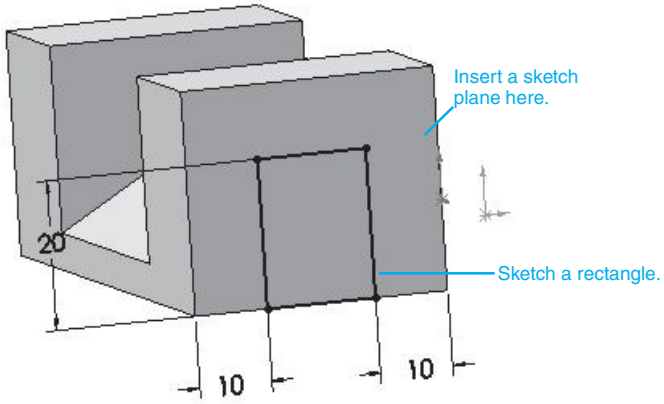
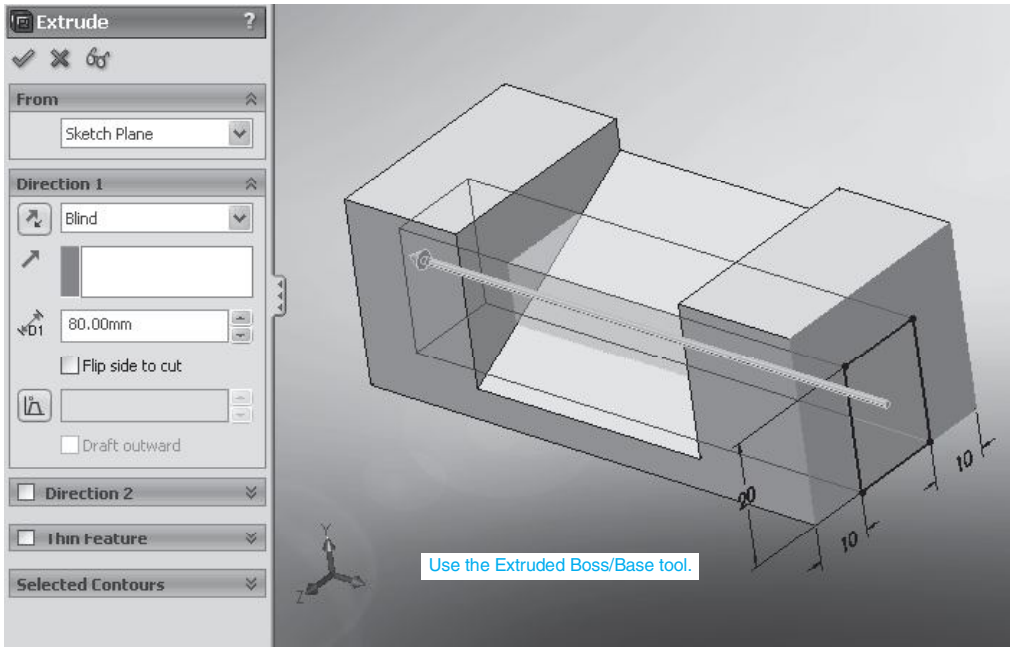


Figure 3-36 ***(continued)***





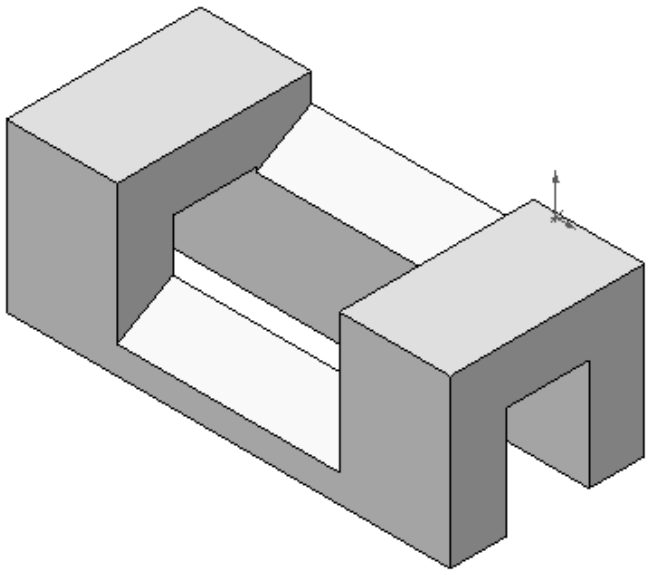


Figure 3-36 ***(continued)***

3-23 PROJECTS

Project 3-1:

Redraw the following objects as solid models based on the given dimensions. Make all models from mild steel.

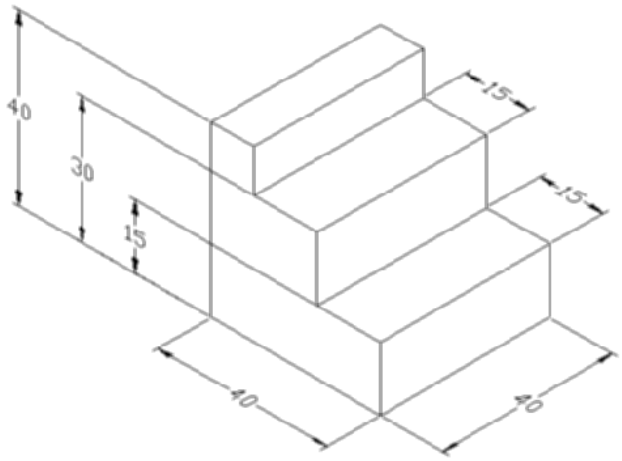


Figure P3-1 MILLIMETERS

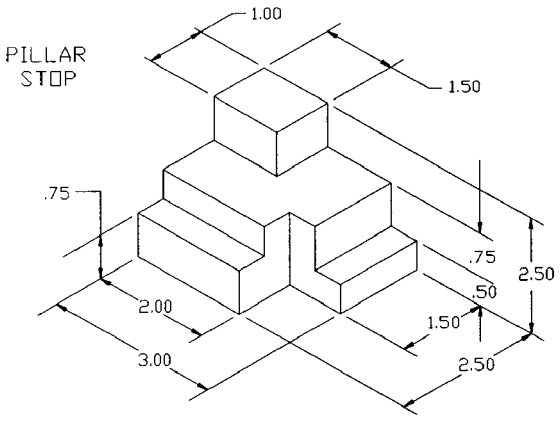


Figure P3-2 INCHES

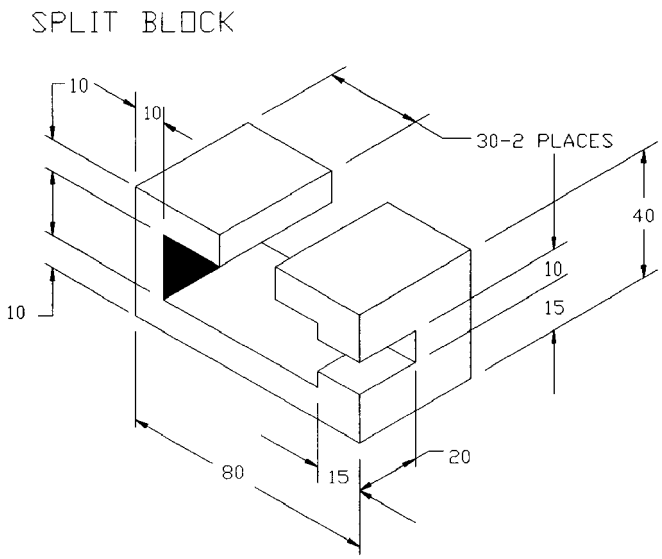


Figure P3-3 MILLIMETERS

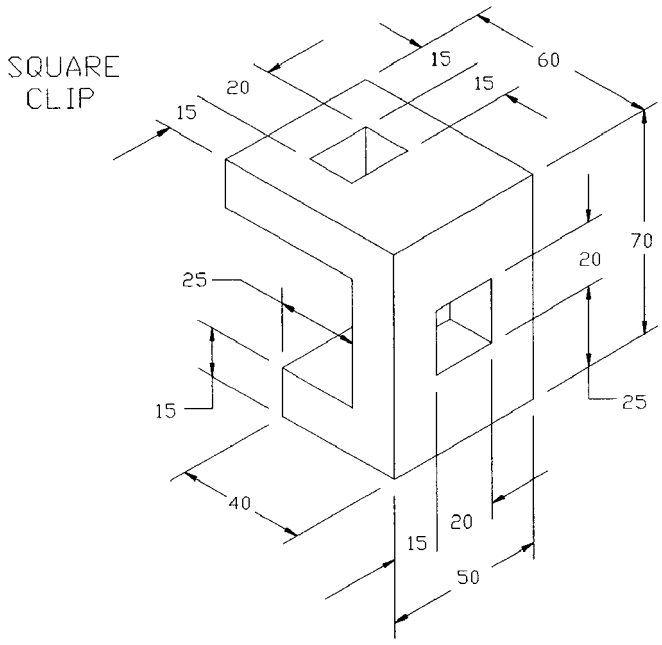


Figure P3-4 MILLIMETERS

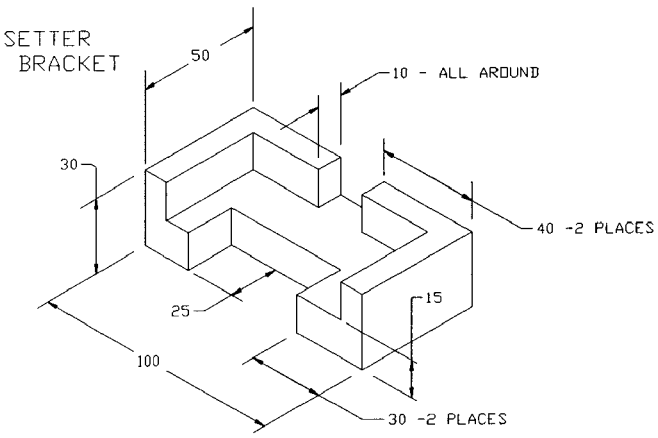
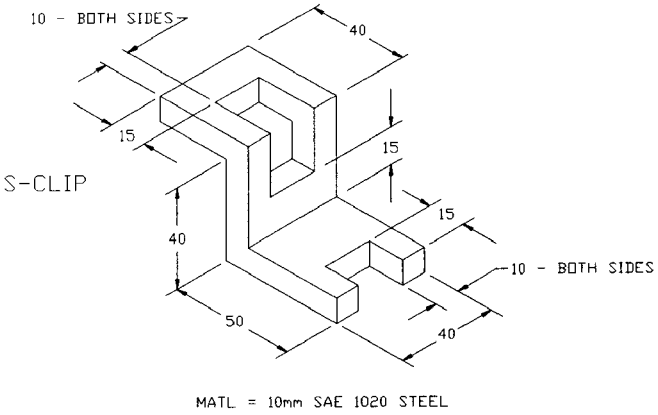


Figure P3-5 MILLIMETERS



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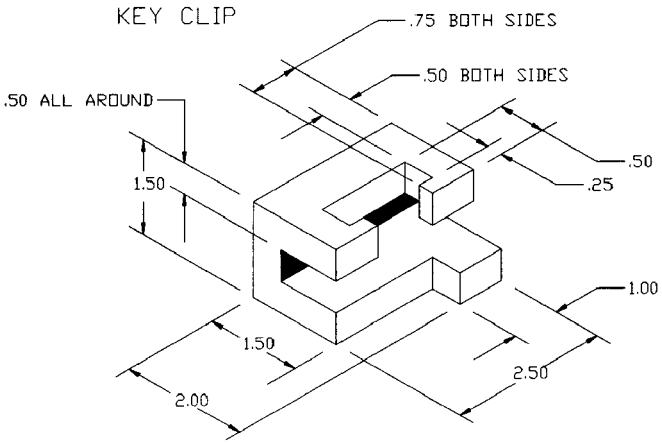

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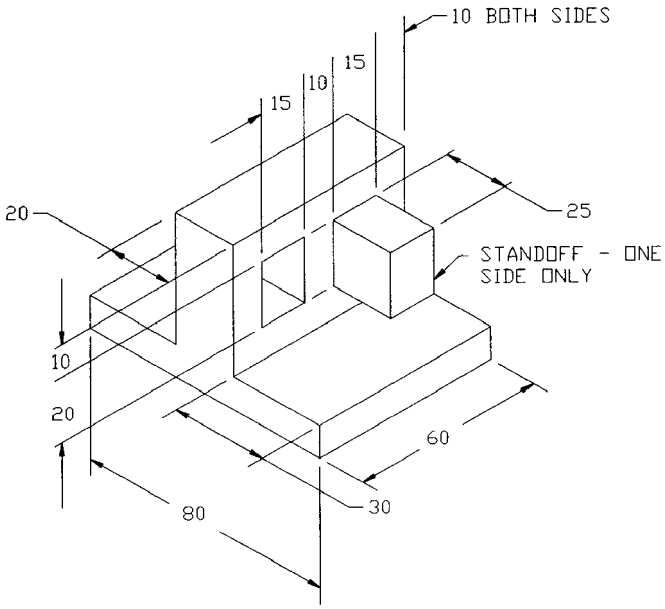

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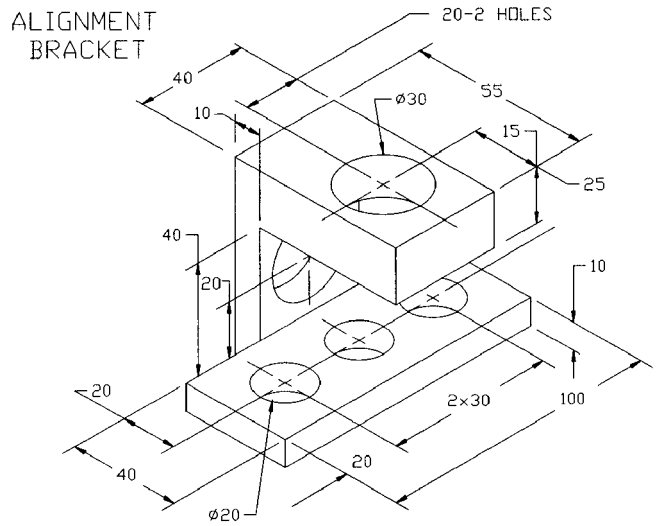

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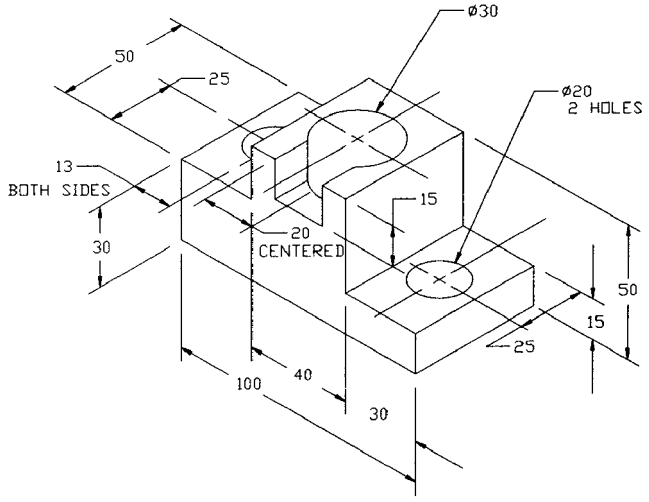

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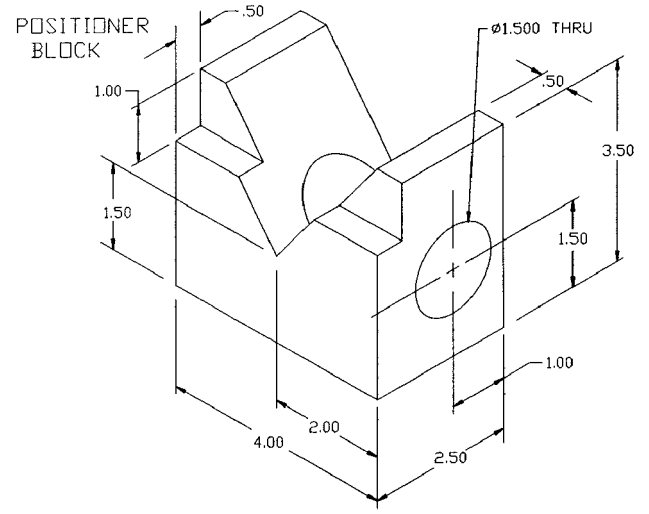

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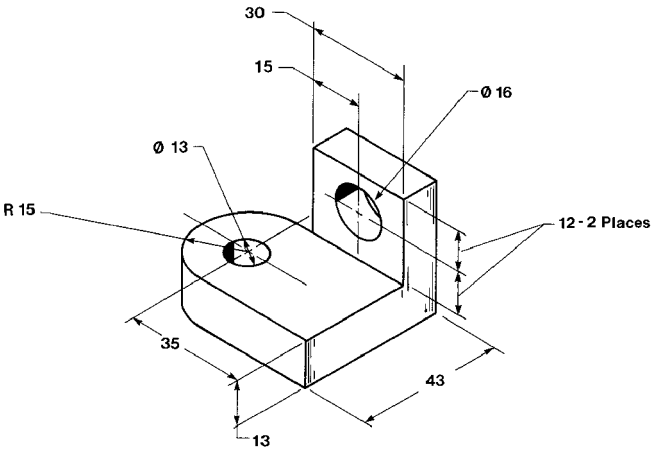

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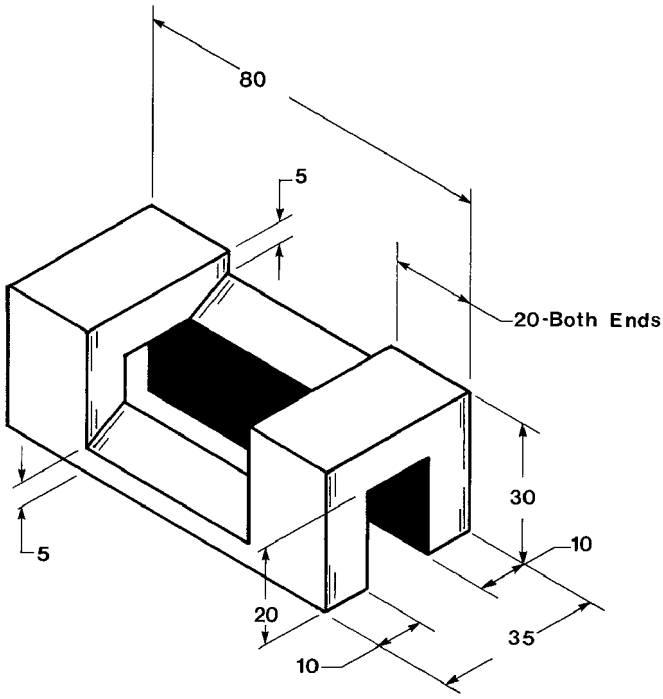

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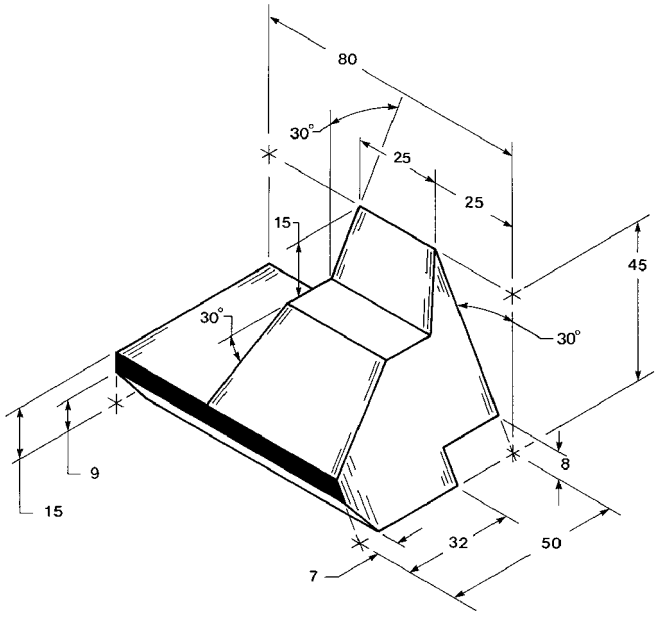

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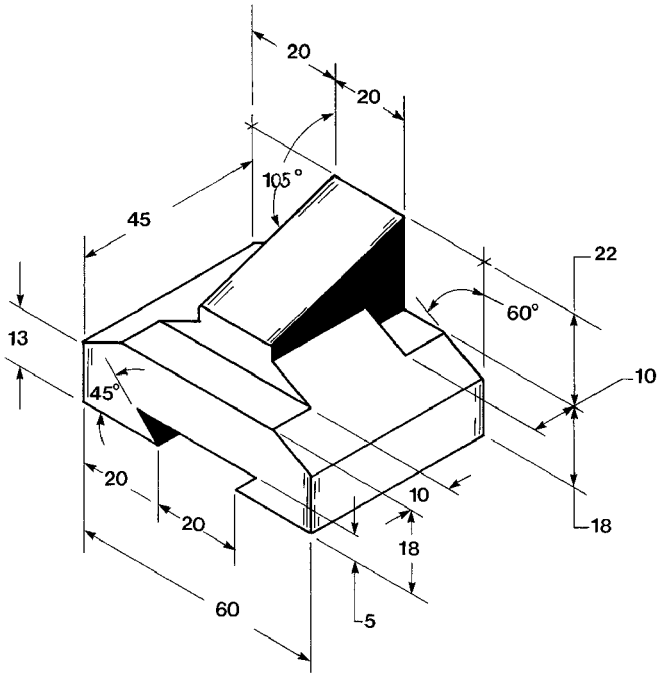

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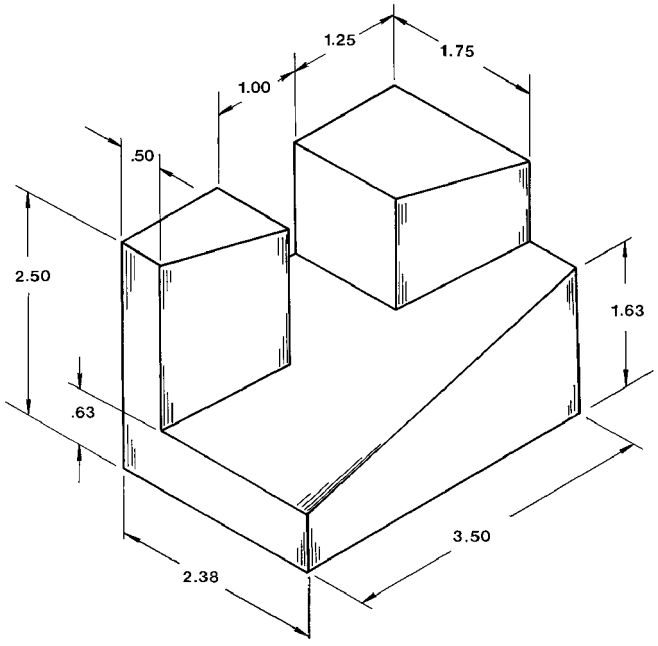

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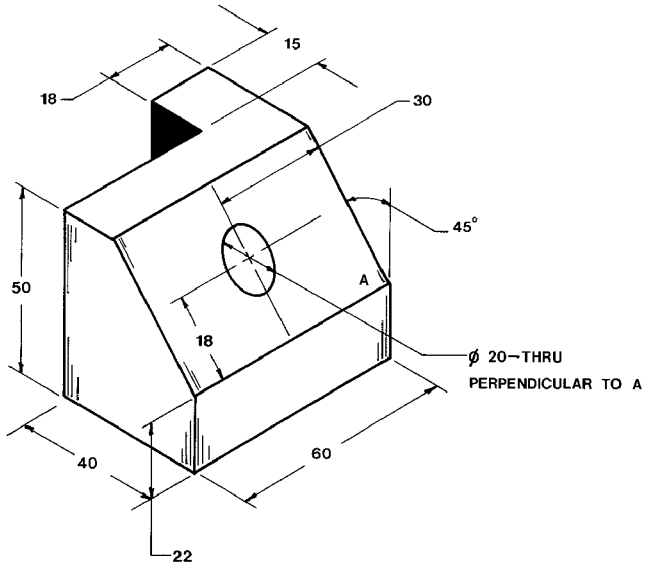

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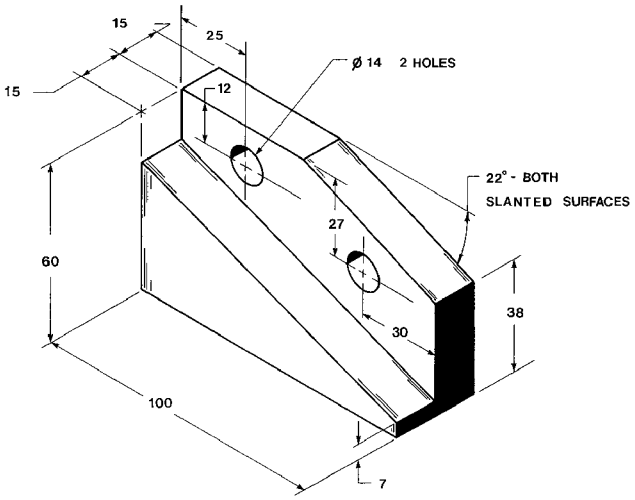

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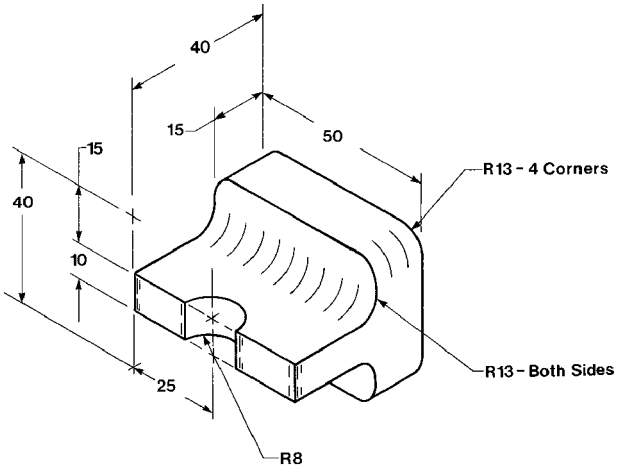

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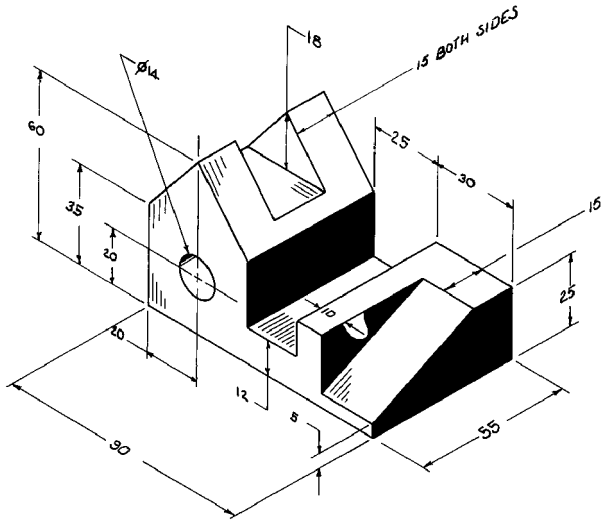

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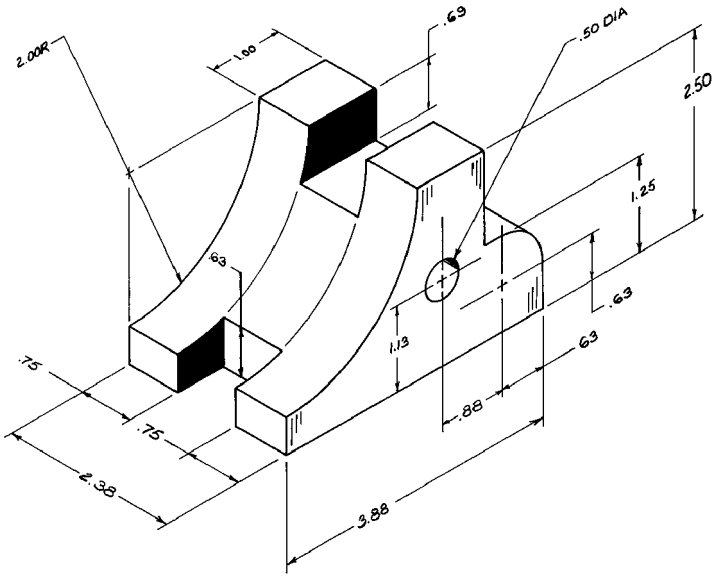

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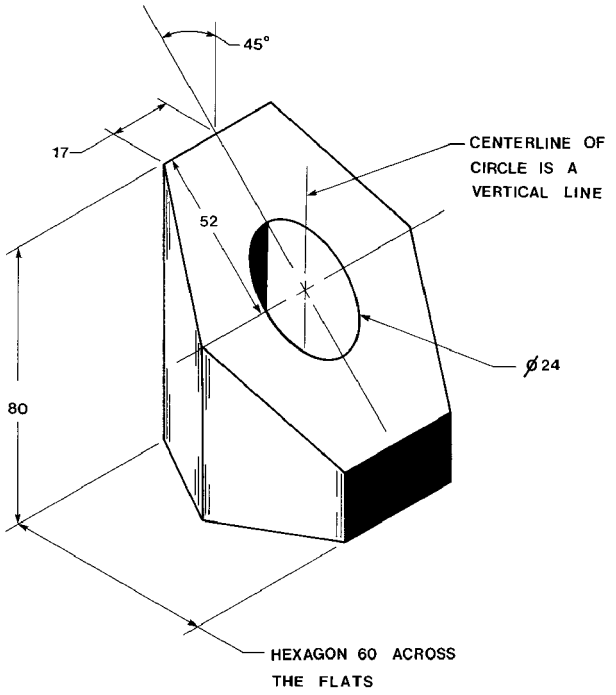
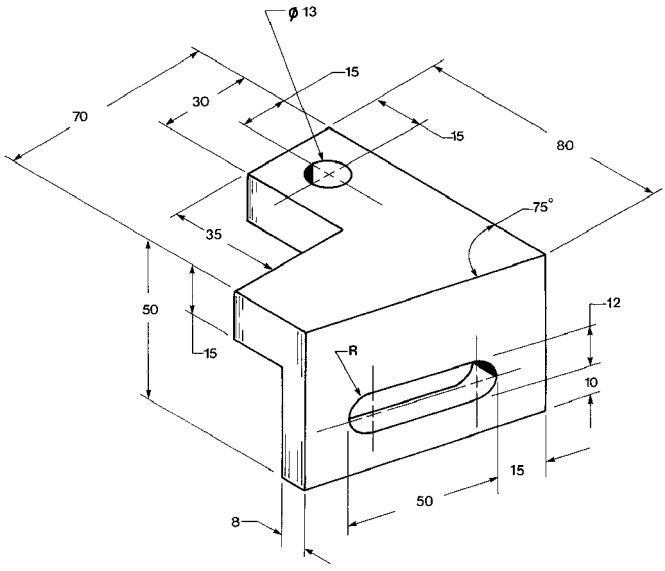


Figure P3-21 MILLIMETERS



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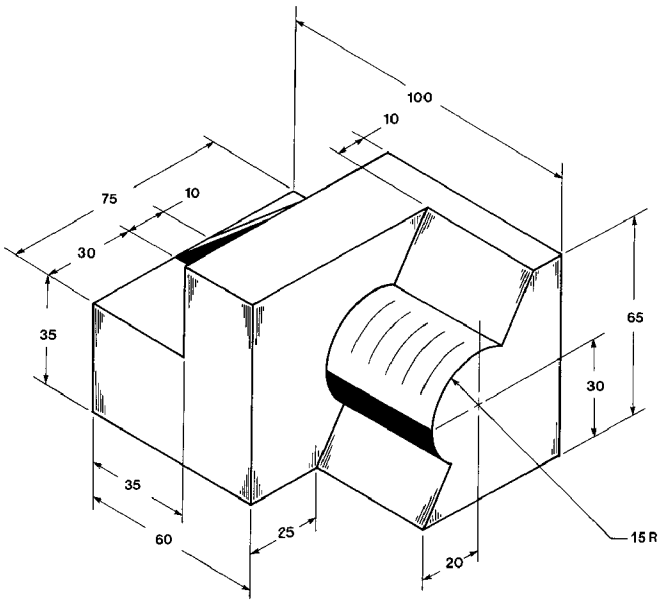

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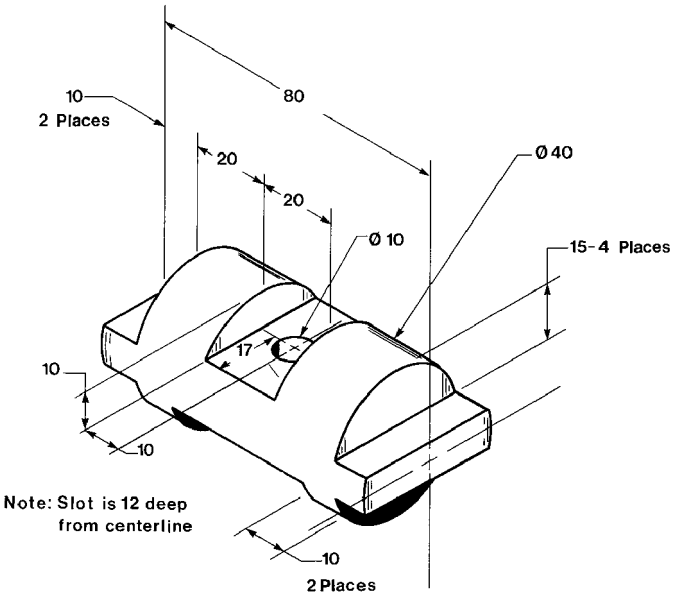

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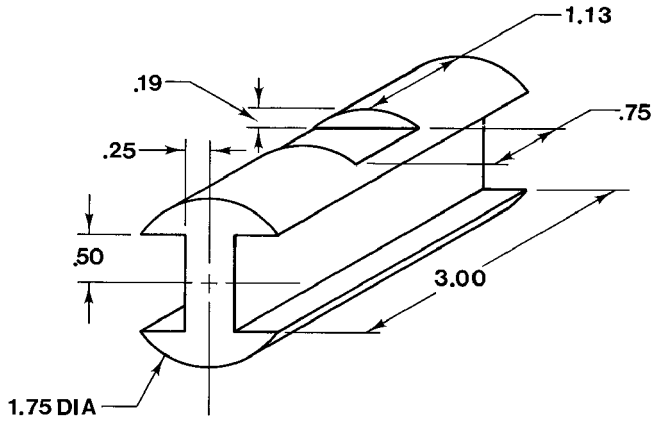
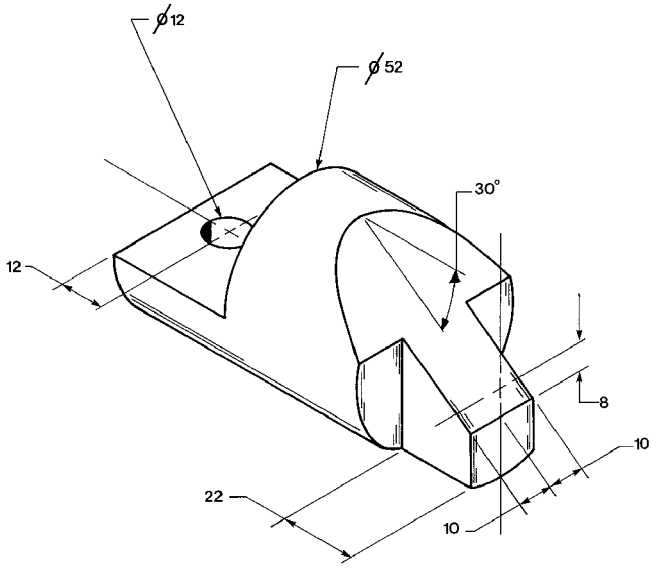


Figure P3-25 INCHES (SCALE: 4=1)



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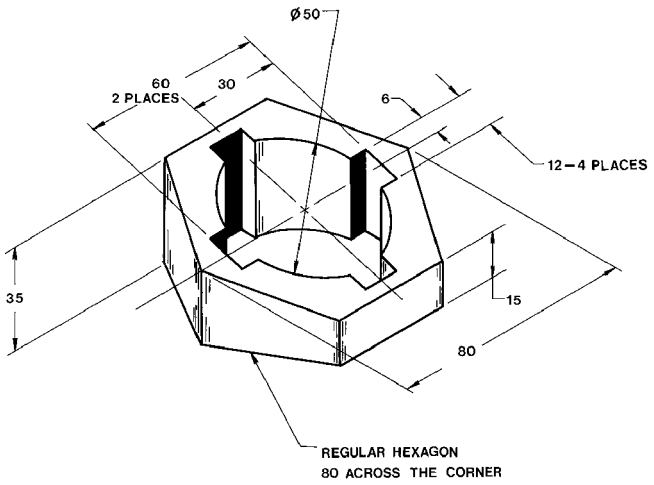

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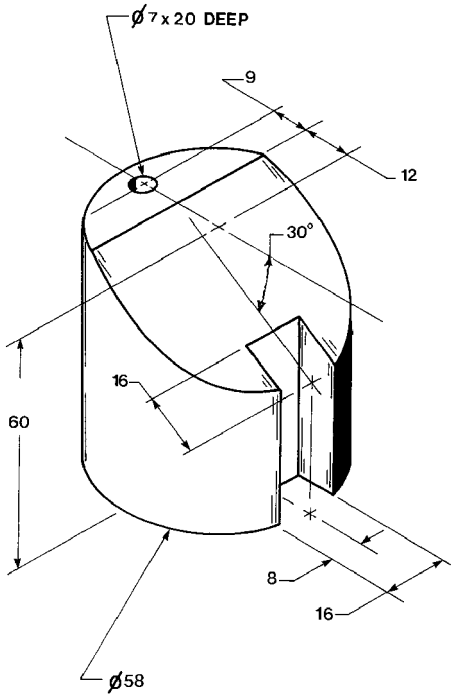


Figure P3-30 MILLIMETERS (SCALE: 2=1)

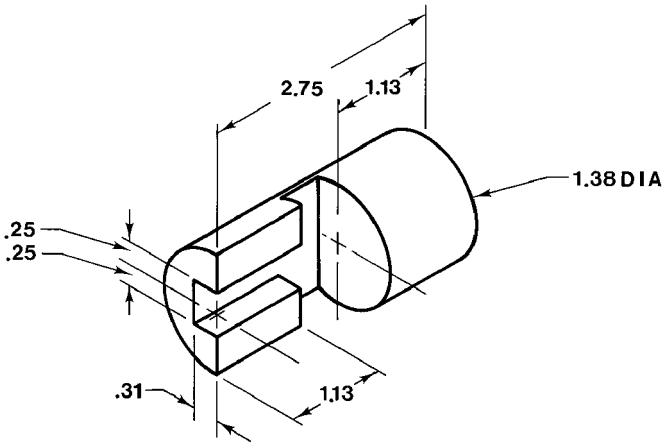


Figure P3-29 INCHES (SCALE: 4=1)

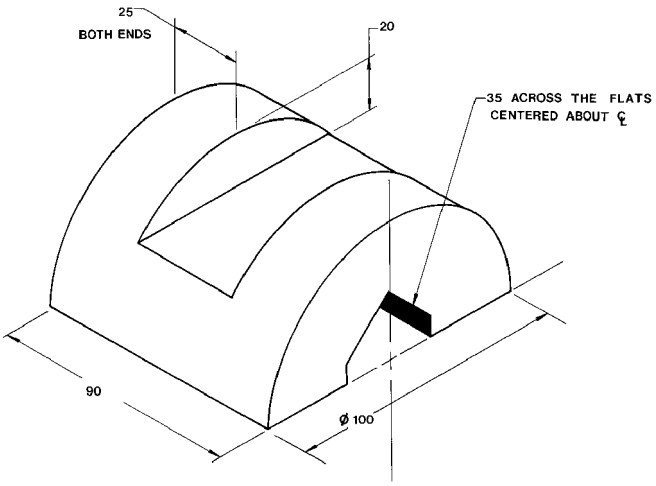
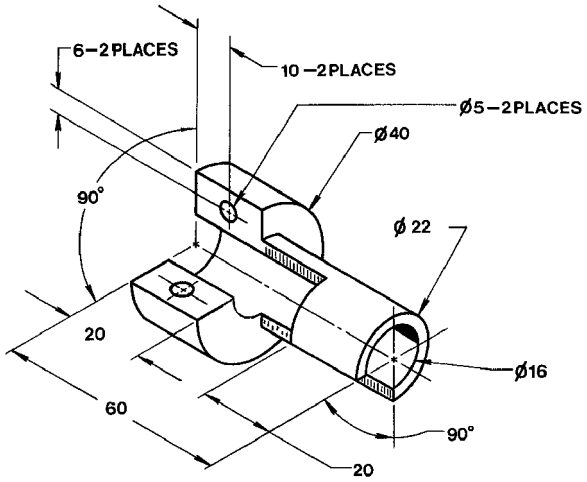


Figure P3-31 MILLIMETERS



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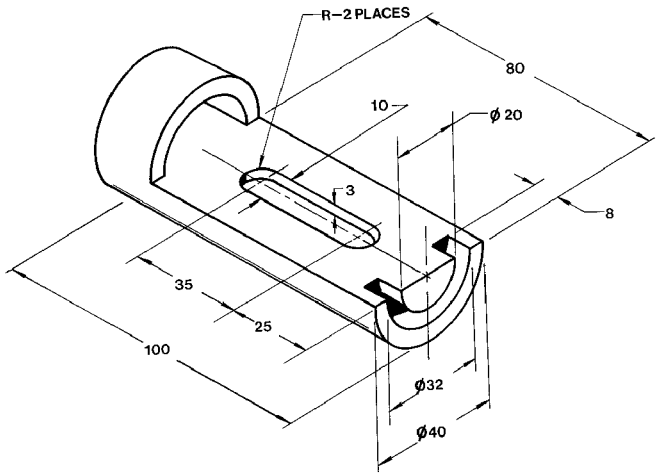

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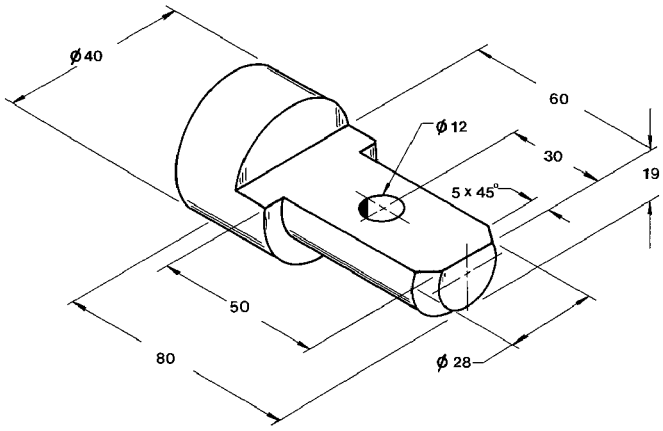

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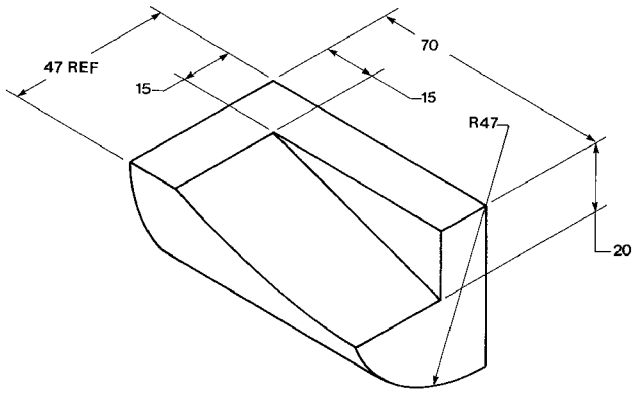
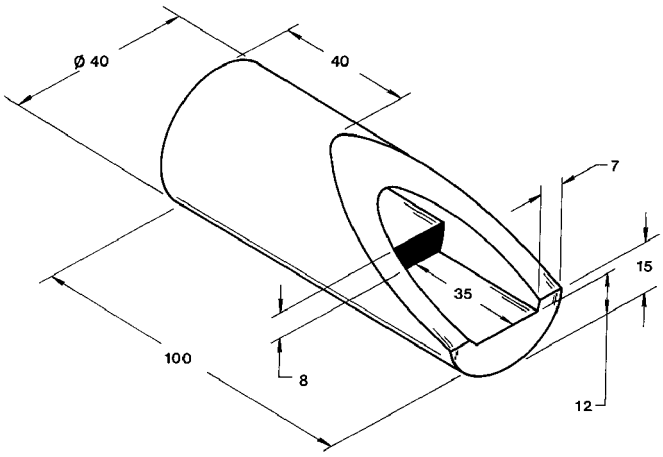


Figure P3-35 MILLIMETERS



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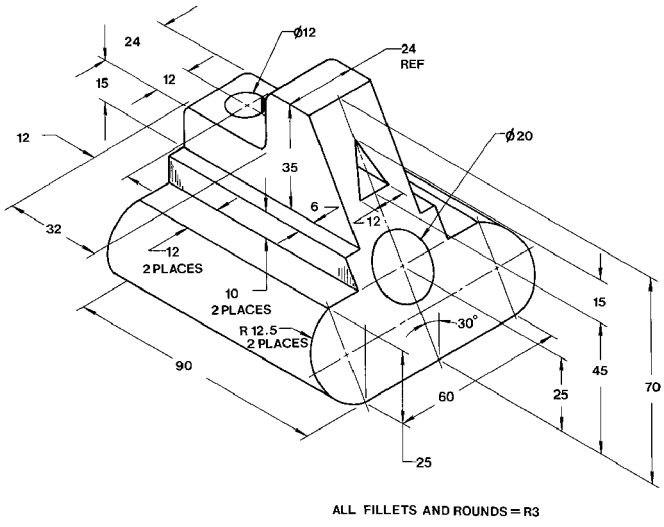

Figure P3-37 MILLIMETERS

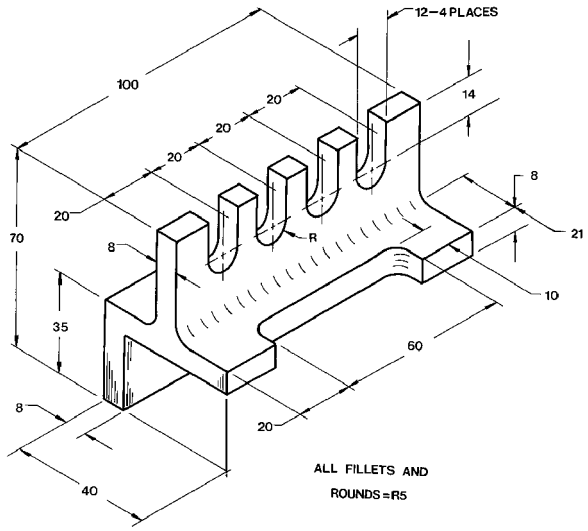

Figure P3-38 MILLIMETERS

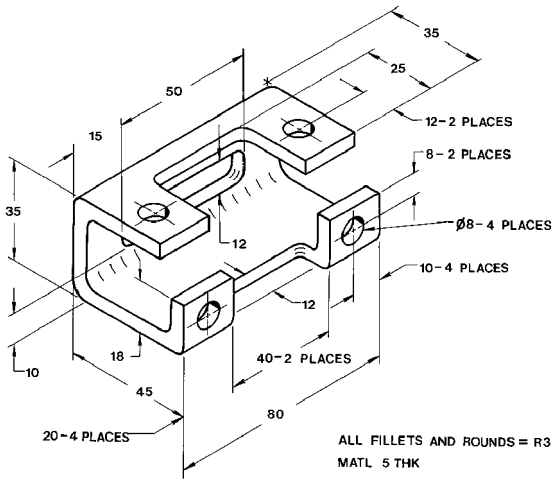

Figure P3-39 MILLIMETERS (CONSIDER A SHELL)

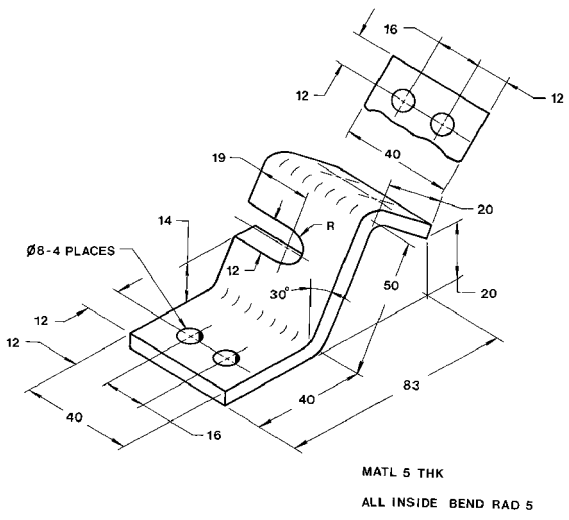

Figure P3-40 MILLIMETERS

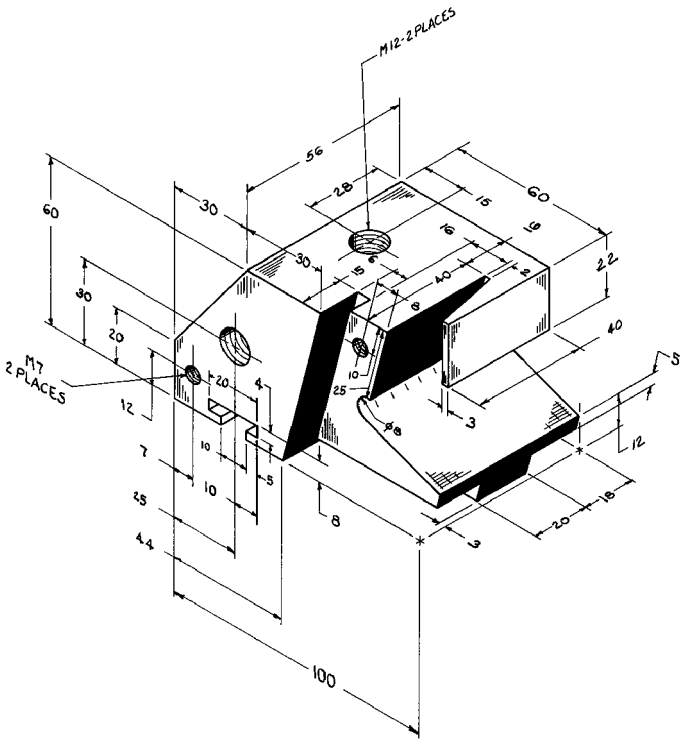

Figure P3-41 MILLIMETERS

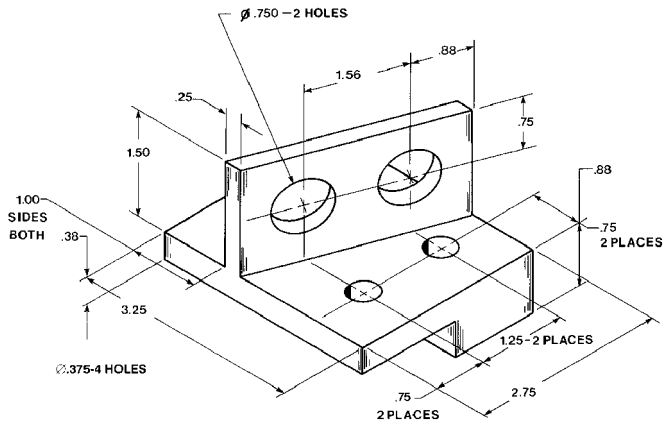

Figure P3-42 INCHES

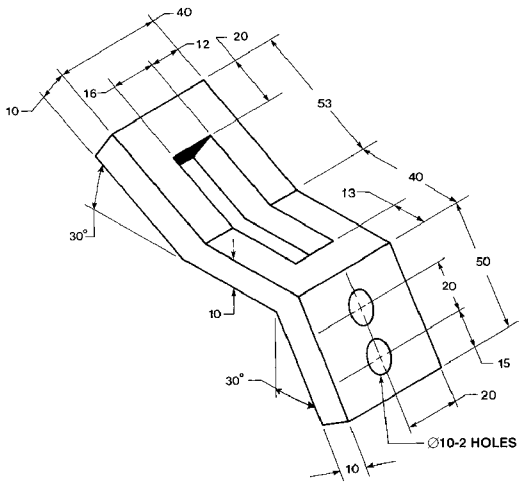

Figure P3-43 MILLIMETERS

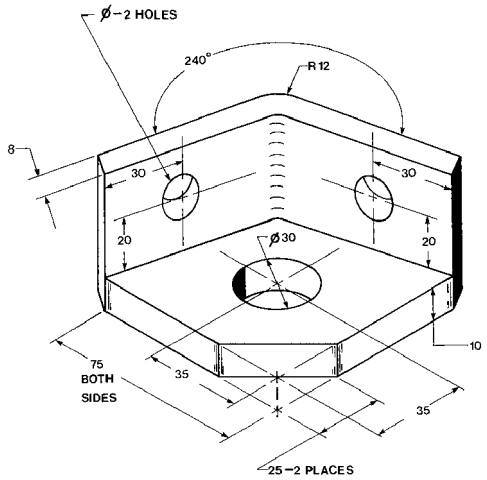

Figure P3-46 MILLIMETERS

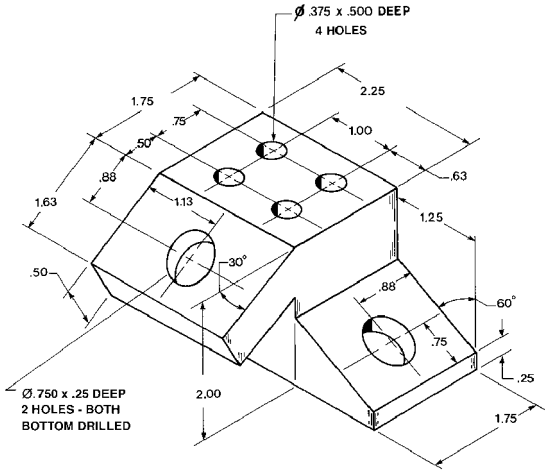

Figure P3-44 INCHES

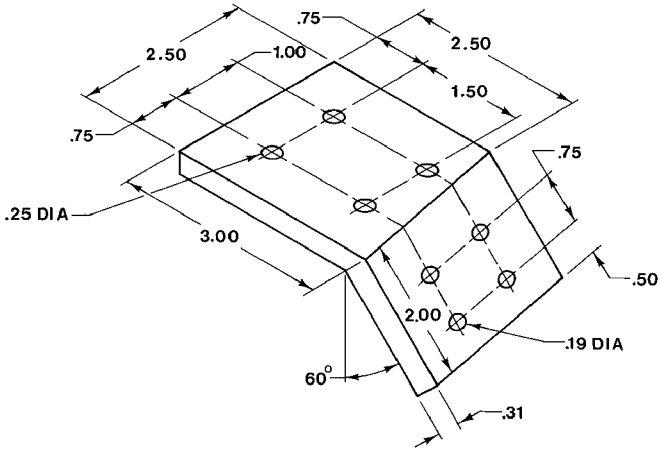

Figure P3-47 INCHES

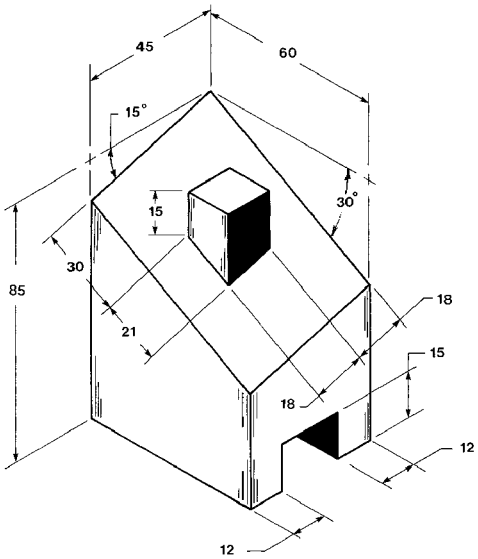

Figure P3-45 MILLIMETERS

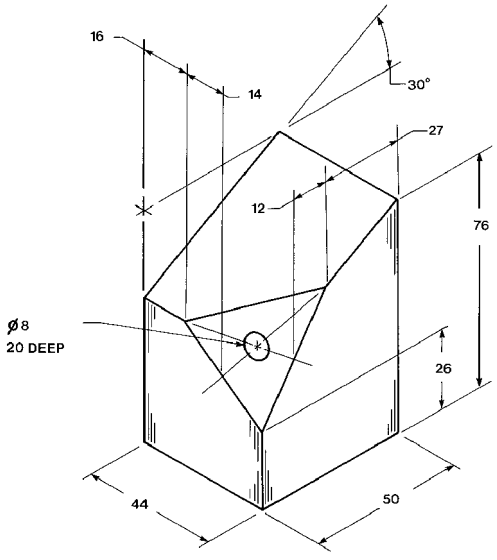

Figure P3-48 MILLIMETERS

CHAPTER 4

Orthographic Views

Objectives

- Introduce orthographic views.
- Learn ANSI standards and conventions.
- Learn how to draw sectional and auxiliary views.

4-1 INTRODUCTION

Orthographic views may be created directly from 3D SolidWorks, models. ***Orthographic views*** are two-dimensional views used to define a three-dimensional model. Unless the model is of uniform thickness, more than one orthographic view is necessary to define the model's shape. Standard practice calls for three orthographic views: a front, top, and right side view, although more or fewer views may be used as needed.

Modern machines can work directly from the information generated when a solid 3D model is created, so the need for orthographic views—blueprints—is not as critical as it once was; however, there are still many drawings in existence that are used for production and reference. The ability to create and read orthographic views remains an important engineering skill.

This chapter presents orthographic views using third-angle projection in accordance with ANSI standards. ISO first-angle projections are also presented.

4-2 FUNDAMENTALS OF ORTHOGRAPHIC VIEWS

Figure 4-1 shows an object with its front, top, and right-side orthographic views projected from the object. The views are two-dimensional, so they show no depth. Note that in the projected right plane there are three rectangles. There is no way to determine which of the three is closest and which is farthest away if only the right-side view is considered. All views must be studied to analyze the shape of the object.

Figure 4-2 shows three orthographic views of a book. After the views are projected they are positioned as shown. The positioning of views relative to one another is critical. The views must be aligned and positioned as shown.

Normal Surfaces

Normal surfaces are surfaces that are at 90° to each other. Figures 4-3, 4-4, and 4-5 show objects that include only normal surfaces and their orthographic views.

Hidden Lines

Hidden lines are used to show surfaces that are not directly visible. All surfaces must be shown in all views. If an edge or surface is blocked from view by another feature, it is drawn using a hidden line. Figures 4-6 and 4-7 show objects that require hidden lines in their orthographic views.