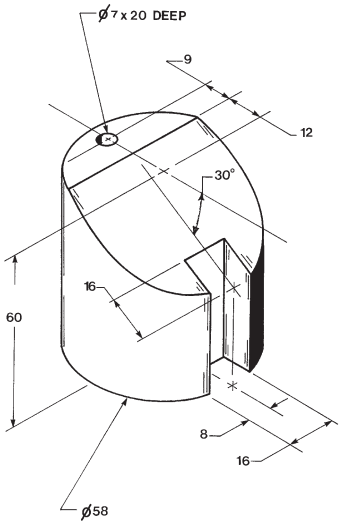
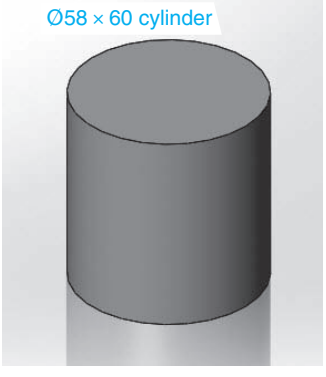
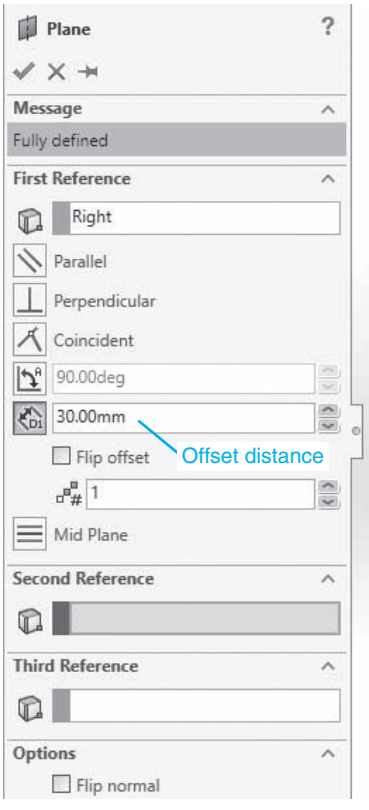
Figure 3-48



To Draw a Cylinder

1. Start a new **Part** document.
2. Define the units as millimeters (**MMGS**), the Overall drafting standards should be **ANSI**, access the top plane, and make the top plane a sketch plane.
3. Draw a **Ø58** circle and extrude it to **60** centered on the origin.





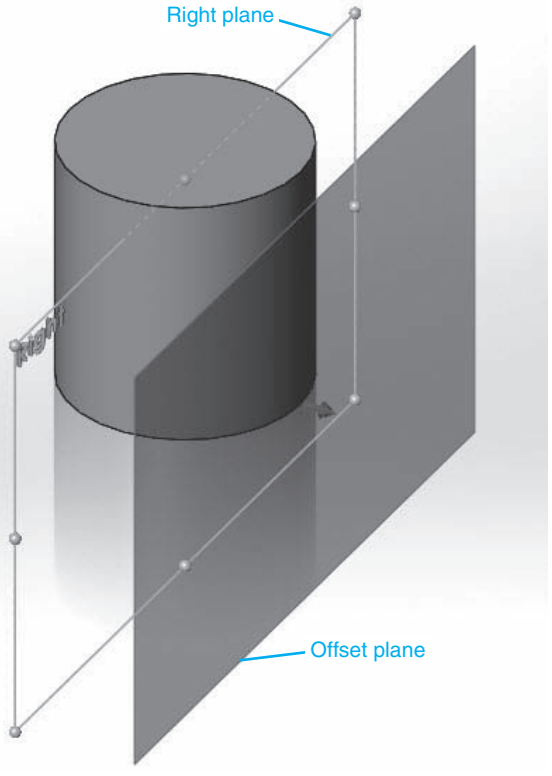
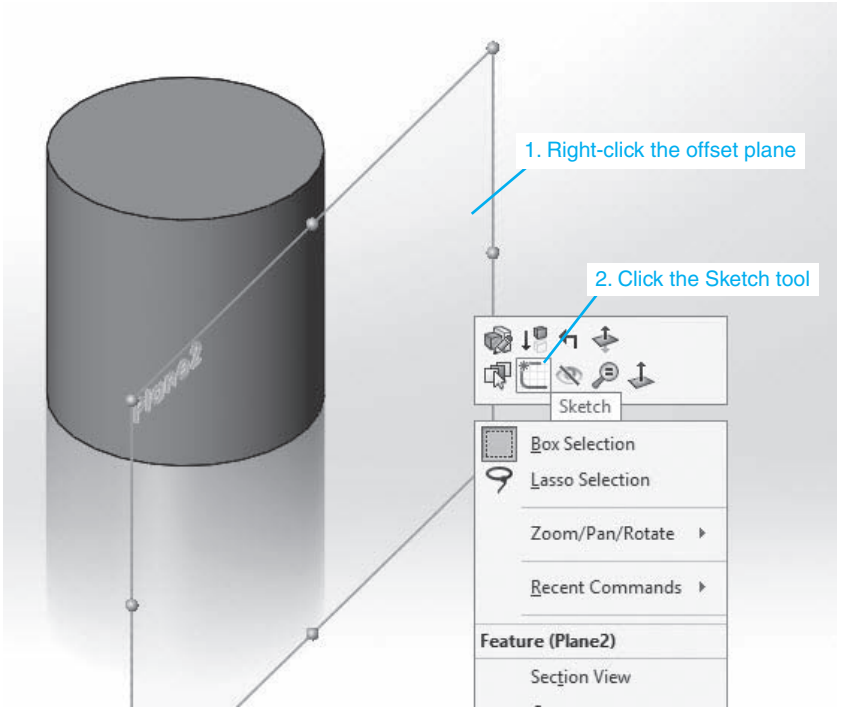
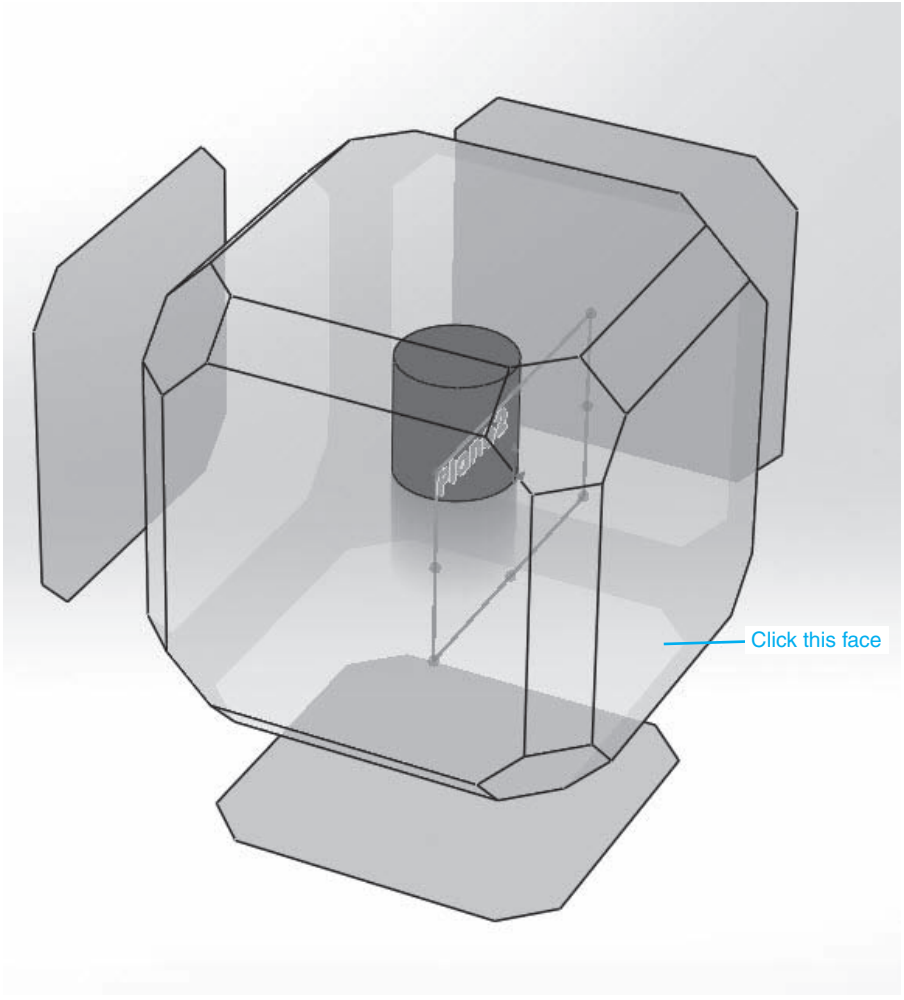
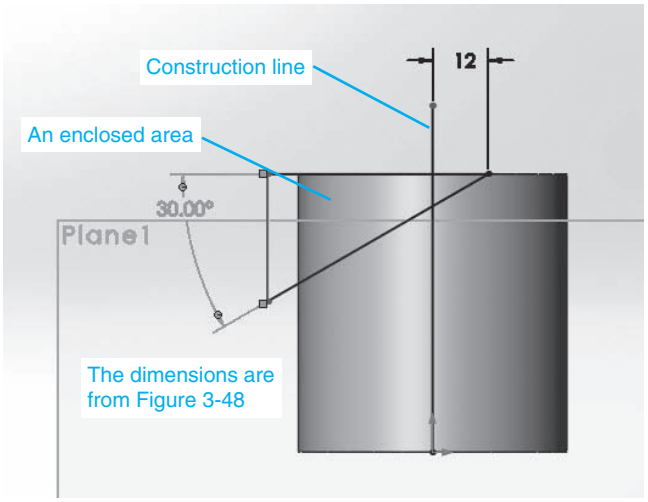


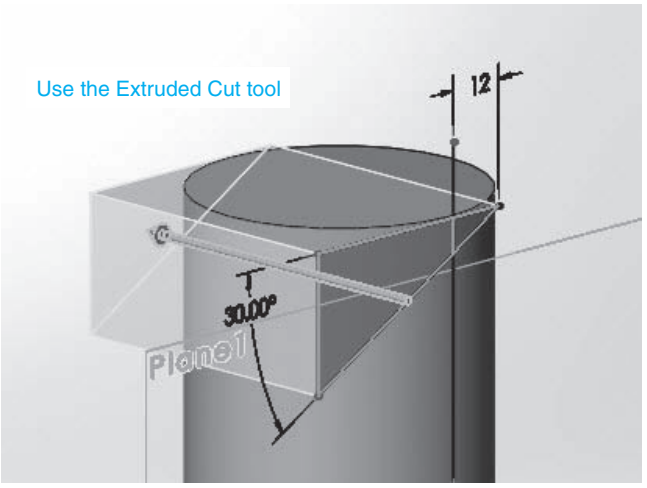
Figure 3-49

Figure 3-49
(*Continued*)









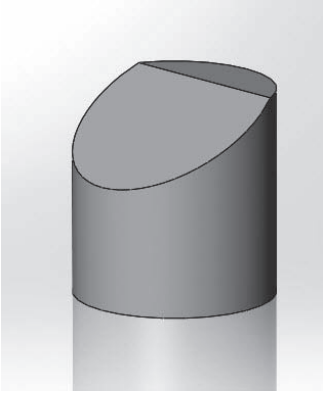


Figure 3-49
(*Continued*)

To Create a Slanted Surface on the Cylinder

1. Click the **Right plane** option, click the **Reference Geometry** tool under the **Features** tab, and click the **Plane** option.
2. Define the offset plane distance in the **Plane PropertyManager** as **30**, and click the green **OK** check mark.
3. Right-click the offset plane and select the **Sketch** option.
4. Change the drawing orientation to the **right plane**.
5. Use the **Line** tool and draw an enclosed triangular shape.

NOTE

The dimensions for the triangle came from Figure 3-48. The triangle must be an enclosed area. No gaps are permitted. A vertical construction line was added to help in the location and creation of the triangular area.

6. Use the **Smart Dimension** tool to define the size and location of the triangle.
7. Change the drawing orientation to **Dimetric** and click the **Extruded Cut** tool in the **Features** tab.
8. Set the length of the cut to **60.00mm** and click the **OK** check mark.
9. Right-click the offset plane and click the **Hide** option.

To Add the Vertical Slot

See Figure 3-50.

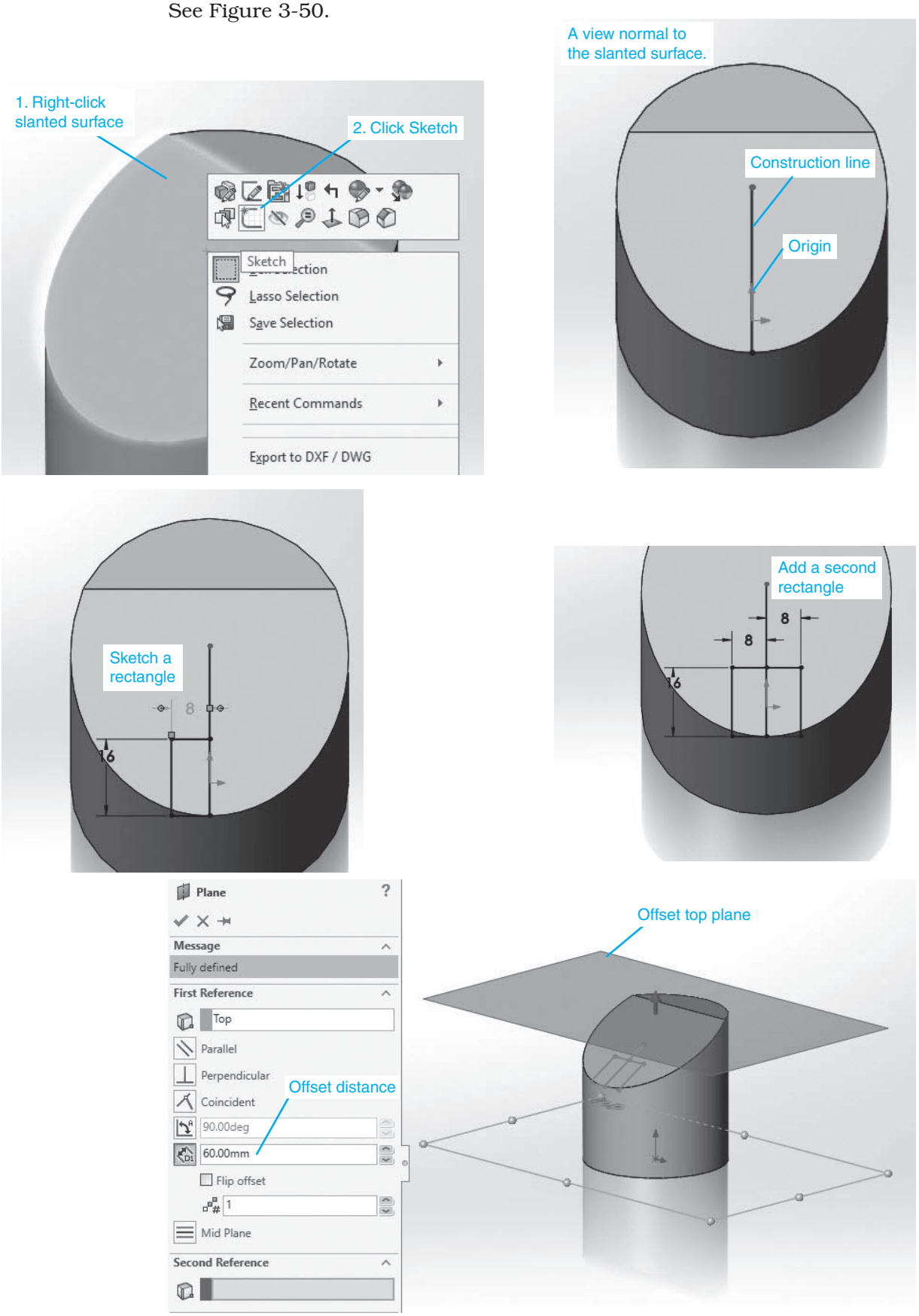
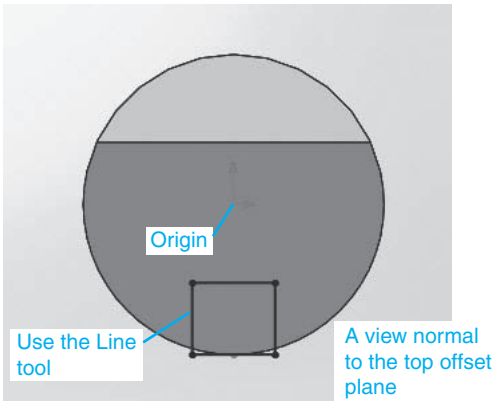
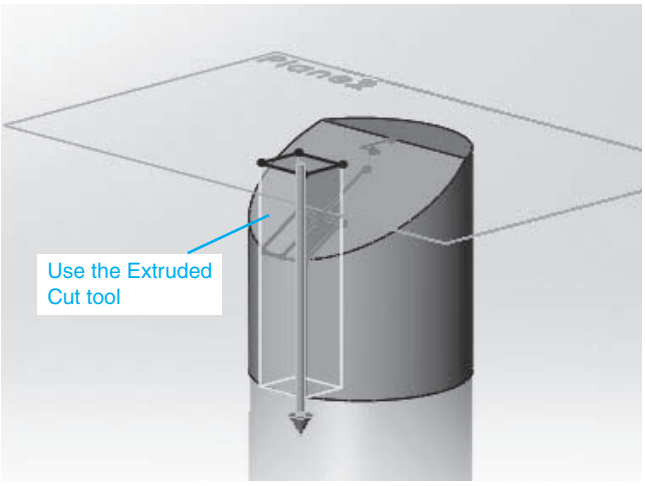


Figure 3-50





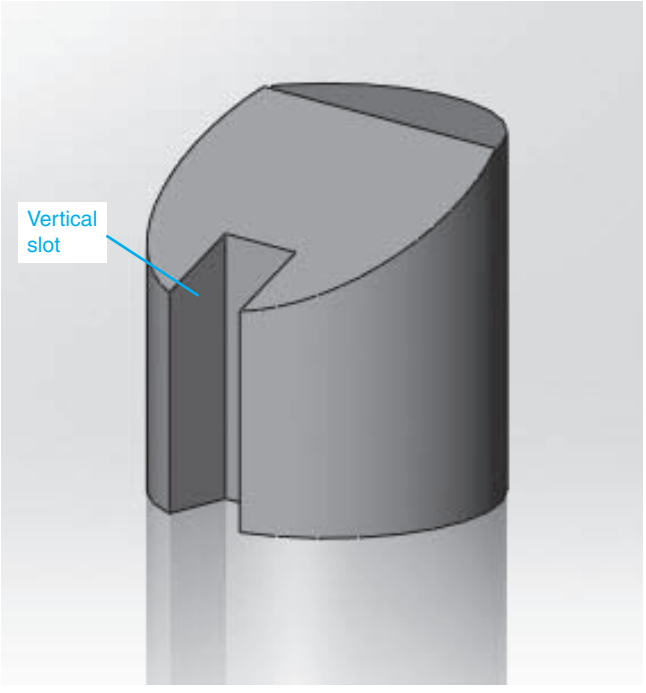


Figure 3-50
(*Continued*)

1. Right-click the slanted surface and click the **Sketch** tool.
2. Click the **View Orientation** tool and click the **Normal to View** tool, or click **<Ctrl-8>**.
3. Use the **Line** tool and sketch a vertical construction 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.

   The lower edge of the rectangle must be aligned with the edge of the cylinder or extend beyond the edge to ensure that the cutout removes all material.
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** option, then use the **Plane** option in the **Reference Geometry** tool on the **Features** tab 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. Right-click the 60 offset plane, select the **Sketch** option, and change the drawing orientation to the top view.
10. Use the **Line** tool and sketch a rectangle on the offset plane over the projected view of the 16 × 16 rectangle on the slanted plane, right-click the mouse, and click the **Select** option.
11. Change the drawing orientation to a dimetric view.
12. Use the **Extruded Cut** tool on the **Features** tab to cut out the slot.
13. Hide the 60 offset plane and hide the 16 × 16 rectangle on the slanted surface.
14. Click the green **OK** check mark.

To Add the Ø8 Hole

See Figure 3-51.

1. Right-click the flat top surface of the object and create a sketch plane.
2. Change the orientation to a view normal to the flat top surface.
3. Use the **Point** tool and sketch a point on the flat portion of the top surface. Use the origin to center the point.
4. Use the **Smart Dimension** tool and locate the point according to the given dimensions, in this example 9.00mm.
5. Change to a trimetric orientation and exit the sketch.

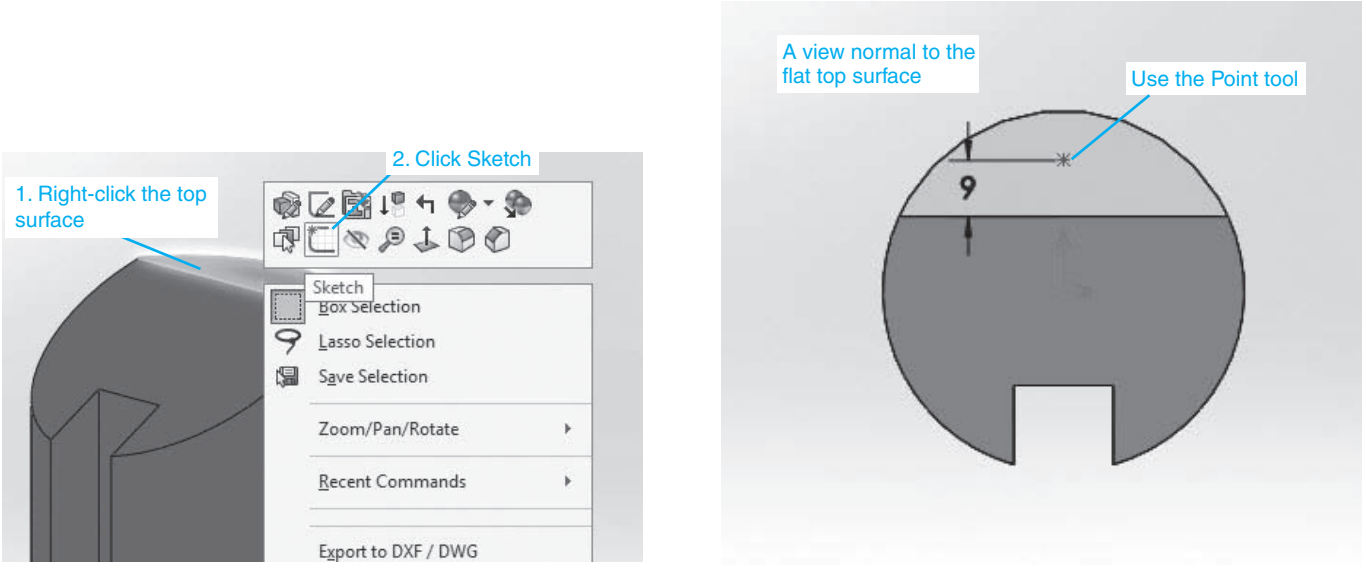
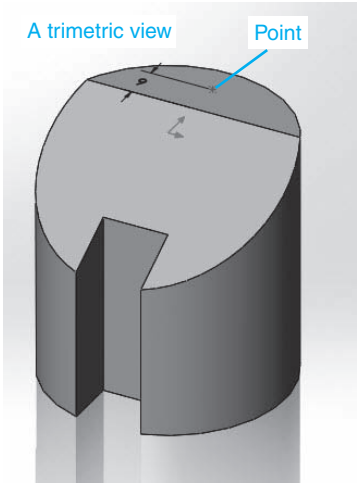
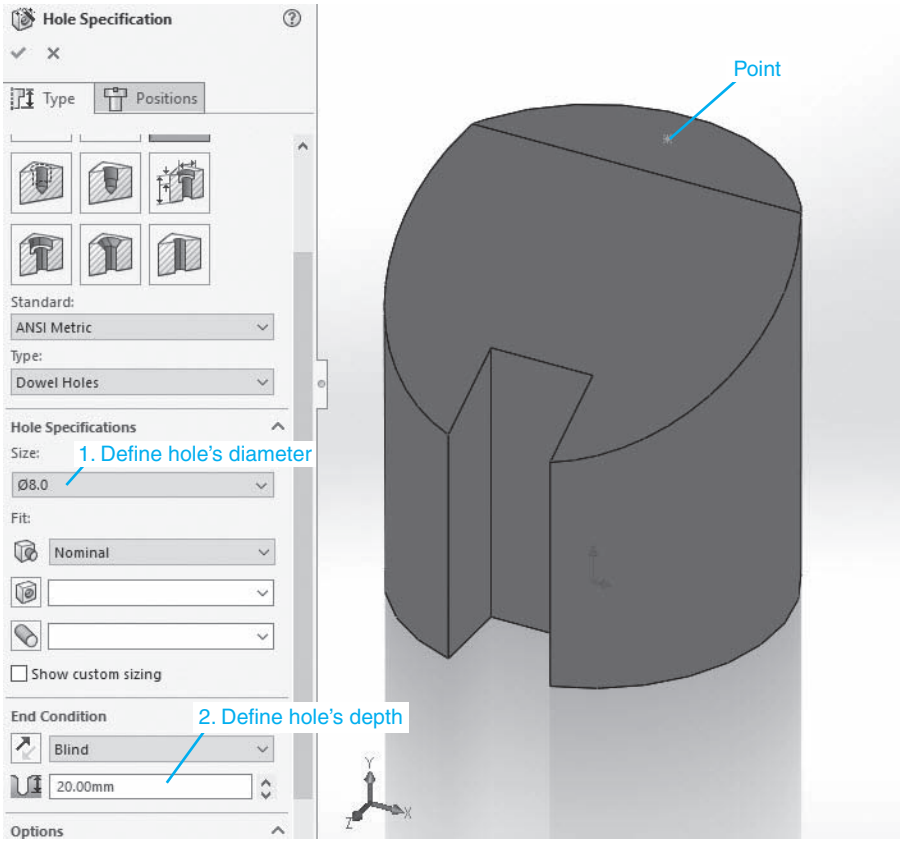
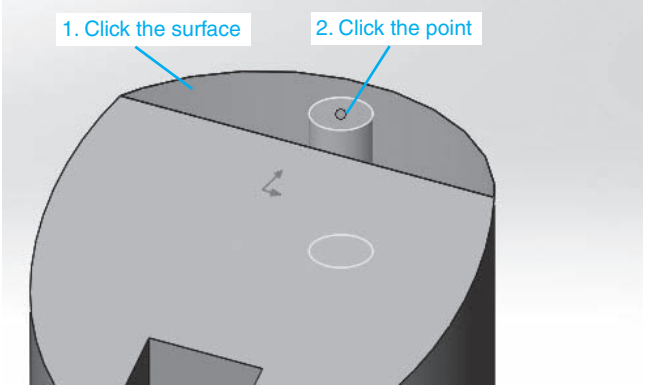


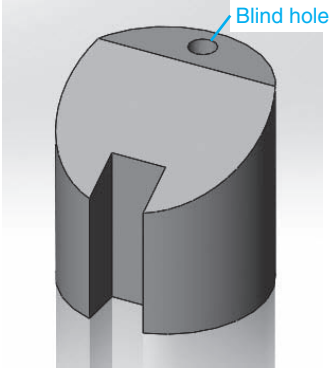
Figure 3-51

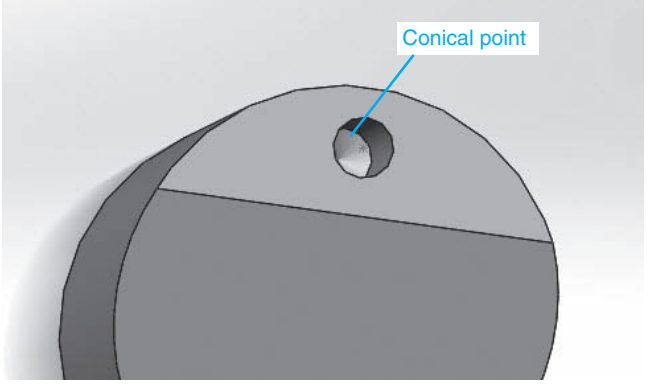
Figure 3-51
(*Continued*)











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.

6. Click the **Hole Wizard** tool on the **Features** tab.
7. Click the **Hole** option and define the hole's diameter and depth.

In this example the hole's diameter is 8.00 and the depth is 20. Note that the hole is defined as a blind hole and that the hole's depth does not include the conical point.

8. Click the **Positions** tab in the **Hole Wizard PropertyManager**.
9. Click the flat top surface, then click the point.
10. Click the green **OK** check mark.
11. Rotate the drawing orientation and verify that the hole has a conical-shaped bottom.

3-27 Sample Problem SP3-3

Figure 3-52 shows a dimensioned object. In this example we will start with the middle section of the object. See Figure 3-53. The solution presented represents one of many possible solutions. The solution uses metric units and ANSI Overall drafting standards.

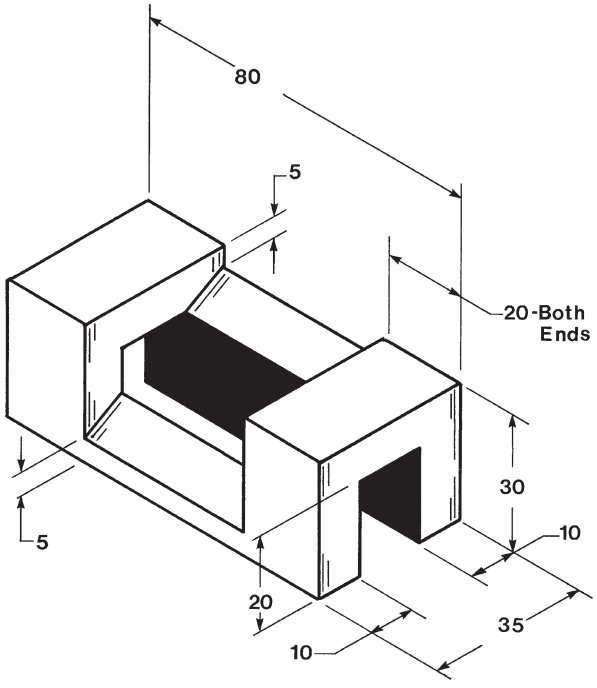


Figure 3-52

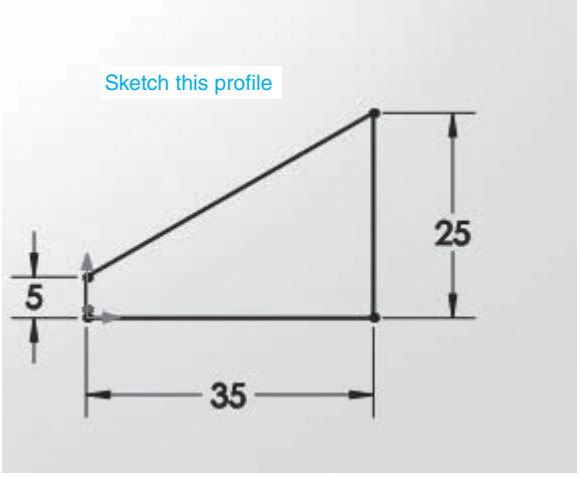
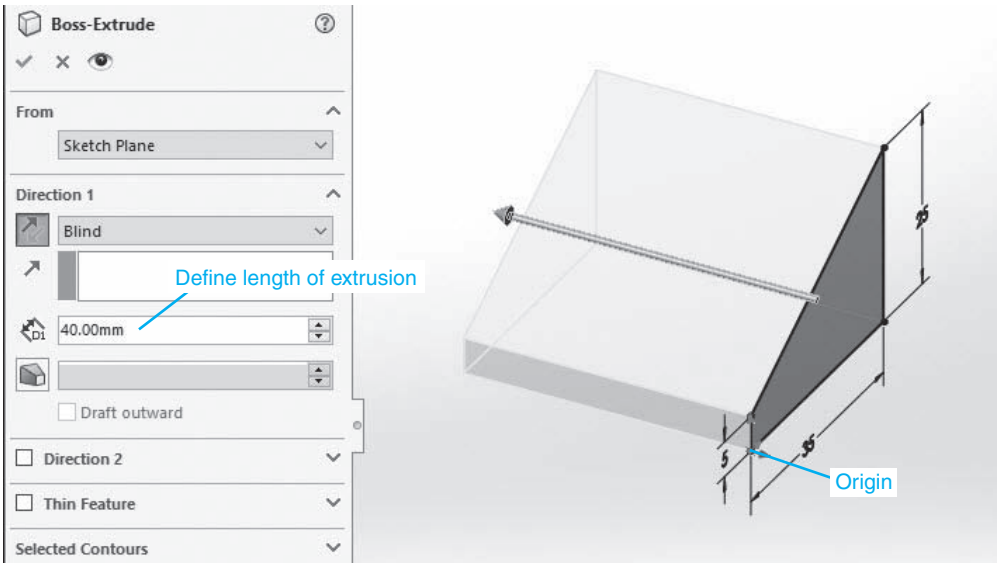
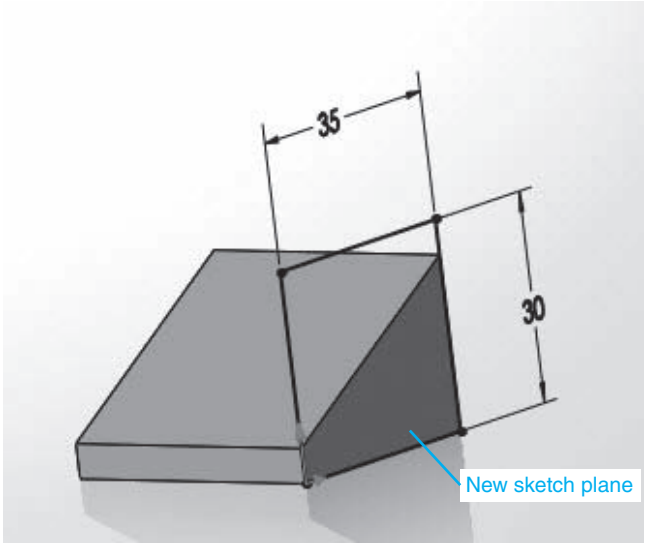
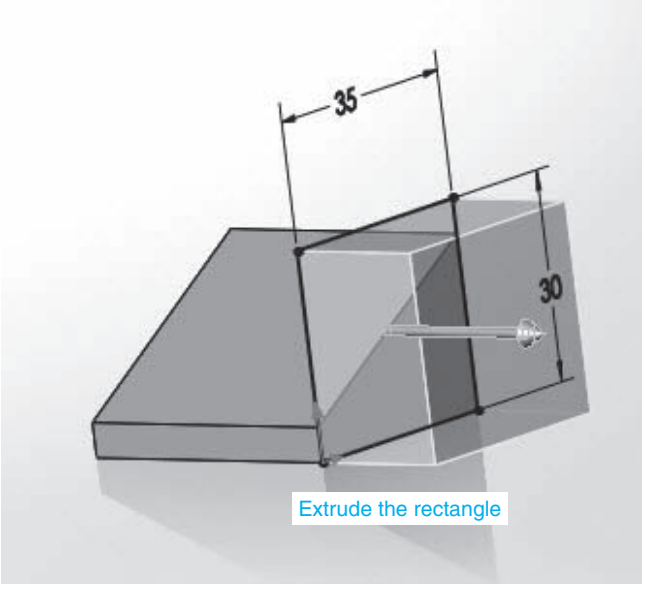


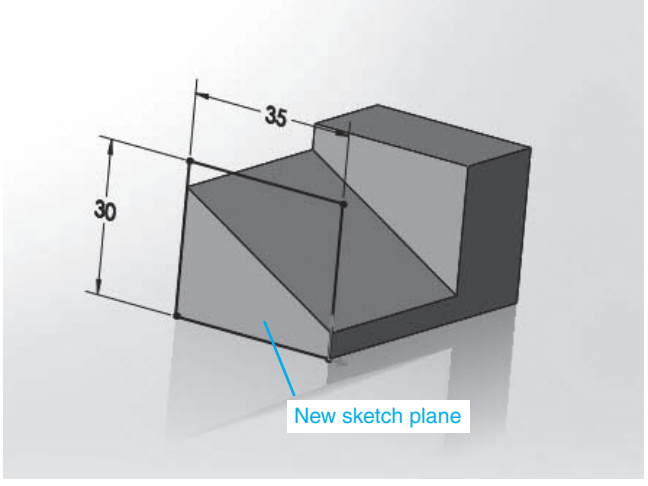
Figure 3-53

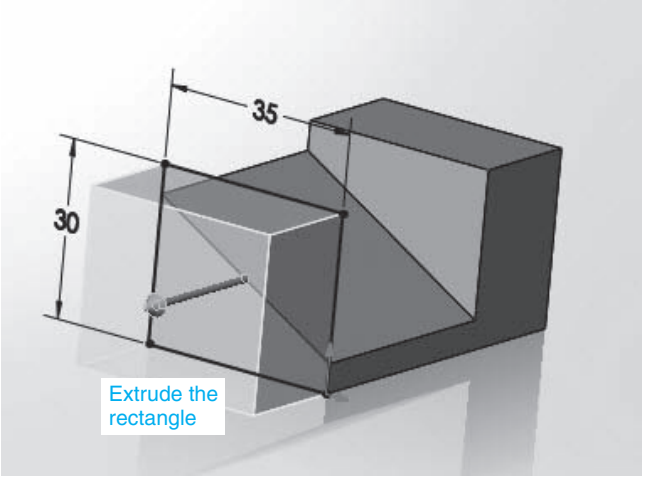
Figure 3-53
(*Continued*)

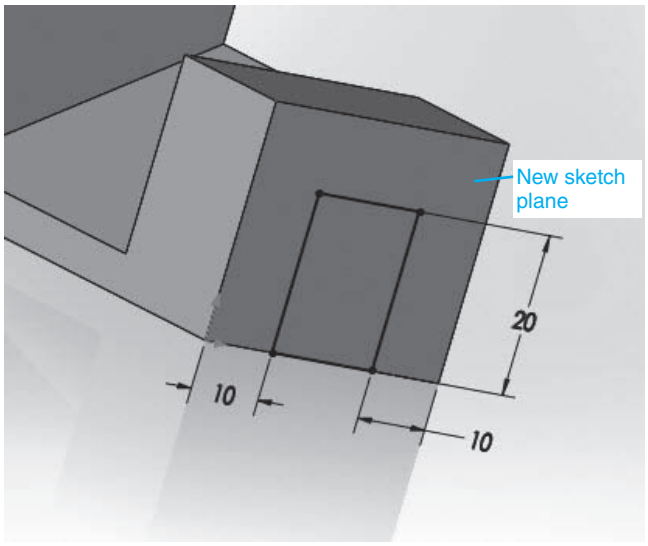


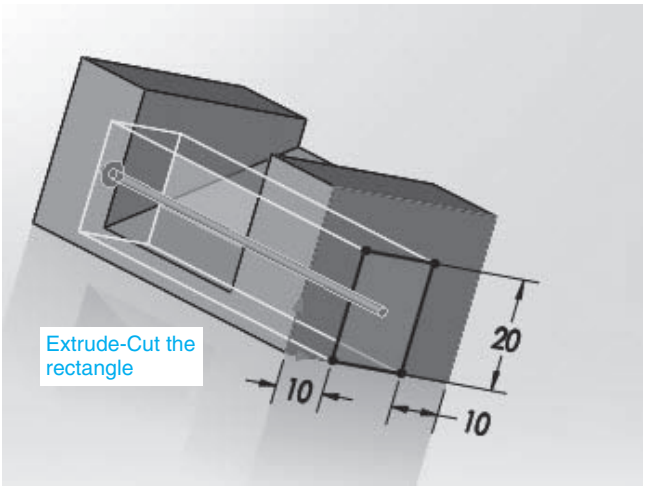












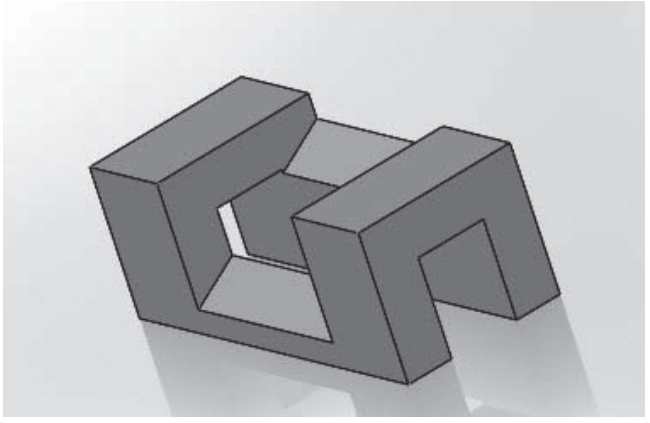


Figure 3-53
(*Continued*)

1. Sketch a profile using the **Right** plane based on the given dimensions.
2. Use the **Extruded Boss/Base** tool to add a thickness of **40** to the profile.
3. Right-click the right surface of the object and select the **Sketch** option.
4. Use the **Rectangle** and **Smart Dimension** tools 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. Reorientate the object and create a sketch plane on the right side of the object. Draw a rectangle based on the given dimensions.
9. Use the **Extruded Cut** tool on the **Features** tab and cut out the rectangle over the length of the object.

3-28 Curve Driven Patterns

Figure 3-54 shows a Ø4.00 inch ring with 12 holes though its side surfaces. The holes were created using the **Curve Driven Pattern** tool.

To Use the Curve Driven Pattern Tool – Example 1

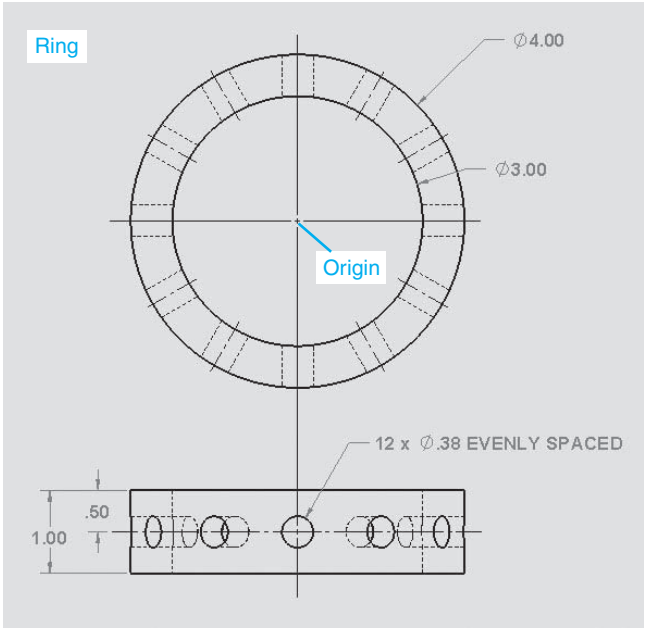
See Figure 3-55.

1. Sketch the ring using the dimensions shown in Figure 3-54.

   The outer diameter is Ø4.00, and the inner ring is Ø3.00. Both are centered on the origin working on the **Top** plane.

2. **Extrude** the ring a distance of 1.00.

3. Click the **Front** plane in the **FeatureManager** box, click the **Plane** option under the **Reference Geometry** tool and create an offset plane tangent to the front edge of the ring.



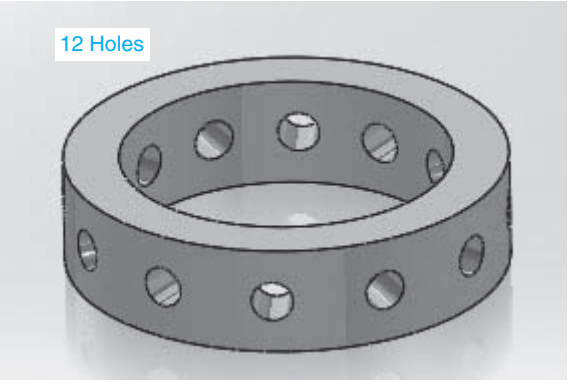
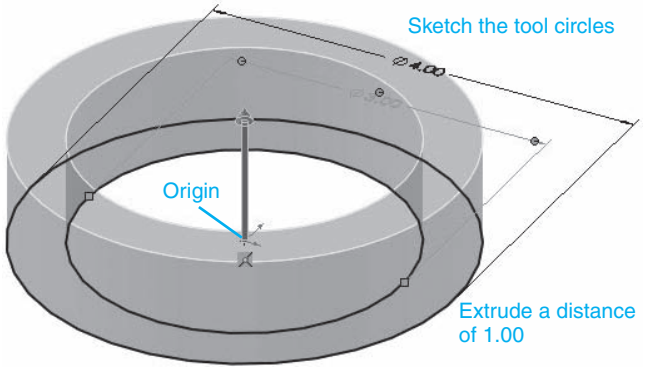


Figure 3-54



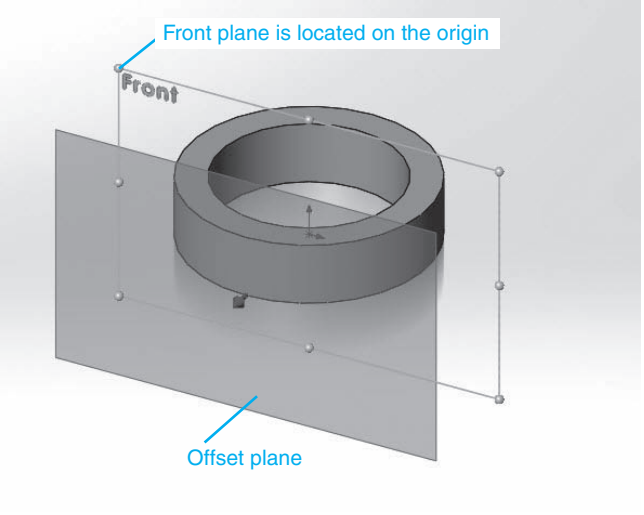
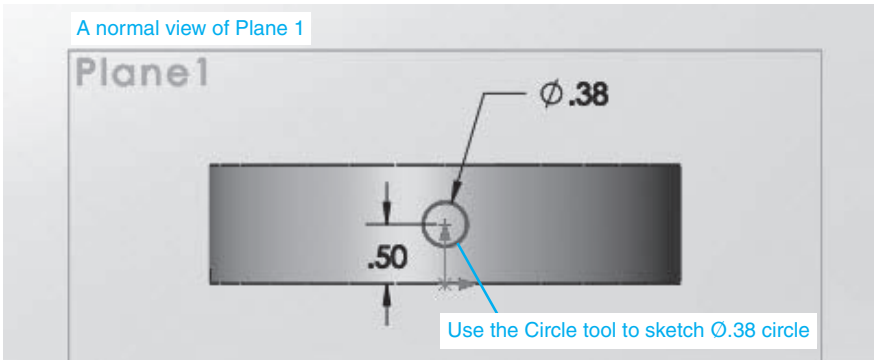
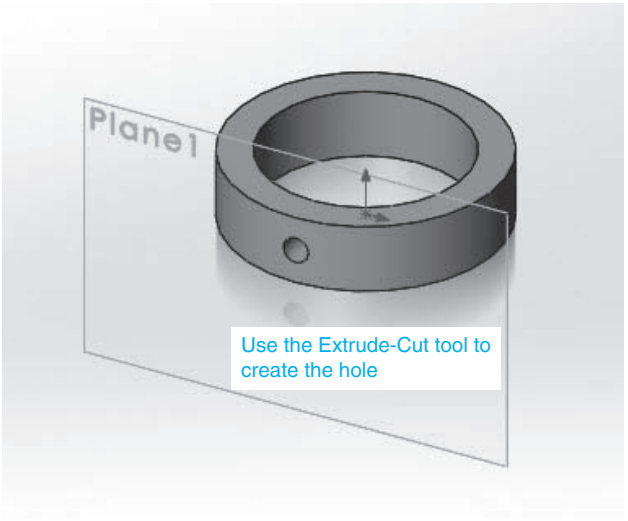
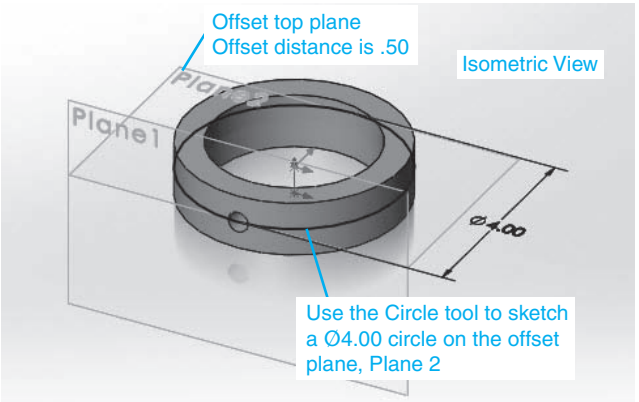


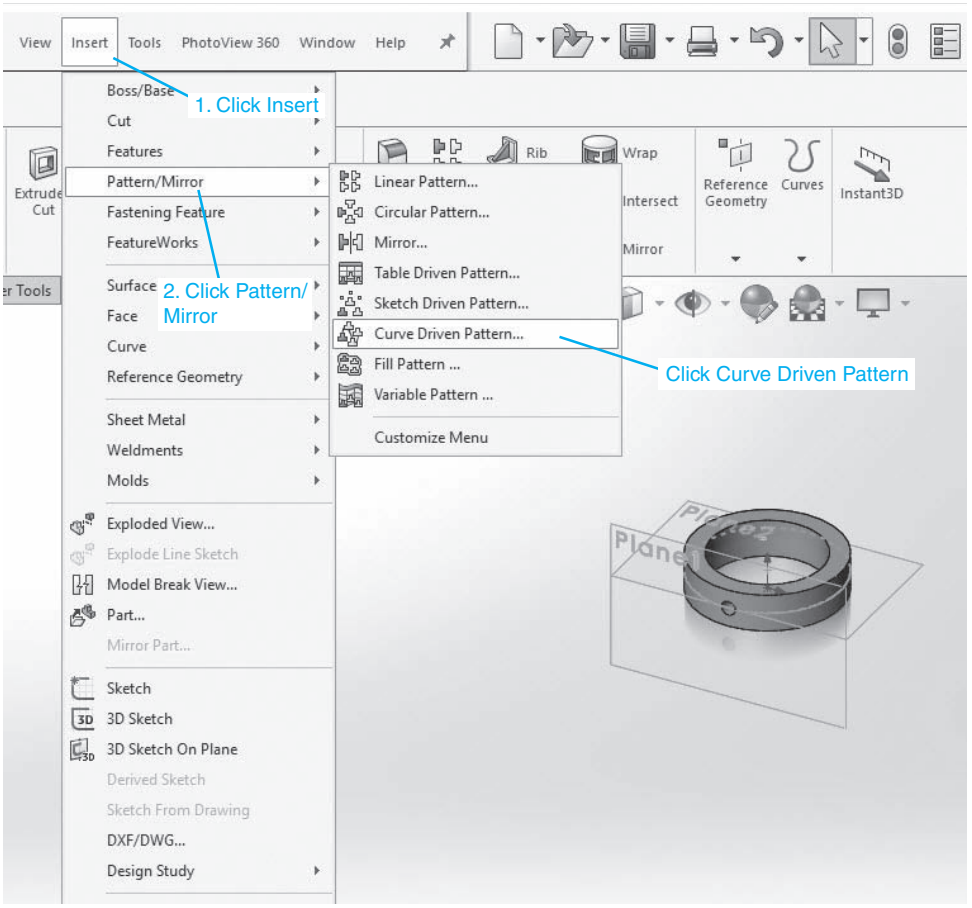
Figure 3-55

Figure 3-55
(*Continued*)









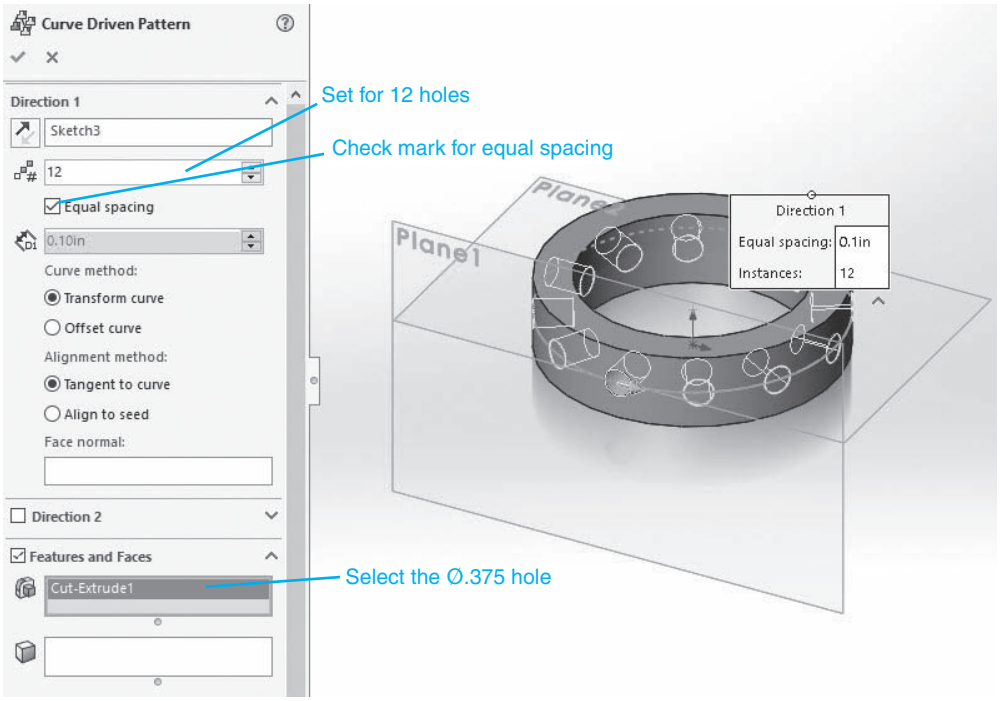


Figure 3-55
(*Continued*)

The new plane is offset 2.00 from the origin, that is, from the center-point of the ring. The new plane is defined as Plane 1 in this example.

4. Create a sketch plane on **Plane 1**, create a normal view to the plane, and sketch a Ø0.375 circle **0.50** from the bottom edge of the ring.
5. Use the **Extruded Cut** tool and create a hole from the circle.
6. Click the **Top plane** in the **FeatureManager** box, click the **Plane** option under the **Reference Geometry** tool, and create an offset Plane **0.50** above the initial Top plane used to create the ring.

In this example this plane is defined as Plane 2. **Plane 2** is offset .50 from the initial top plane or halfway up the 1.00 thickness of the ring.

7. Create a sketch plane on **Plane 2**, and create a normal view.
8. Sketch a Ø4.00 circle on sketch plane, click the **Exit Sketch** icon, and orientate the drawing to an isometric view.

9 Click the **Insert** toolbar heading at the top of the screen, click the **Pattern/Mirror** option, and click the **Curve Driven Pattern** tool.

10 Select the hole as the **Feature to Pattern**, set the **Number of Instances** to **12**, click the **Equal spacing** box, and click **OK**.

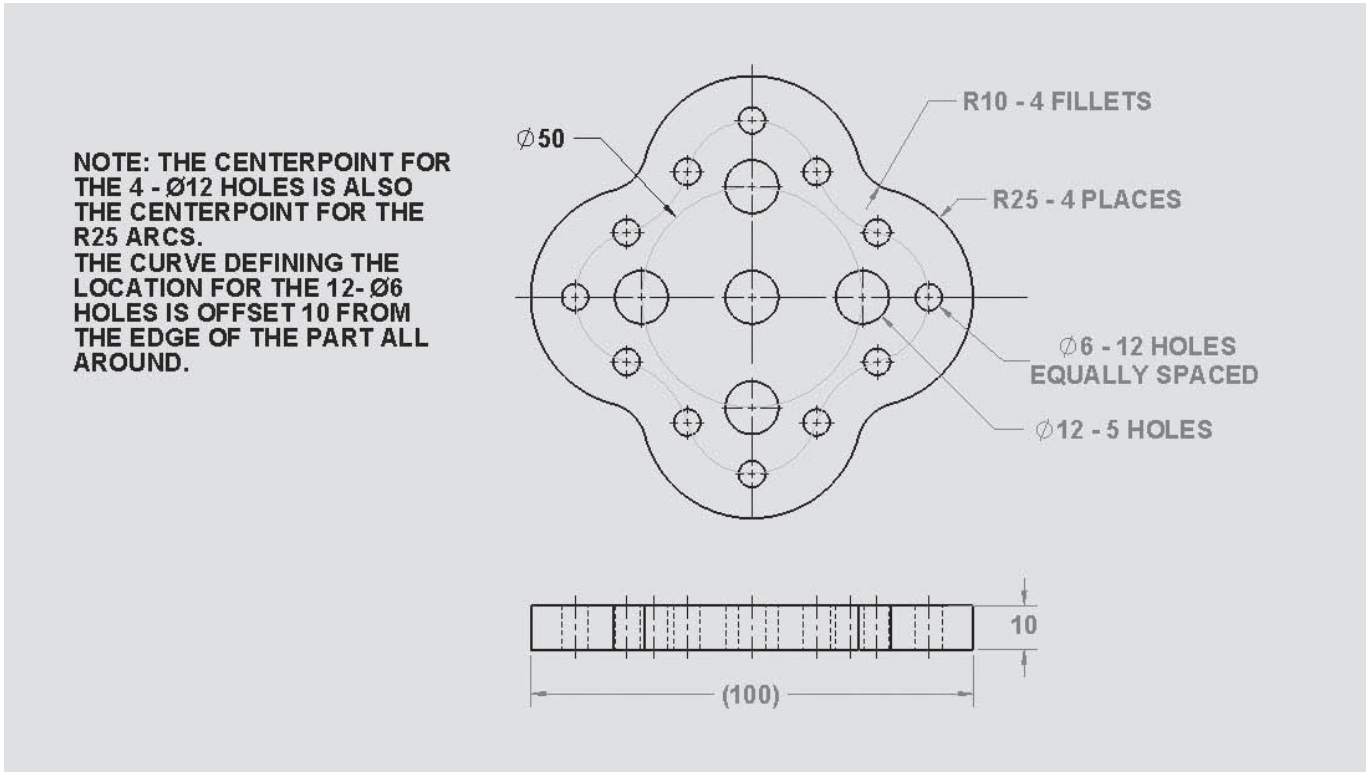
11 Hide Planes 1 and 2, and the circle used to define the pattern.

To Use the Curve Driven Pattern Tool – Example 2

Figure 3-56 shows a part that has 12 holes offset 10 from the part's outer edge surface.

1 Use the given dimensions and draw the part as shown in Figure 3-56.

See Figure 3-57.



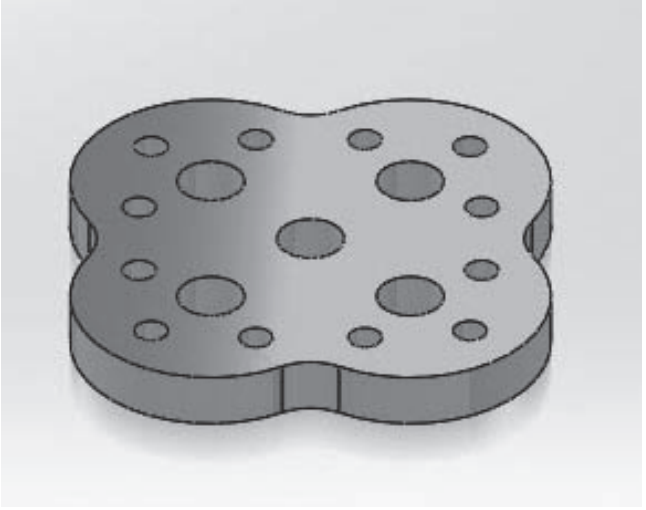
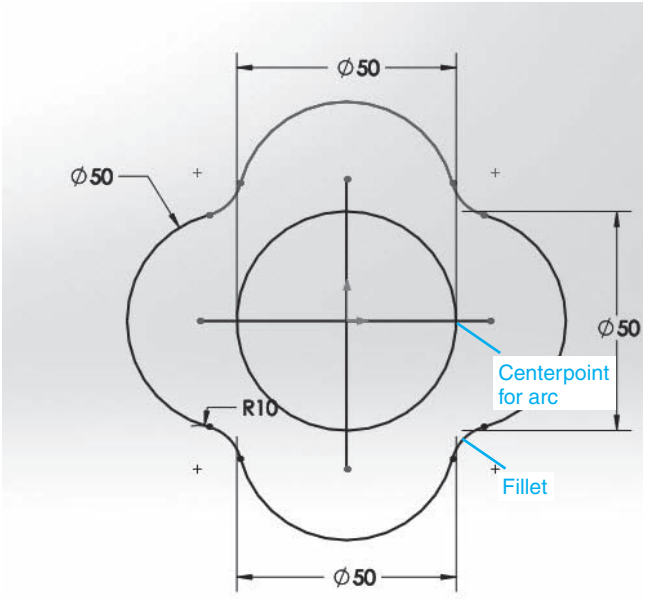
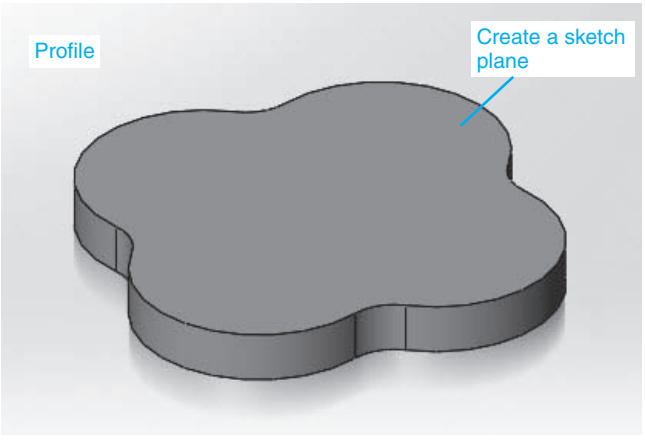
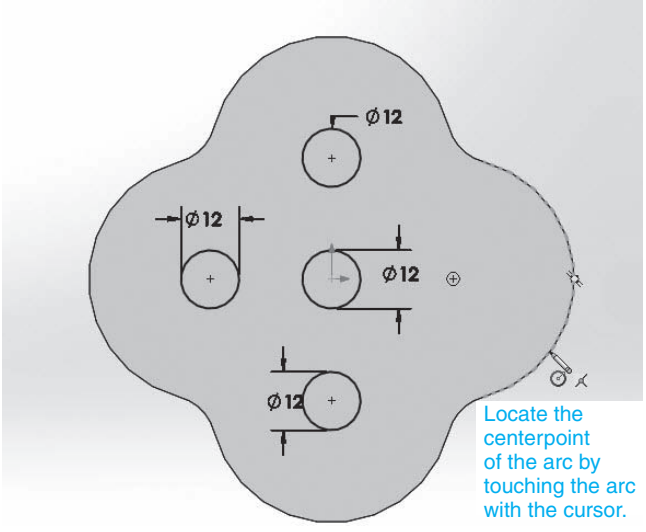
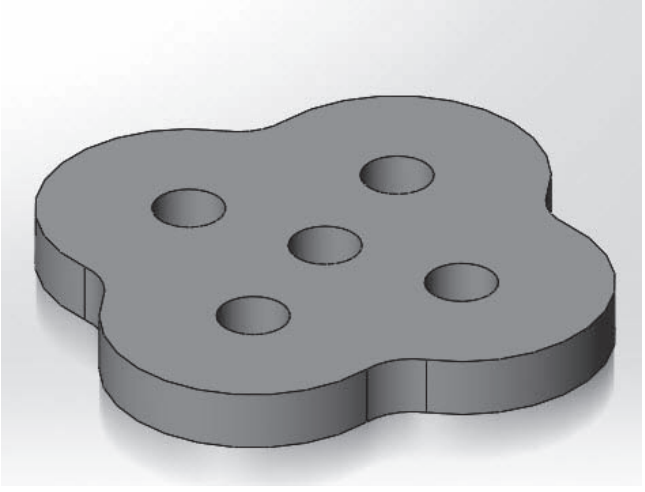


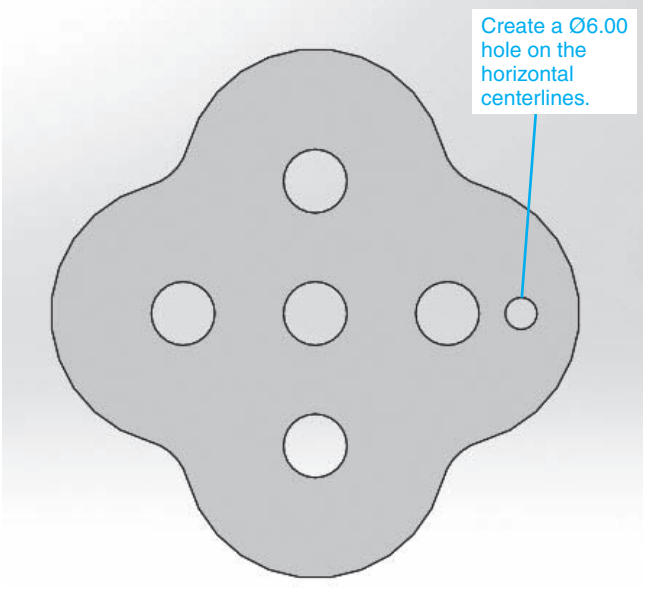
Figure 3-56


Ø50
Ø50
Ø50
R10
Centerpoint for arc
Fillet
Ø50


Profile
Create a sketch plane


Ø12
Ø12
Ø12
Ø12
Locate the centerpoint of the arc by touching the arc with the cursor.




Create a Ø6.00 hole on the horizontal centerlines.

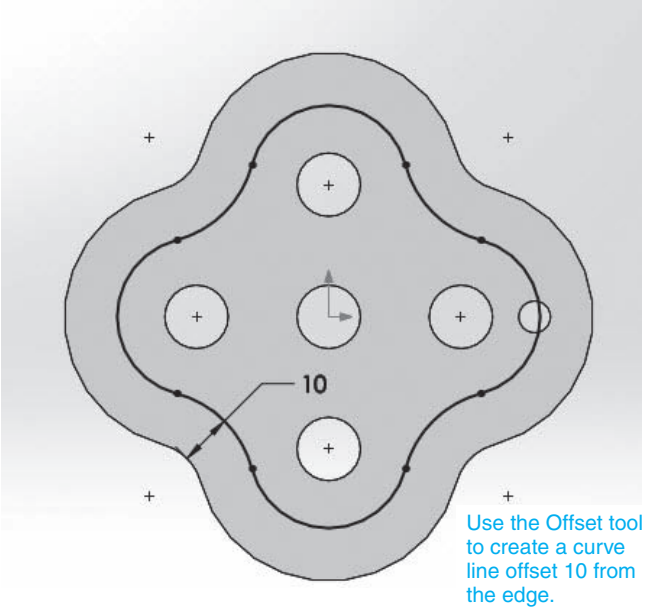
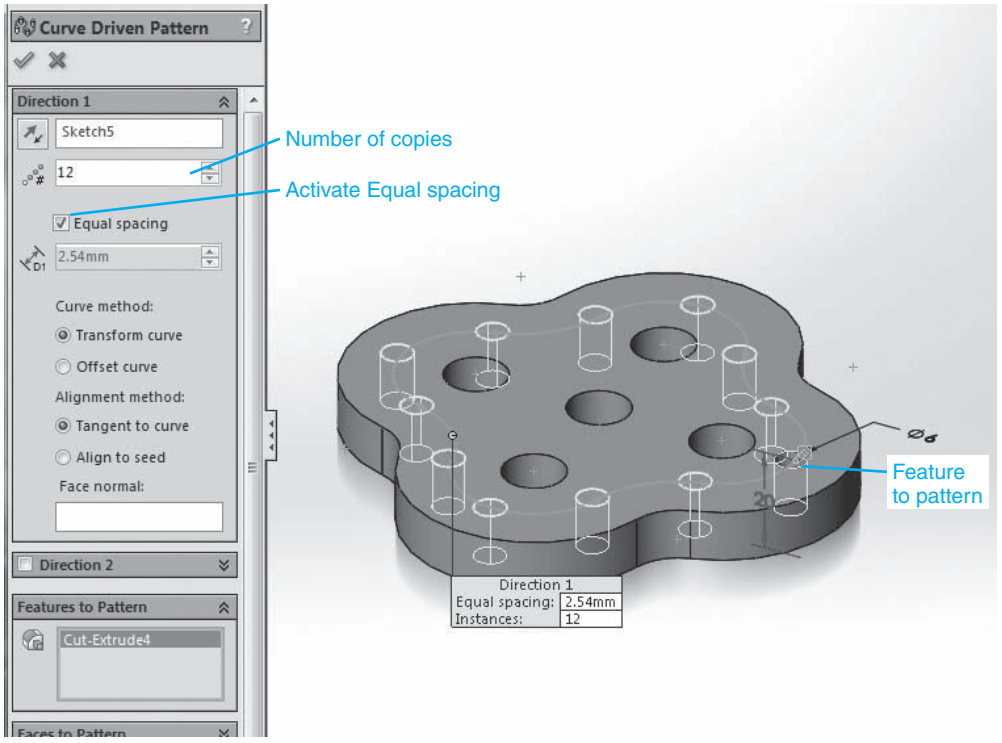
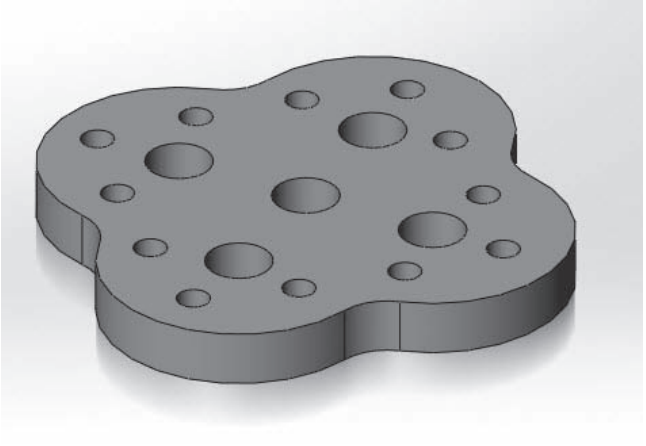

10
Use the Offset tool to create a curve line offset 10 from the edge.

Figure 3-57

Figure 3-57
(*Continued*)





2. Define a sketch plane on the top surface of the part and use the **Offset** tool and create a curve offset **10** from the part's outer edge.

Figure 3-57 shows a normal view of the top surface. The offset curve is created using the **Offset** tool. Both the arcs and fillets can be offset to create a continuous curve.

3. Draw a Ø6 hole centered on the intersection of the offset curve and the horizontal center line of the part. The centerpoint is 40 from the part's origin.

4. Click the **Insert** toolbar heading at the top of the screen, click the **Pattern/Mirror** option, and click the **Curve Driven Pattern** tool.

5. Select the hole as the **Feature to Pattern**, the offset curve as the **Path**, and set the **Number of Instances** to **12**; click the **Equal spacing** box, and click **OK**.

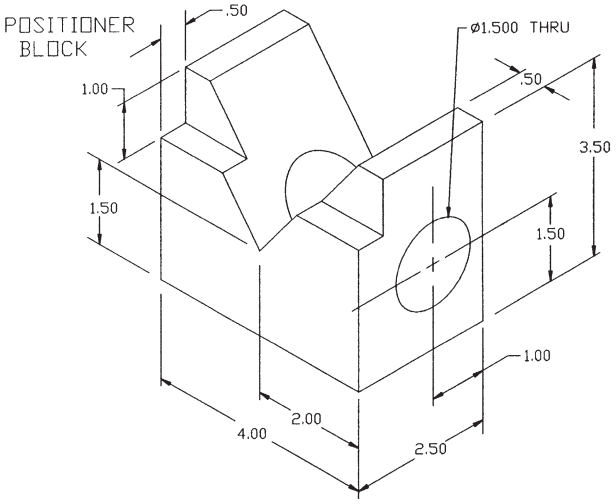


Figure P3-11
INCHES

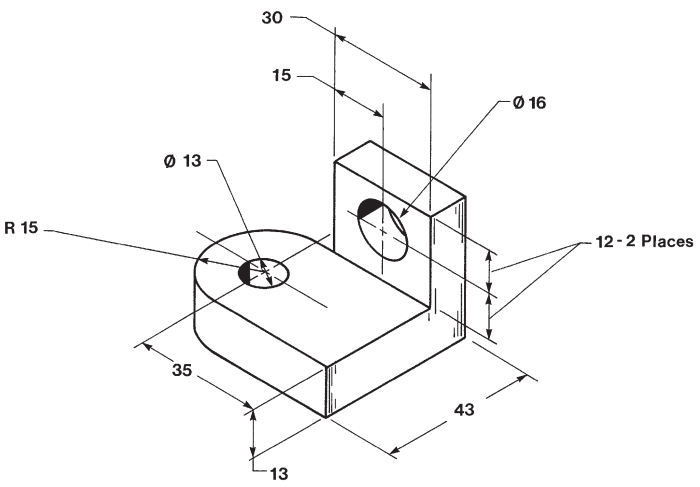


Figure P3-12
MILLIMETERS

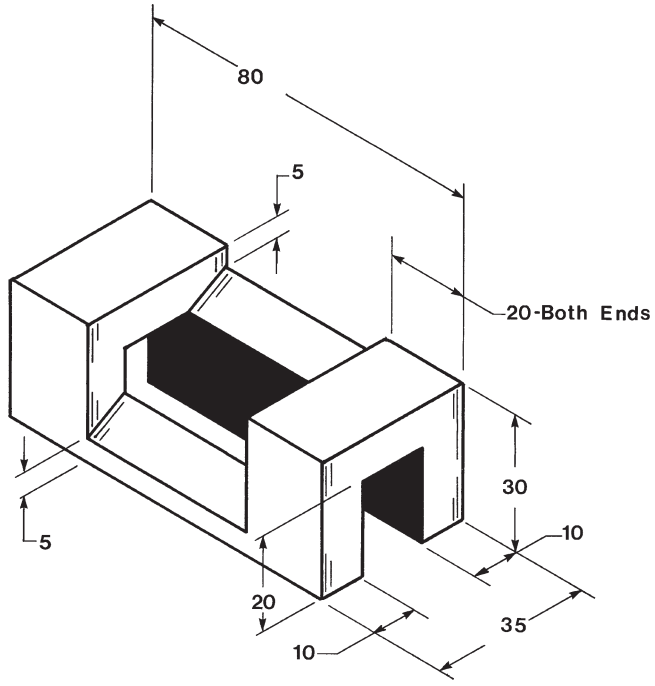


Figure P3-13
MILLIMETERS

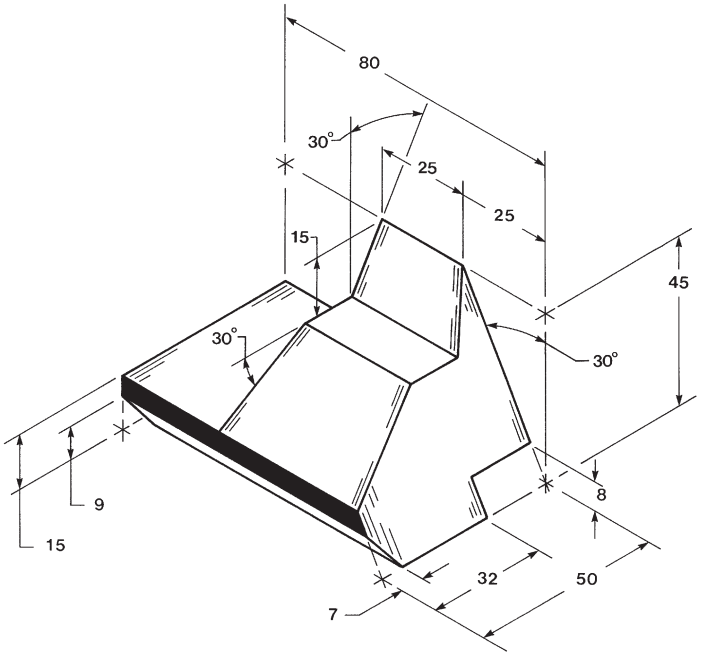


Figure P3-14
MILLIMETERS

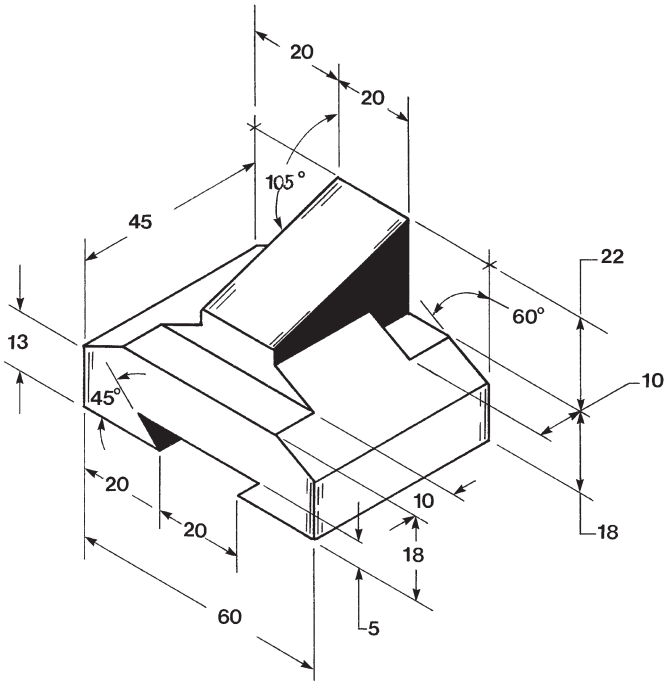


Figure P3-15
MILLIMETERS

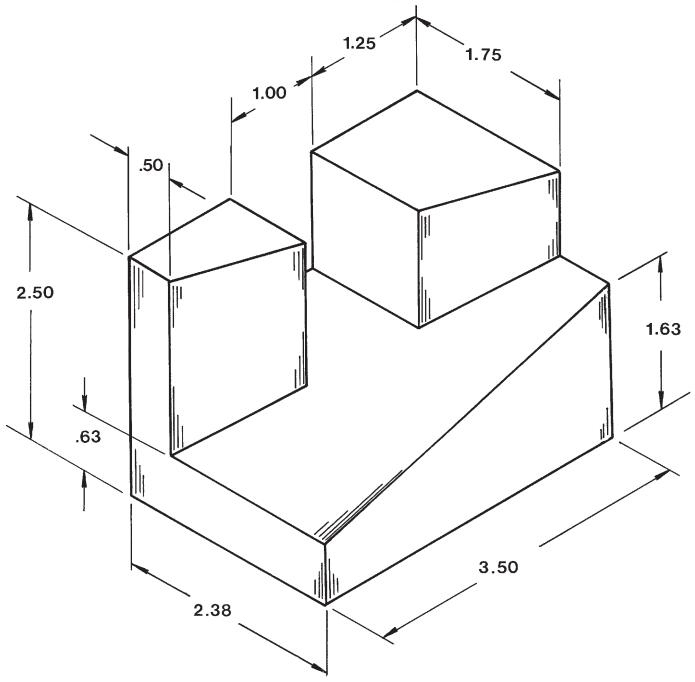


Figure P3-16
INCHES

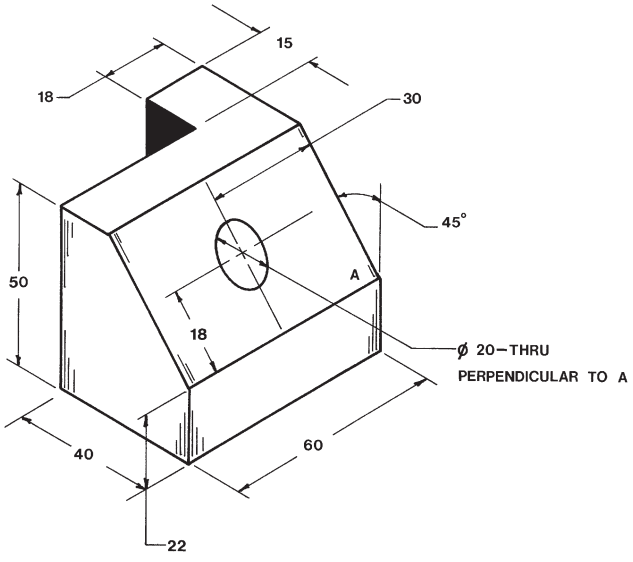


Figure P3-17
MILLIMETERS

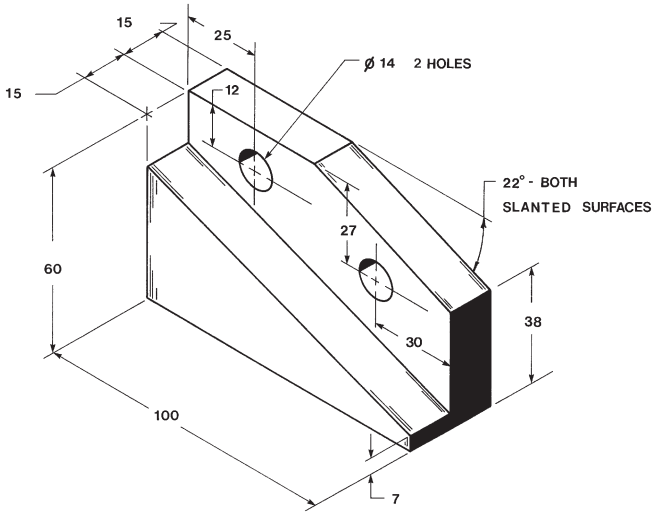


Figure P3-18
MILLIMETERS

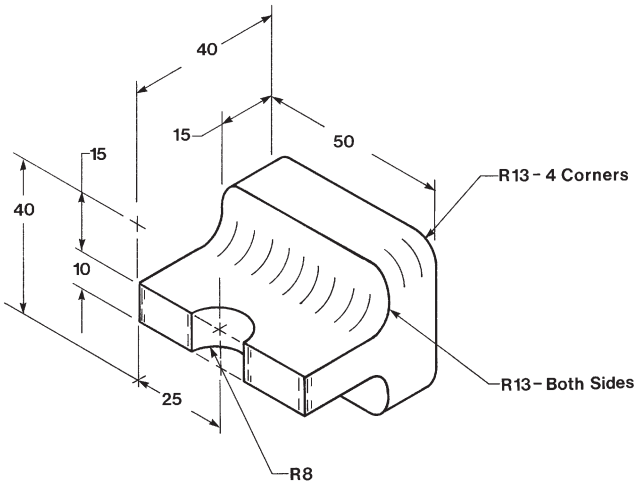


Figure P3-19
MILLIMETERS

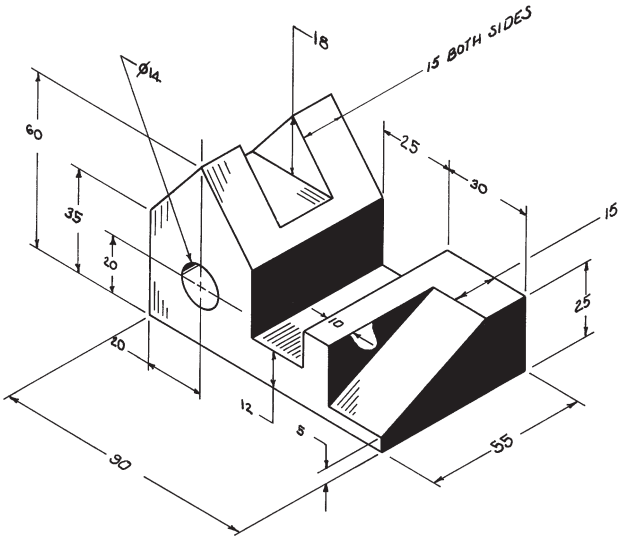


Figure P3-20
MILLIMETERS

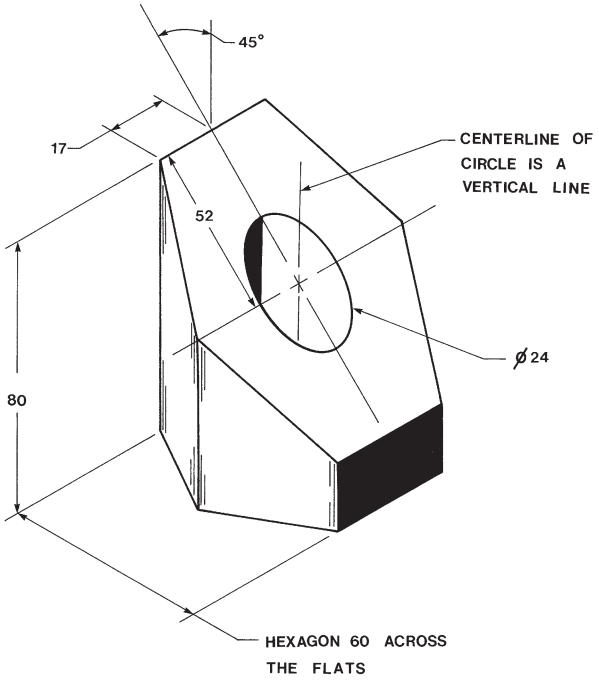


Figure P3-21
MILLIMETERS

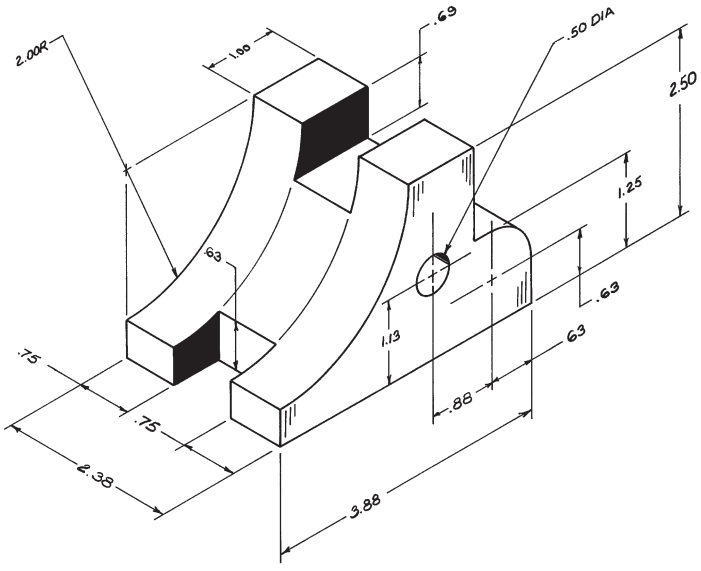


Figure P3-22
INCHES

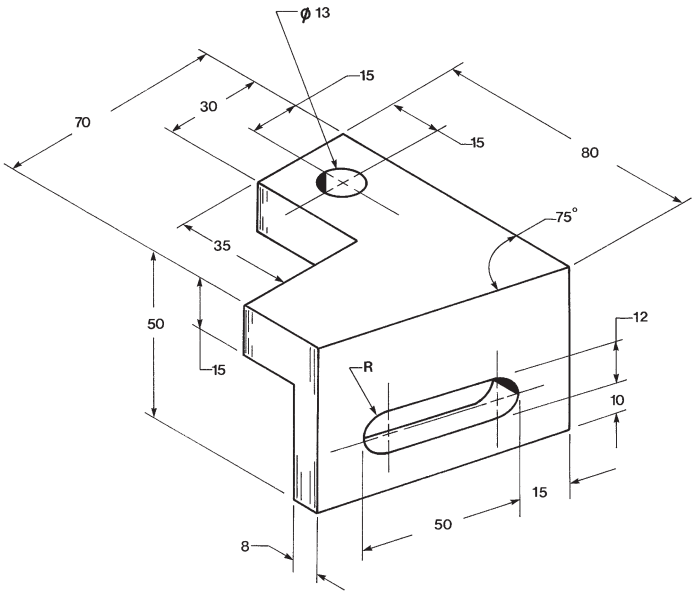


Figure P3-23
MILLIMETERS

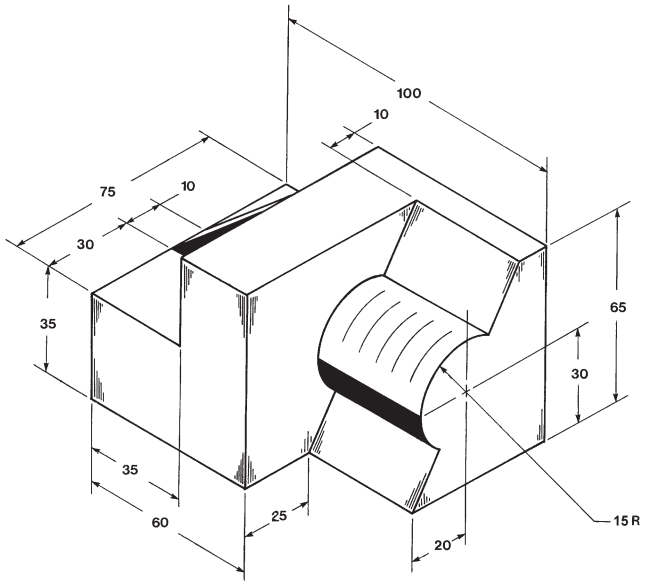


Figure P3-24
MILLIMETERS

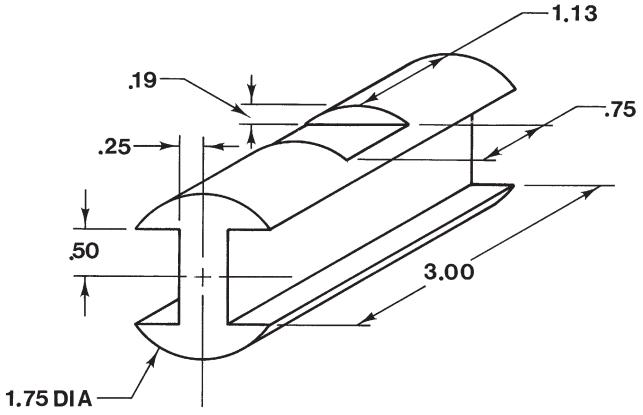


Figure P3-25
INCHES (SCALE: 4 = 1)

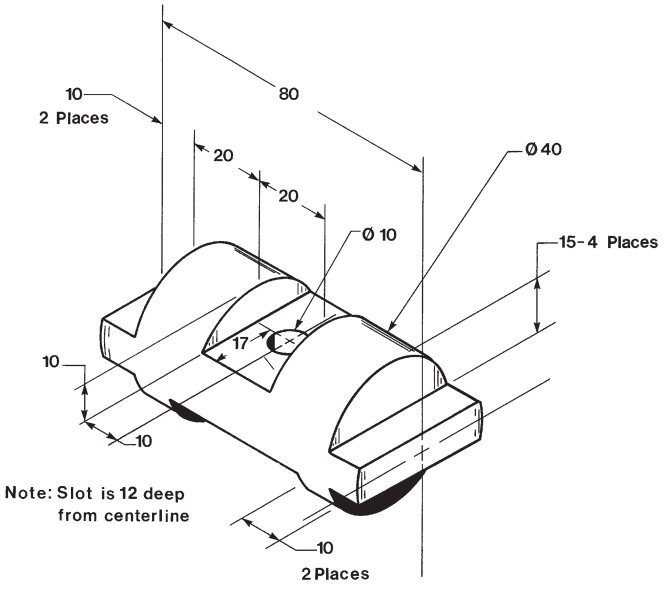


Figure P3-26
MILLIMETERS