CHAPTER 1

Getting Started

Objectives

- Learn how to create a sketch.
- Learn how to create a file/part.
- Learn how to create a solid model.
- Learn how to edit and modify a sketch.
- Learn how to draw angular and circular shapes.
- Learn how to draw holes.
- Learn how to use the Sketch tools.
- Change units of a part.

1-1 INTRODUCTION

This chapter presents a step-by-step introduction to SolidWorks 2008. The objective is to have first-time users access SolidWorks and be able to start drawing shapes within a few minutes. The use of the tools initially presented in Chapter 1 will be expanded in Chapters 2 and 3.

1-2 SKETCHING A LINE

Figure 1-1 shows the opening SolidWorks screen. This screen should appear when you first access the SolidWorks program. Move the cursor to the icon in the upper left corner of the screen located under the heading **Files**. This is the **New** tool. It is used to create a new drawing.

1. Click the **New** tool.

The **New SolidWorks Document** dialog box will appear. See Figure 1-2.

2. Click the **Advanced** box located in the lower left corner of the box.

TIP

The **Advanced** tool access box may not appear after your first use. SolidWorks will go directly to the **New SolidWorks Document** box. The **Novice** box can be used to return to the **New SolidWorks Document** box if needed.

The next **New SolidWorks Document** dialog box will appear. See Figure 1-3. SolidWorks can generate three different types of drawings: **Part**, **Assembly**, and **Drawing** documents. Individual parts are drawn using the **Part** document. This section will use **Part** documents. **Assembly** and **Drawing** documents will be covered in later chapters.

3. Click the **Part** tool, then click **OK**.

The initial screen display will appear. See Figure 1-4. This screen shows the components of a new **Part** document,

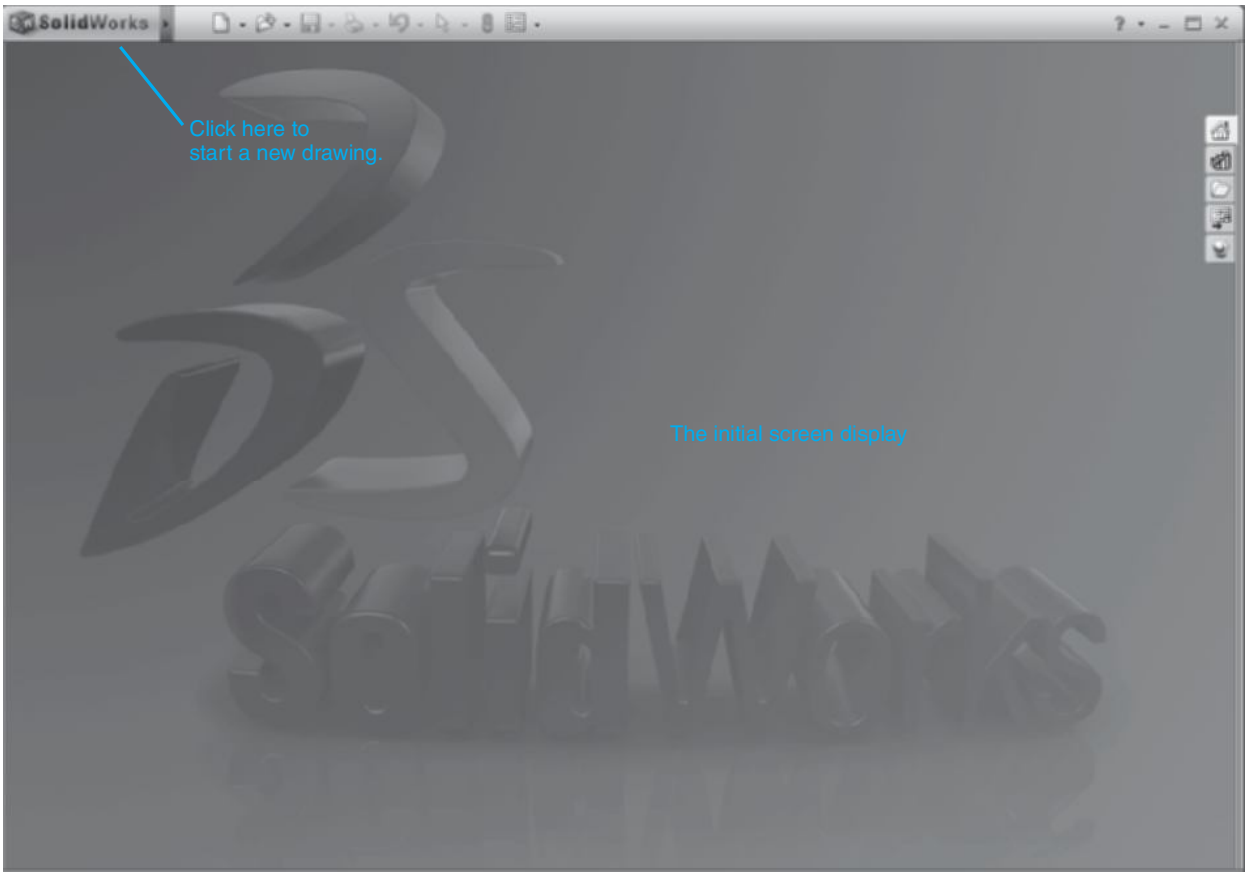


Figure 1-1

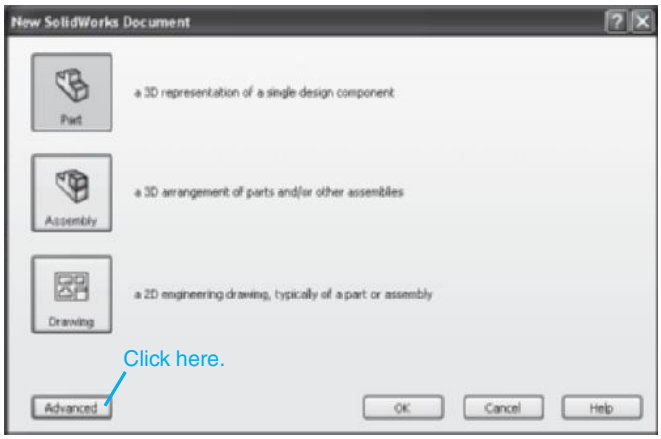


Figure 1-2

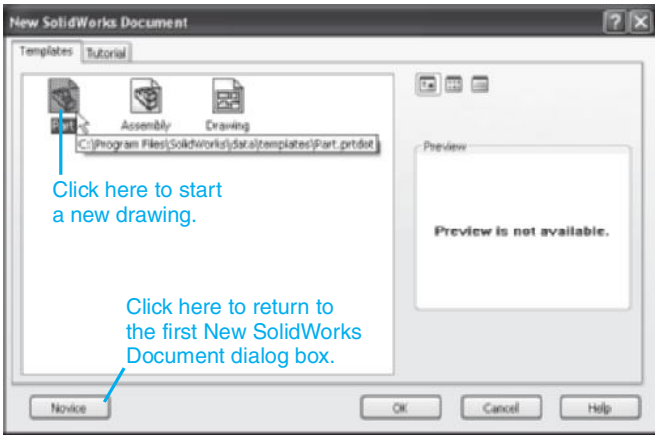


Figure 1-3

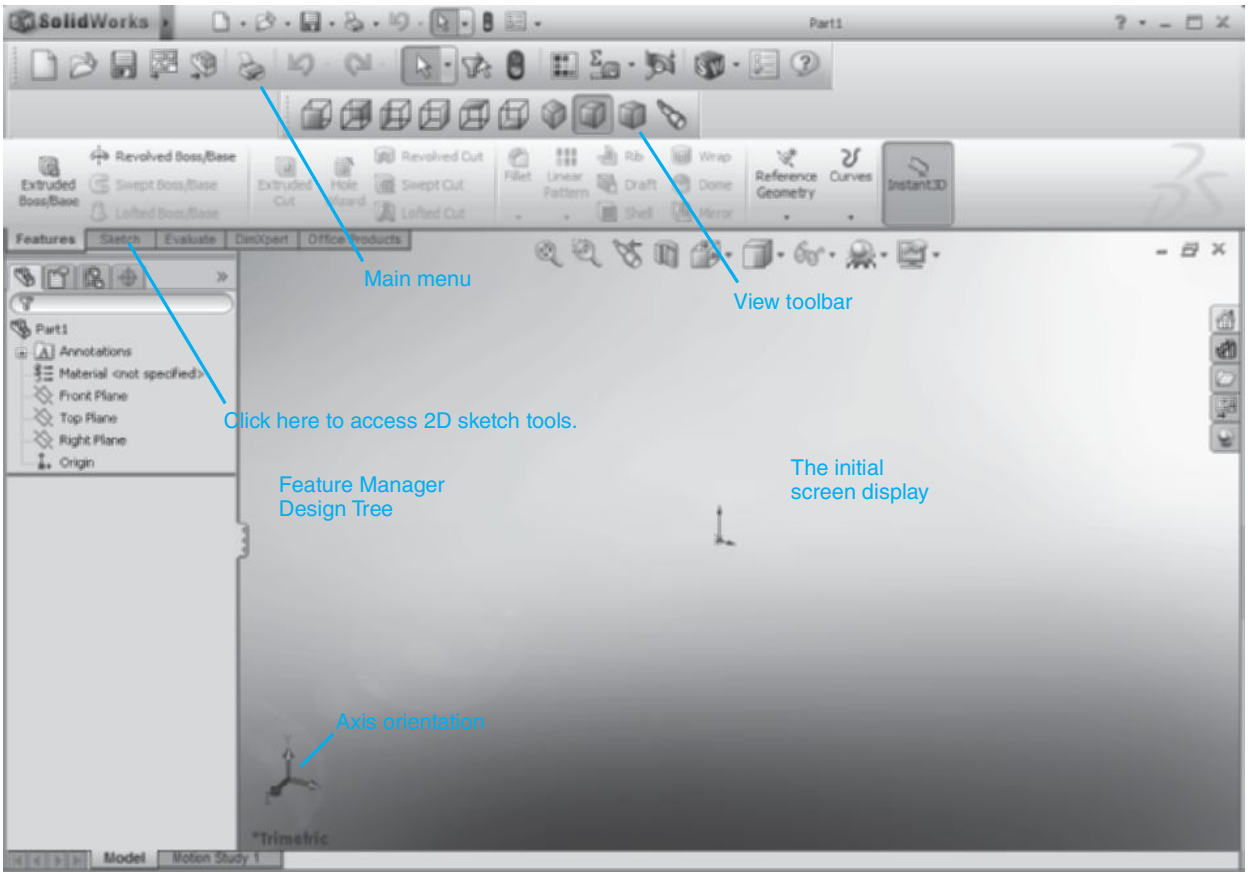


Figure 1-4

which includes toolbars, the **Command Manager,** main menu headings, the **Features Manager**, and the axis orientation icon.

4. Click the **Sketch** tool located on the **Command Manager**.
5. Click the **Front Plane** tool located on the **Features Manager**.

A reference plane will appear. See Figure 1-5. The plane appears in a trimetric orientation but will be automatically oriented normal (at right angles to) the selected view once sketching begins. There are three basic sketching planes: front, top, and right side. These views correspond to the three basic orthographic views that will be covered in Chapter 4.

6. Click the **Sketch** tool again.

A grouping of 2D sketching tools will appear on the **Command Manager**. See Figure 1-6.

7. Click the **Line** tool.

The front plane will rotate normal to a 2D sketching mode. Figure 1-7 shows the default screen display in the sketching environment.

Note:

The triangular-shaped area in the upper right corner of the drawing screen indicates that the document is in sketch mode.

8. Locate the cursor in the drawing area, and select a starting point for the line.
9. Click the left mouse button to start the line, and move the cursor horizontally across the screen. Determine an endpoint for the line and again click the mouse button. Click the green check mark on the **Features Manager**, press the **<Esc>** key, or right-click the mouse and click the **Select** option to end the **Line** tool.

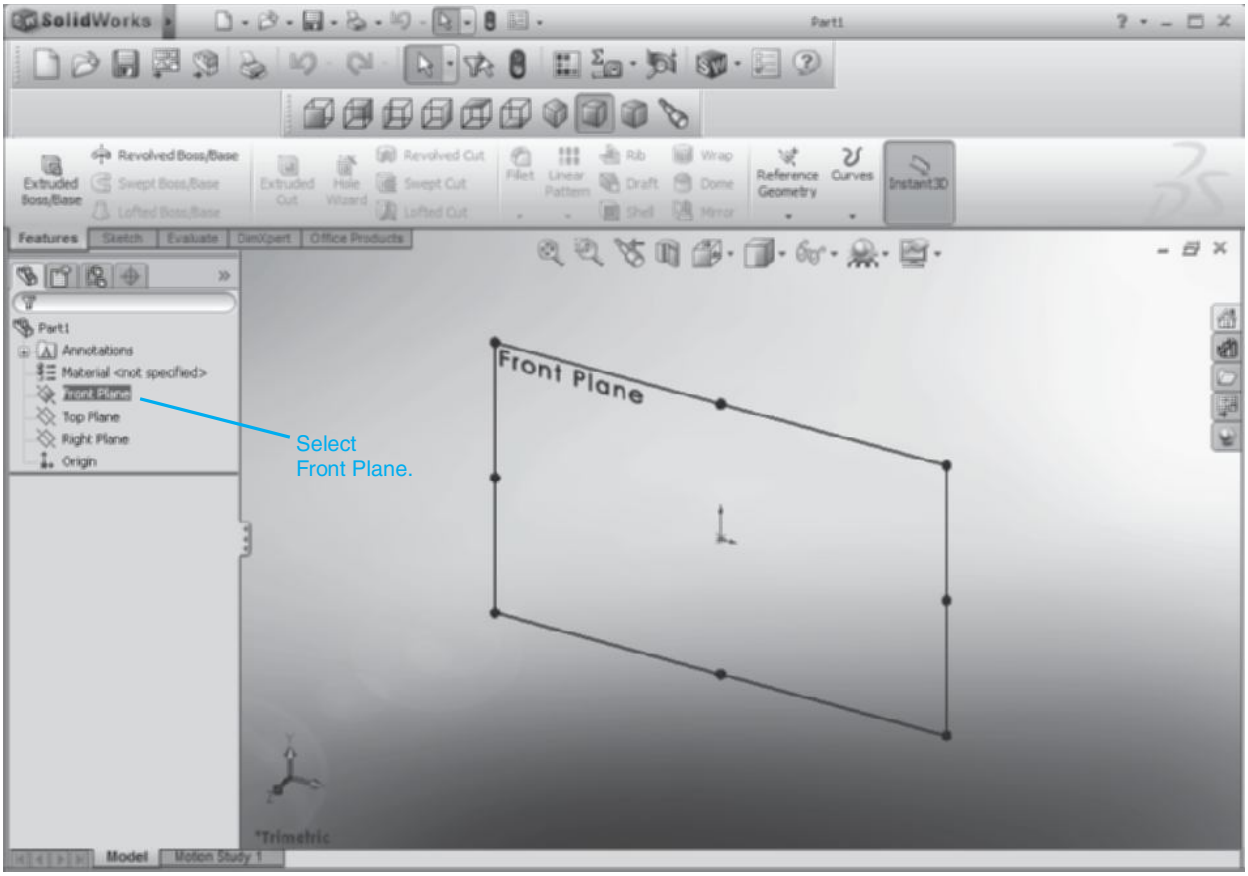


Figure 1-5

TIP

Lines can also be drawn by selecting a starting point and holding the left mouse button down as the cursor is moved. The end of the line is defined by releasing the mouse button.

Note:

As you sketch, the line will change colors. The colors help you determine the status of the line. When you initially draw a line it will be green, meaning it has not been dimensioned. If you press the **<Esc>** key or start a continuing line, the line will turn blue, indicating that you have accepted the sketched length. If you pass the cursor over an existing line, the line will turn red, indicating that the line is activated and may be edited. If you click the **Smart Dimension** tool and move the cursor to a line, the line will initially turn red and be identified, then green when it is clicked, indicating it can be modified.

Text will appear as you draw the line indicating the length of the line and its angle. Make the line about 4 in. long at 180°. See Figure 1-8.

Note:

The examples given in this chapter are dimensioned in inches. To change units,

1. Click the **Tools** heading at the top of the screen and select **Options**.
2. Click the **Document Properties** tab, then **Units**, and select the desired **Unit system** radio button.
3. Click **OK**.

See Section 1-9 for a more detailed expalanation.

TIP

This drawing is a sketch, so exact dimensions are not required. The **Smart Dimension** tool will be used to define an exact length for the line.

The small shaded square with the horizontal bar across it indicates that the line is a horizontal line.

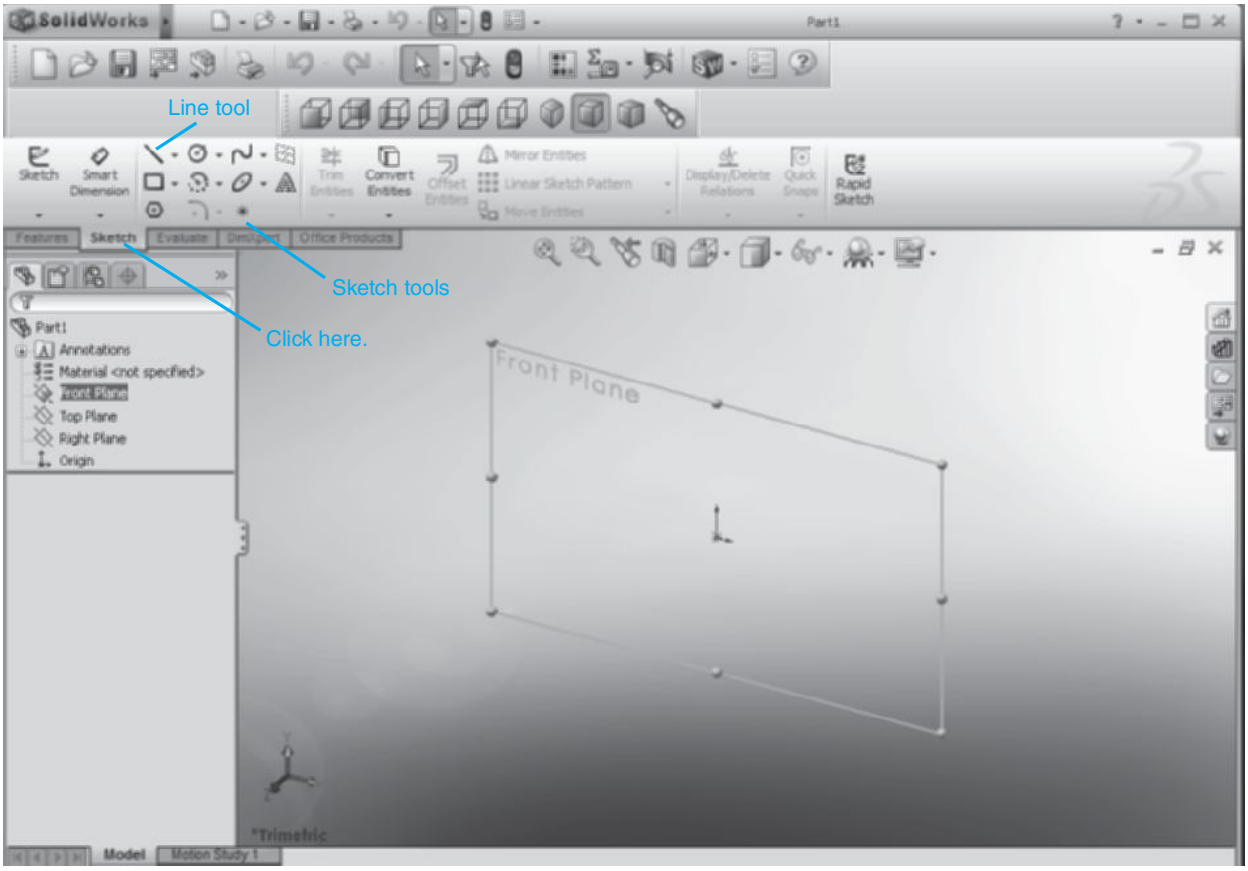


Figure 1-6

1-3 MODIFYING A LINE

The line created in Figure 1-8 is a sketched line; that is, it has an approximate length. We will now define an exact length for the line.

1. Click the **Smart Dimension** tool in the **Sketch** group on the **Command Manager**.

   See Figure 1-8.

2. Click on the line and move the cursor upward away from the line.
3. Determine a location for the line's dimension and click the mouse.

   The **Modify** dialog box will appear. See Figure 1-9.

4. Enter a dimension value of **4.00** and click the check mark in the lower left of the **Modify** dialog box.

TIP

The dimension for the line is in inches. The units can be changed to millimeters. Millimeters will be applied in a later example.

The line's length will be defined as 4.00 in. The line's length will be modified to this length. See Figure 1-10.

5. Click an open area of the drawing screen or press the **<Esc>** key.

The line is now drawn and sized (dimensioned). We will now close the drawing and create another drawing.

6. Select the **File** heading on the main menu.

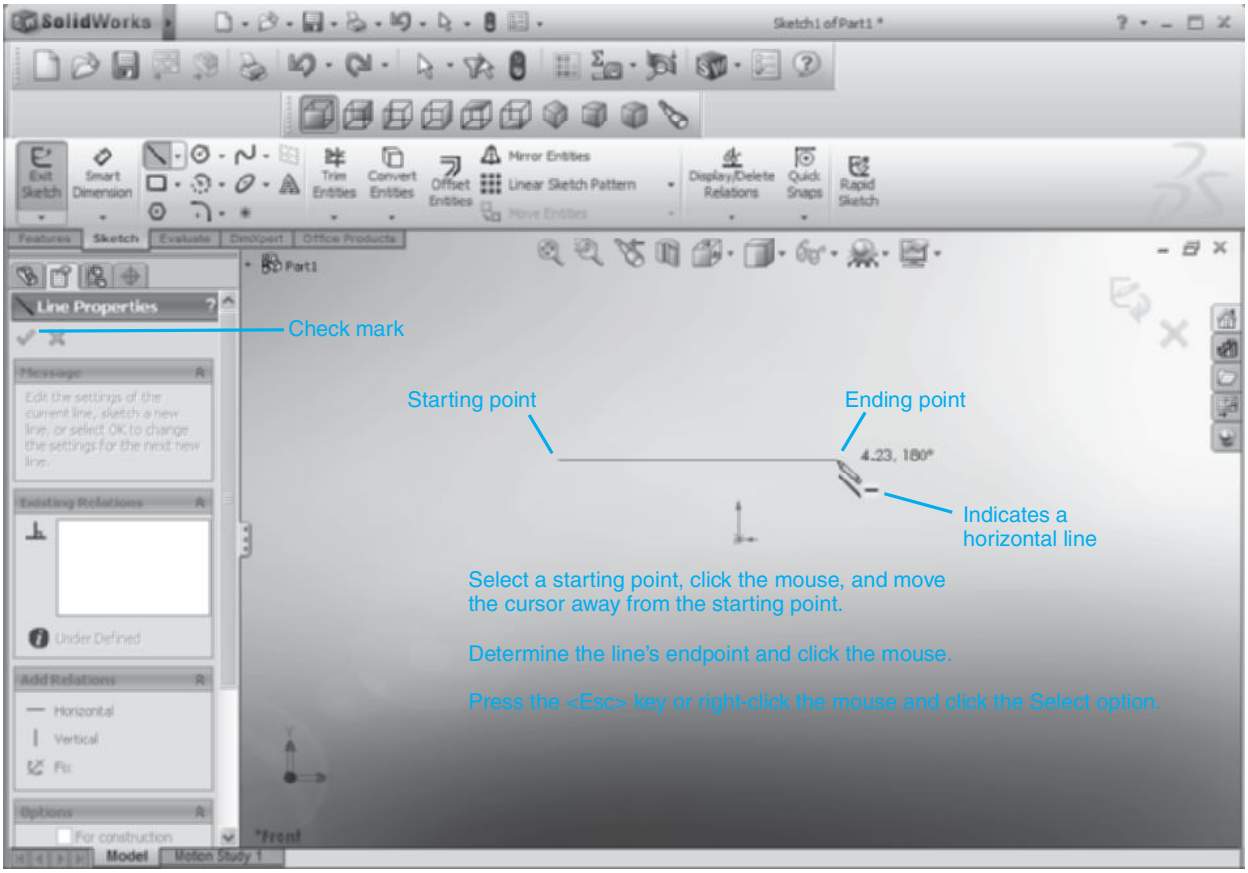


Figure 1-7

A series of commands will cascade down. See Figure 1-11.

7. Select the **Close** tool.

   A dialog box will appear on the screen. See Figure 1-12.

8. Select the **No** option unless you want to save the line.

1-4 THE RECTANGLE TOOL

Start a new **Part** document file as defined in Section 1-2. Click the **Sketch** group on the **Command Manager** to display the **Sketch** tools. Select **Front Plane** from the **Features Manager**. See Figure 1-13.

1. Click the **Rectangle** tool in the **Sketch** group on the **Command Manager**.
2. Use the **Corner Rectangle** tool to sketch a rectangle by clicking a selected starting point, dragging the cursor down and across the screen, and selecting an endpoint for the rectangle by releasing the mouse button.

   See Figure 1-14.

3. Click the **Smart Dimension** tool and create a **3.00 × 5.00-in**. rectangle. See Figures 1-15, 1-16, and 1-17.
4. Click the OK check mark on the **Features Manager**, or right-click the mouse and click the **Select** option.
5. Access the **View** toolbar, usually located at the top of the screen.

The **View** toolbar defines 10 different orientations that can be applied to the screen.

6. Select **Isometric**.

The rectangle will change to an isometric orientation. See Figure 1-19. Now, we will extend the first shape to create a solid feature.

Creating a Solid

1. Click the **Features** tool on the **Command Manager**.

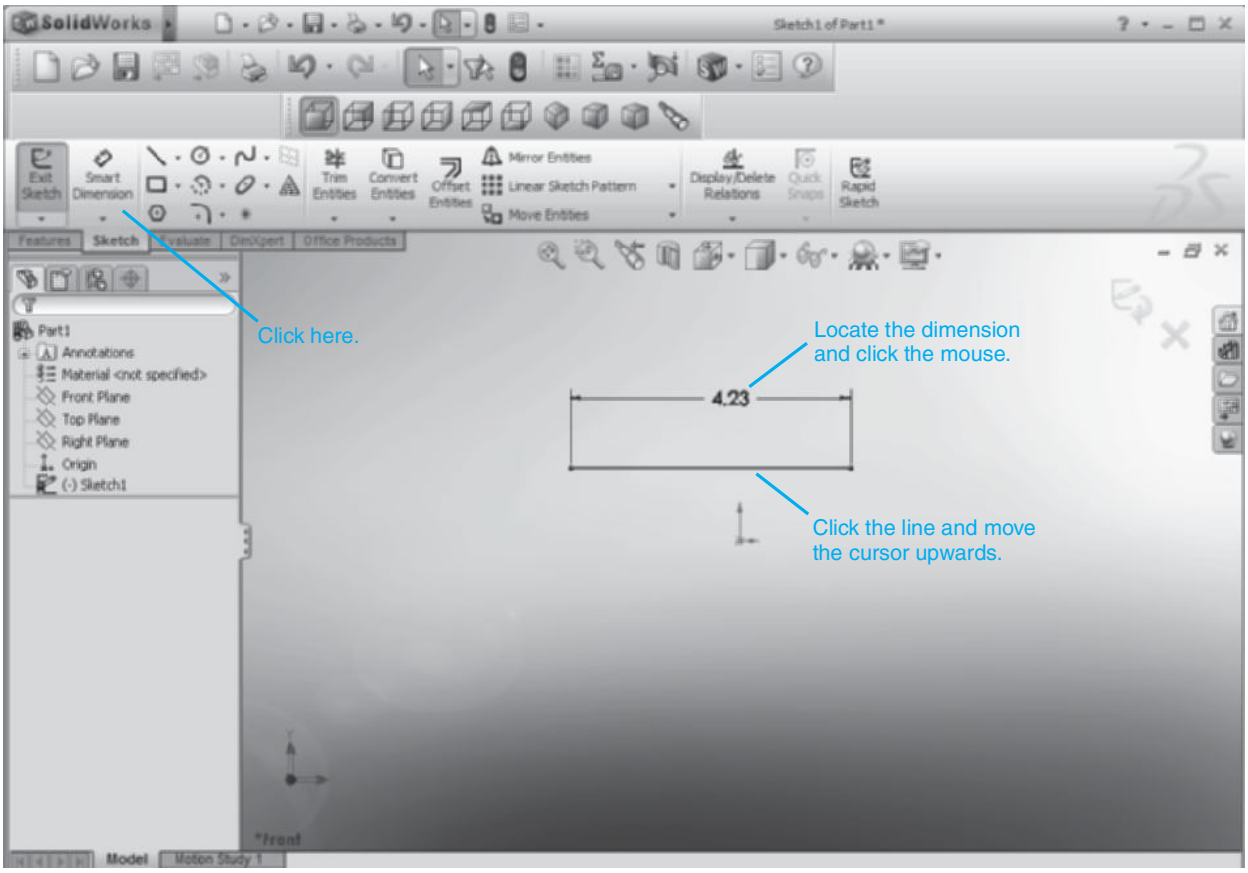


Figure 1-8

The tools on the **Command Manager** will change from **Sketch** tools to **Features** tools. See Figure 1-20. The **Features** tools are used to convert sketches into solid models. The **Features** tools will be covered in detail in Chapter 3.

2. Click the **Extrude Boss/Base** tool.

The **Features Manager** will change to display the **Extrude Properties Manager**. See Figure 1-21.

3. Define the rectangle's thickness as **0.50 in**.

TIP

As the arrows to the right of the thickness definition are clicked the thickness values change, and the thickness of the rectangle also changes in real time. You may also click and drag the arrow shown in the rectangle to change the thickness.

4. Click the OK check mark.

Note:

The rectangle can also be transitioned to a solid model by right-clicking the mouse. A list of options will appear. Click the **OK** tool. See Figure 1-22.

Figure 1-23 shows the finished rectangle. The rectangle has been used to create a 3D solid model. The shape is now a rectangular prism.

1-5 DRAWING A SHAPE WITH 90° ANGLES

Figure 1-24 shows an object that includes only right (90°) angles.

1. Start a new **Part** document as explained in Section 1–2.
2. Select **Front Plane** from the **Features Manager**.
3. Select the **Sketch** group and access the **Line** tool.
4. Sketch the shape with horizontal and vertical lines. Approximate the dimensions.

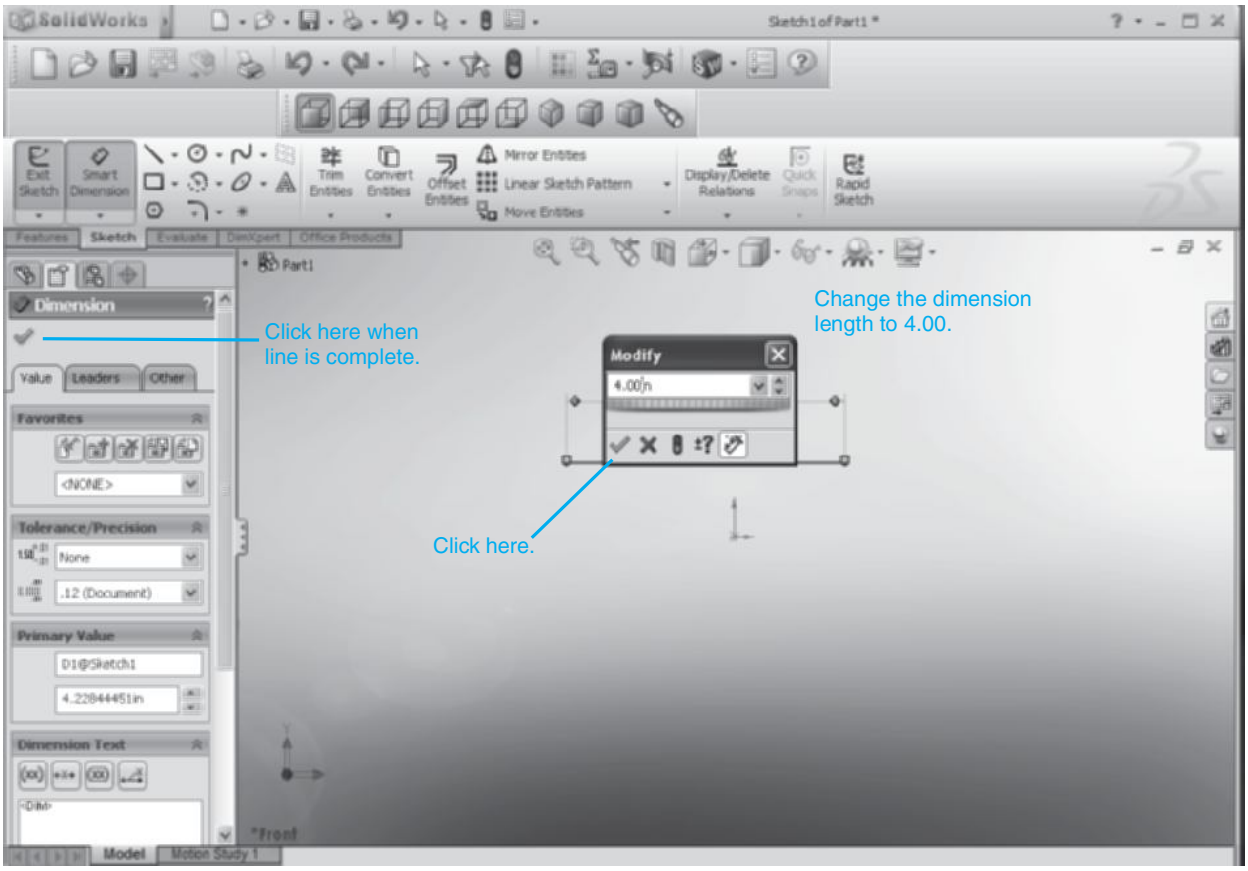


Figure 1-9

TIP

Note that as you sketch lines other lines and icons appear on the screen to tell you if you are aligned with a point or parallel or perpendicular to other lines.

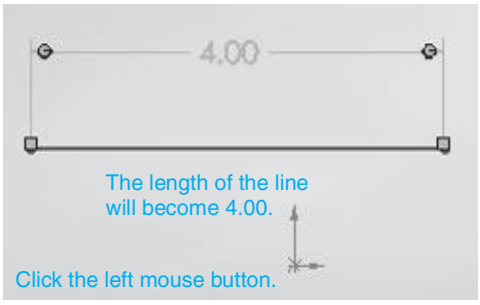


Figure 1-10

See Figure 1-25.

5. Use the **Smart Dimension** tool and size the object as shown in Figure 1-25.
6. Select the check mark in the **Line Properties Manager** and select the **Isometric** option from the selection flyout adjacent to the axis orientation icon. See Figure 1-18.
7. Right-click the mouse and click the **Select** option.
8. Click the **Features** tool, then the **Extrude Boss/Base** tool.

The screen orientation will automatically change to a three-dimensional orientation.

9. Set the object's thickness for **0.60 in**. Move the cursor into the drawing area and right-click the mouse.

See Figure 1-26.

10. Click the **OK** option in the menu that appears.

The object should look like the one shown in Figure 1-24.

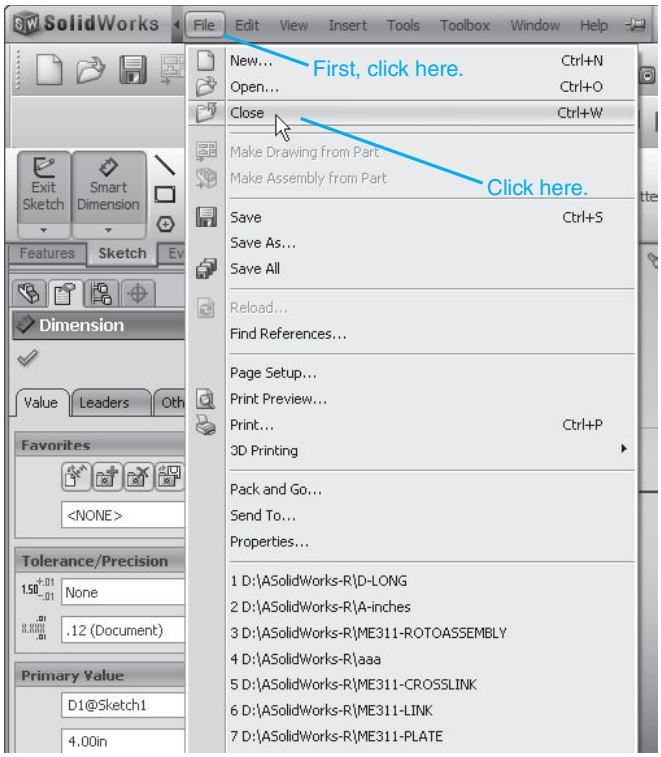


Figure 1-11

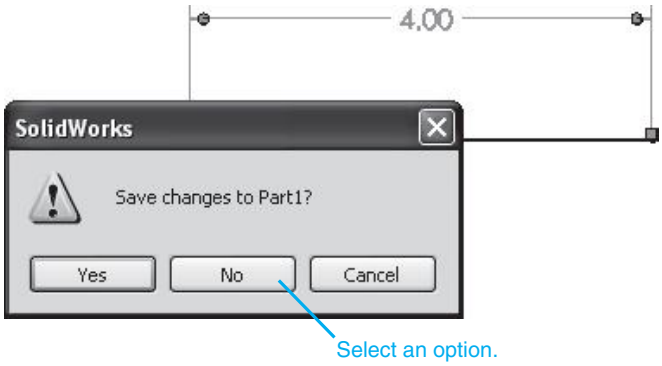


Figure 1-12

1-6 EDITING A SKETCH

It is possible to edit an existing shape using SolidWorks without resketching the object. For example, the shape created in the last section can be edited to change both the dimensions and the thickness. We will first change the depth of the cutout from 1.50 to 1.25 in. This procedure is called ***editing a sketch***. In the next section we will change the thickness of the object from 0.60 to 0.40 in. This is called ***editing a feature***. In general, changes to shapes created using the tools included in the **Sketch** group will be called *editing a sketch*, and shapes made using the tools included in the **Features** group will be called *editing a feature*. The **Features Manager** has recorded all the operations used to define the object. Click on the plus sign next to a feature to see the operations associated with that feature.

To Change the Dimensions

1. Right-click the mouse on the drawing screen. A listing of tools will appear.

   See Figure 1-27.

2. Select the **Edit Sketch** tool.
3. Double-click the 1.50 vertical dimension on the cutout.

   See Figure 1-28. The **Modify** dialog box will appear.

4. Dimension the vertical distance again using a value of **1.25** (the distance was 1.50).
5. Click the OK check mark.

   See Figures 1-29 and 1-30.

6. Double-click the second 1.50 dimension and change it to **1.25** so that the top surfaces align.
7. Click the OK check mark in the **Modify** dialog box to upgrade the dimension.

See Figures 1-31 and 1-32. These figures show the modified sketches. Click the **Exit Sketch** tool or the **Exit Sketch** icon in the triangular-shaped area in the upper right corner of the drawing screen to save the changes and upgrade the 3D feature. Figure 1-33 shows the edited object.

1-7 EDITING A FEATURE

This section will show how to change the extruded thickness of the feature from 0.60 to 0.40 in.

1. With the object on the screen, right-click the mouse button.

   A selection of tools will appear. See Figure 1-34.

2. Select the **Edit Feature** tool.

   Tools listed in the **Features** group require the **Edit Feature** tool to edit.

3. The **Extrude2 Properties Manager** will appear on the left side of the screen.

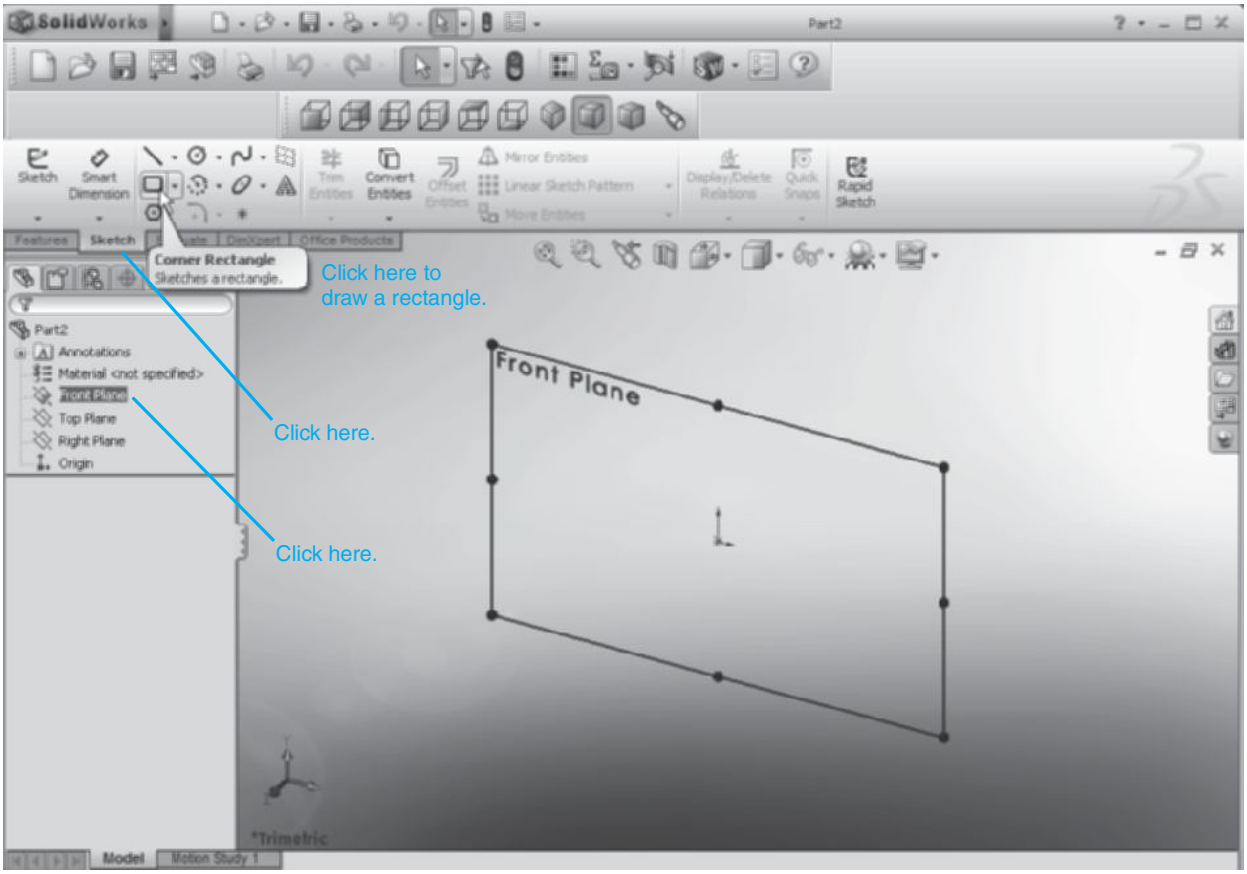


Figure 1-13

See Figure 1-35.

4. Change the thickness value from 0.60 in. to **0.40 in.**, then click the green check mark to save and update the object.

   Figure 1-36 shows the edited object.

1-8 THE CIRCLE AND SMART DIMENSION TOOLS

In this section we will create an object that includes angular corners and holes. It will be drawn in the top plane.

1. Start a new drawing using the procedures presented in Section 1-2.
2. Select the **Top Plane** orientation from the **Features Manager**.
3. Select the **Sketch** group icon on the **Command Manager**.

   See Figure 1-37.

4. Select the **Line** tool and approximately sketch the shape shown in Figure 1-38.
5. Right-click the mouse and click the **Select** tool or click the check mark in the **Line Properties Manager**.

   See Figure 1-39.

6. Select the **Smart Dimension** tool and dimension the overall width of the part to be **5.00 in**. and the top horizontal line to be **2.25 in**.

   See Figure 1-40.

7. Continue dimensioning the other lines of the object as shown.
8. Continue dimensioning the second angular measurement as shown.

To create an angular dimension, click an angular line and then click an adjacent line. Move the cursor away from the lines. The dimension will appear. Insert the dimension as shown in Figure 1-41.

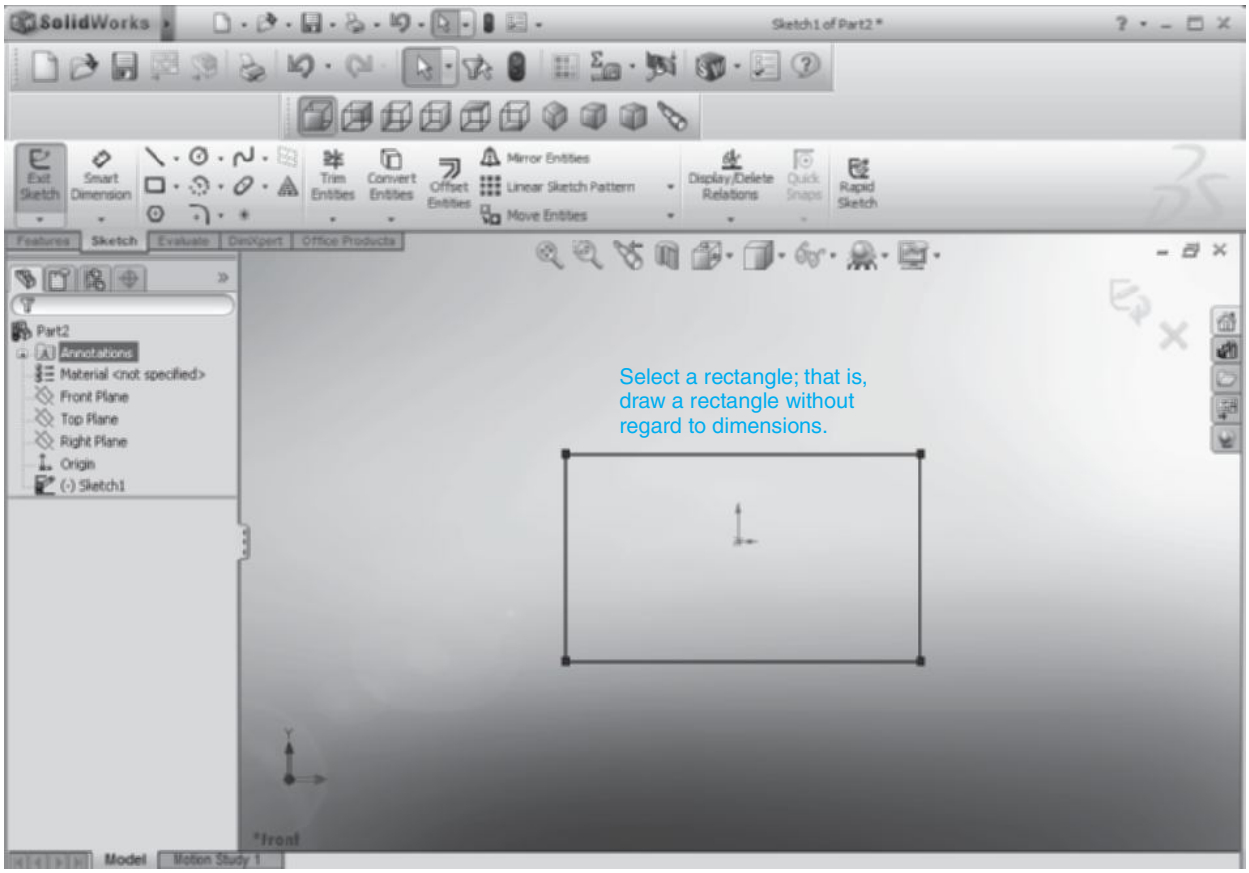


Figure 1-14

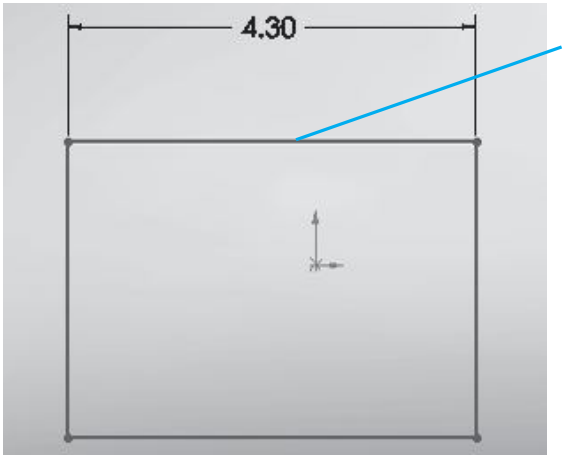


Figure 1-15

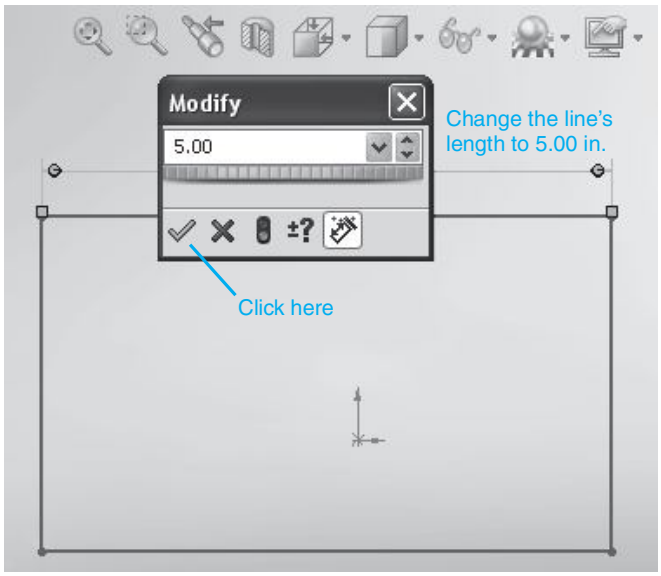


Figure 1-16

TIP

Move the cursor around the screen and note how different angular values appear.

1. Select the **Features** group on the **Command Manager** and then the **Extrude Boss/Base** tool.

The drawing's orientation will automatically change to three dimensional (trimetric).

2. Extrude the object to a thickness of **0.50 in**.
3. Click the OK check mark in the **Extrude Properties Manager** to change the figure into a solid object.

See Figure 1-42.

To Add a Hole

A hole is created in an object by first sketching a circle on a new sketch plane. The circle is then cut out of the object using the **Extrude Cut** tool, creating a hole.

Note:

Remember that a circle is a two-dimensional shape, and a hole is a three-dimensional shape.

1. Click the top surface of the object.

The surface will change colors, indicating that it has been selected.

2. Right-click the mouse and select the **Sketch** tool.

See Figure 1-43.

3. Use the **Circle** tool of the **Sketch** group on the **Command Manager** and draw a Ø0.25 circle. Use the **Smart Dimension** tool and locate the circle 0.75 from two edges as shown.

See Figures 1-44 and 1-45.

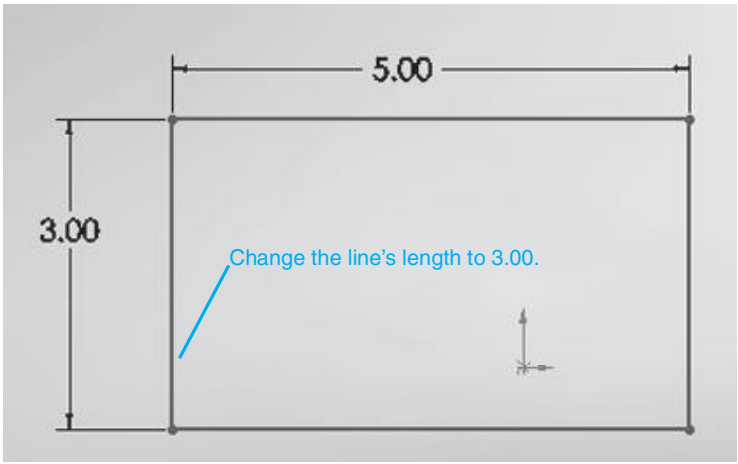


Figure 1-17

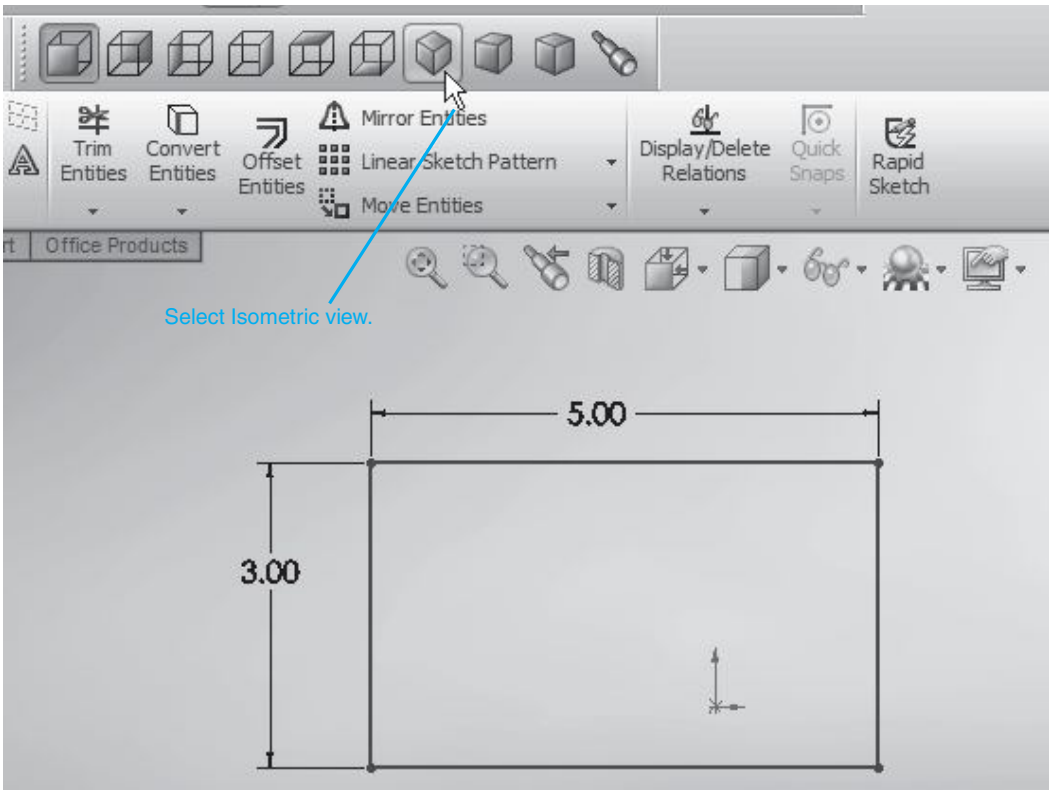


Figure 1-18

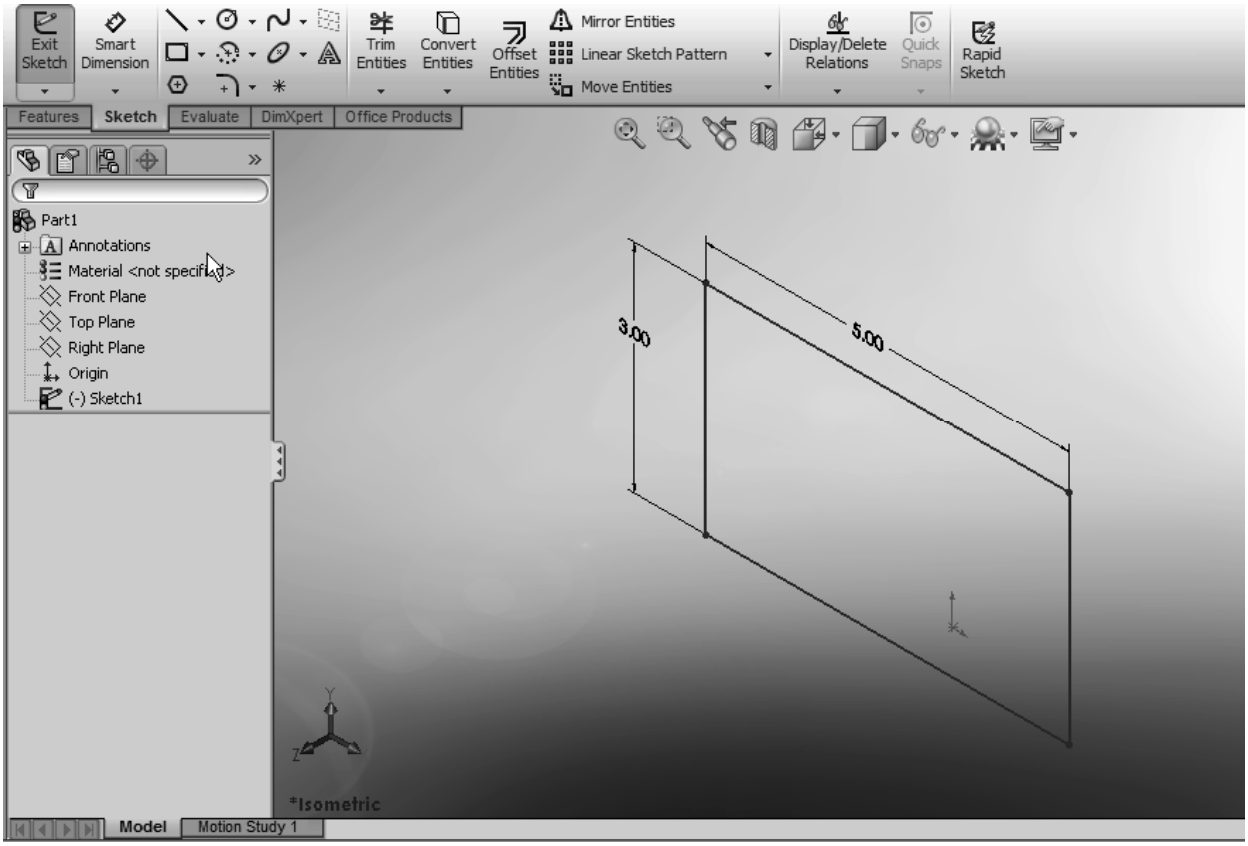


Figure 1-19

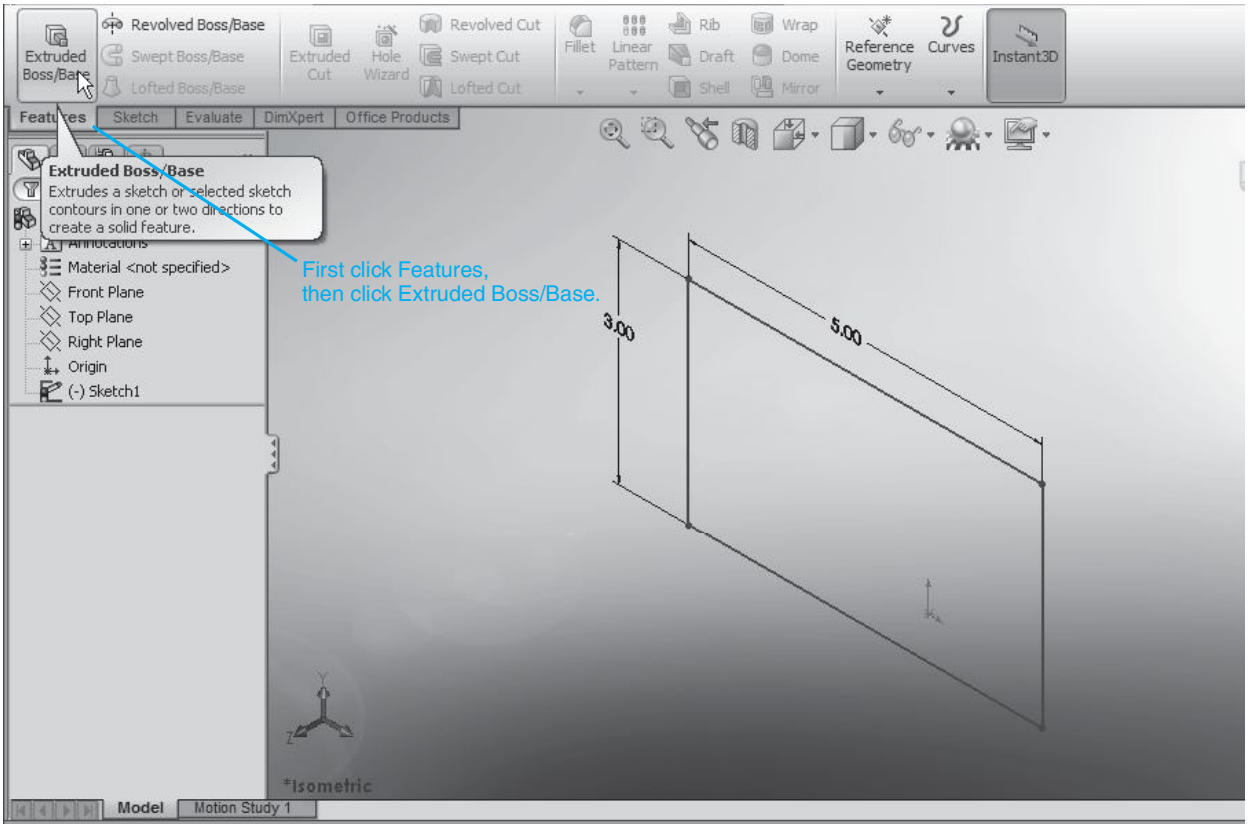


Figure 1-20

4. Click the **Features** group on the **Command Manager** and then select the **Extruded Cut** tool.

   See Figure 1-46.

5. Click the circle. A preview of the extruded cut will show on the object.
6. Click the OK check mark in the **Properties Manager**.

See Figure 1-47. Figure 1-48 shows the resulting hole in the object.

7. Click the **File** heading at the top left of the screen.

   A series of tools will cascade down.

8. Click the **Save As** tool.
9. Define the drawing's file name and click **Save**.

1-9 SETTING UNITS ON THE DOCUMENT OPTIONS

The default settings for drawing units may be modified using the **Document Properties** dialog box. In this section we will define the drawing units as millimeters.

1. Start a new **Part** document.
2. Click **Tools** on the menu bar located at the top of the screen.

   A series of tools will cascade down. See Figure 1-49.

3. Click **Options**.

The **Document Properties - Units** dialog box will appear. See Figure 1-50.

4. Click the **Document Properties** tab.

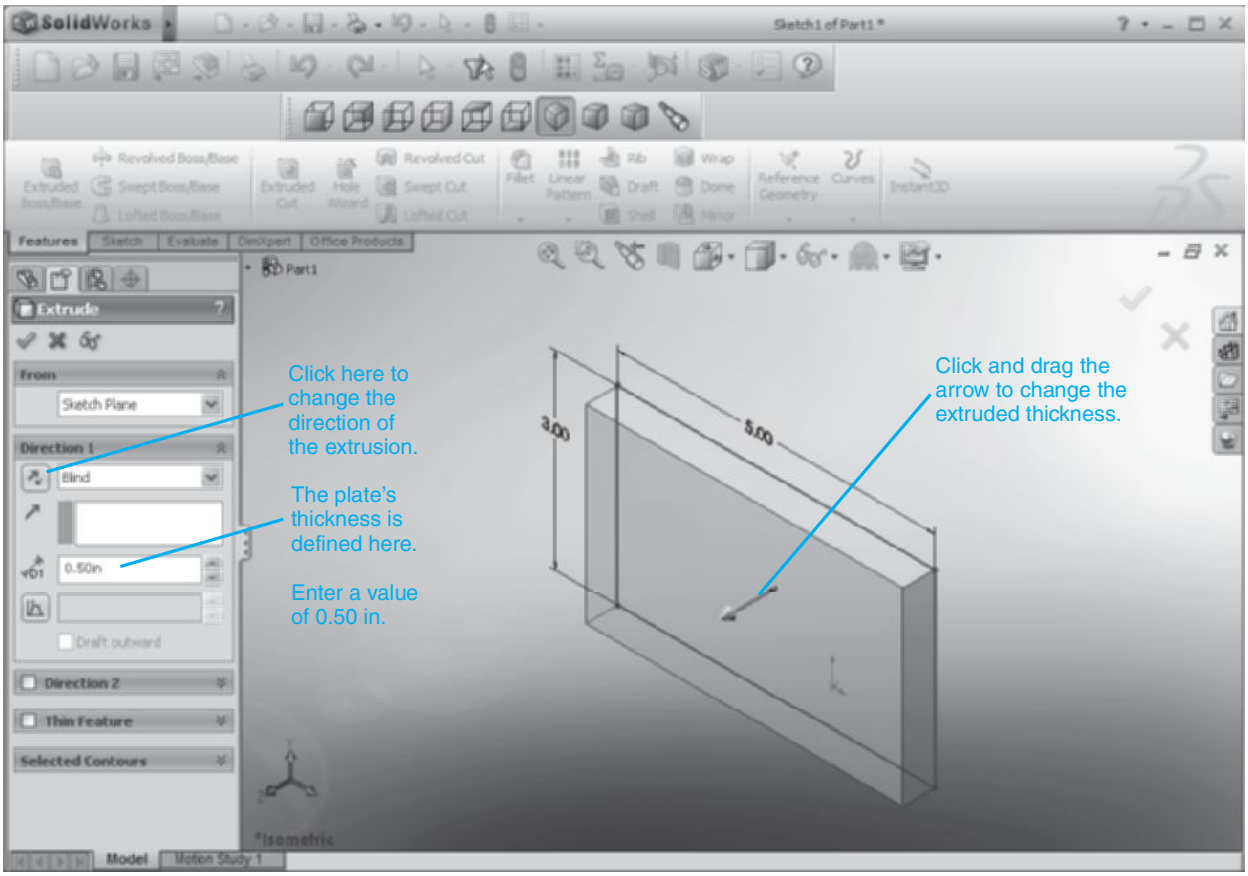


Figure 1-21

5. Click **Units** in the left column.

   The **Units** dialog box will appear.

6. Click the **MMGS (millimeter, gram, second)** tool listed in the **Unit system** box.
7. Click **OK**.

   The drawing units are now calibrated to millimeters.

8. Return to the drawing screen and proceed with the following section.

TIP

In SolidWorks the positive direction is the counter-clockwise direction.

1-10 THE CENTERPOINT ARC AND TANGENT ARC TOOLS

To Use the Tangent Arc Tool

Use the **Part** document started in the previous section. Select the top plane. Select the **Sketch** group from the **Command Manager**.

1. Select the **Line** tool and draw two lines **80 mm** long, **50 mm** apart, and parallel to each other. Assure that the line's starting points are vertically aligned.

See Figure 1-51. Use the **Smart Dimension** tool to locate and size the lines.

2. Click the **Tangent Arc** tool.
3. Click the left end of the upper 80-mm line.

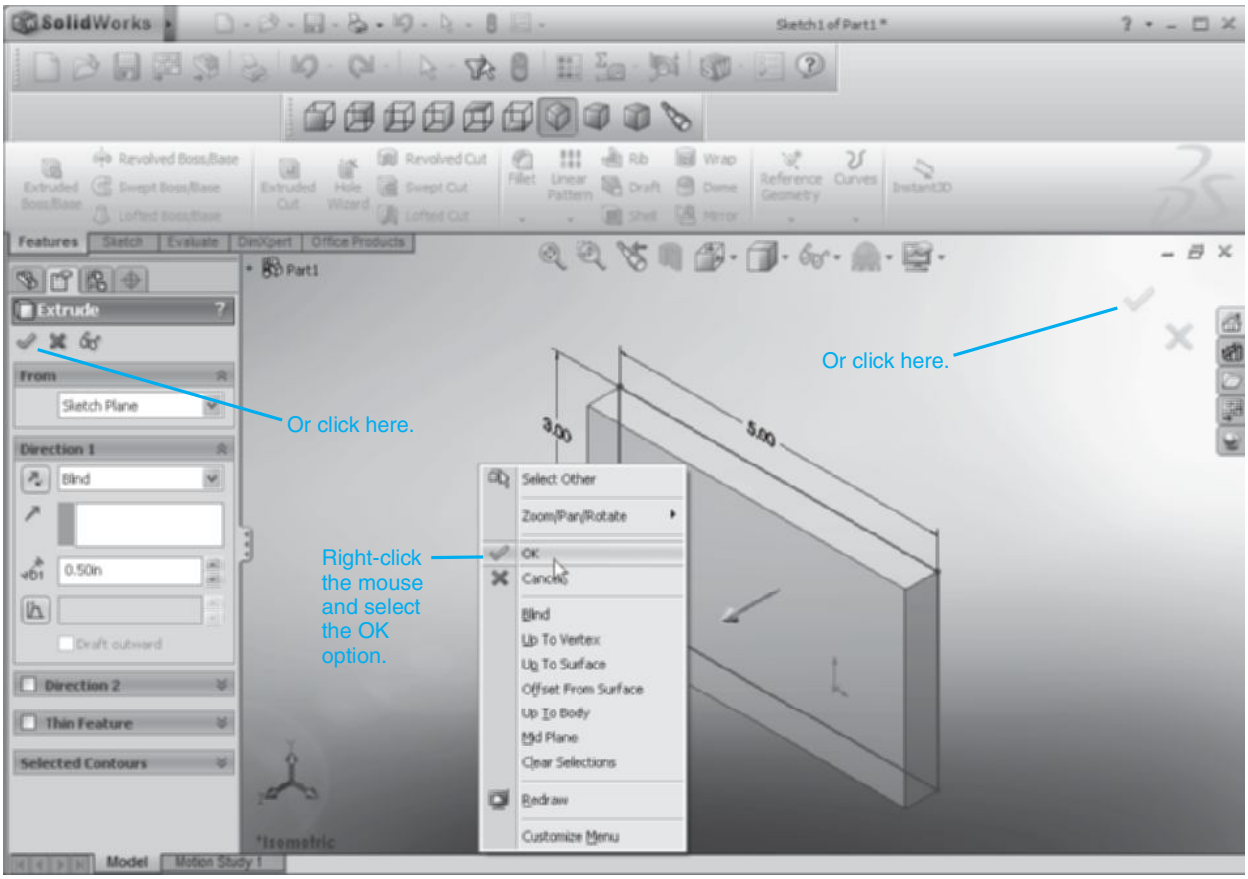
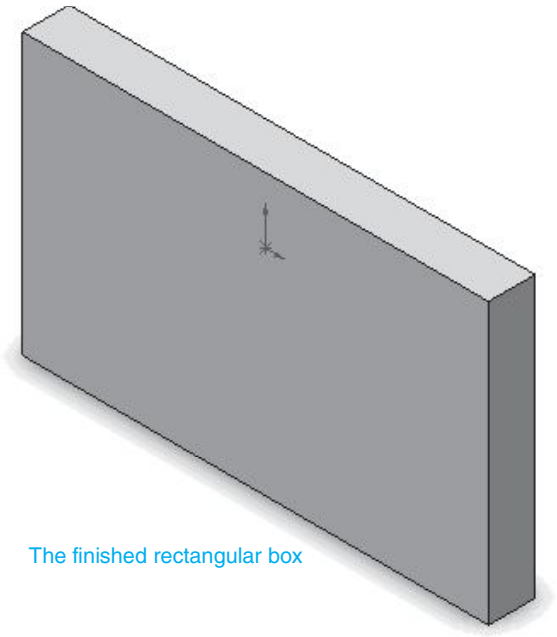
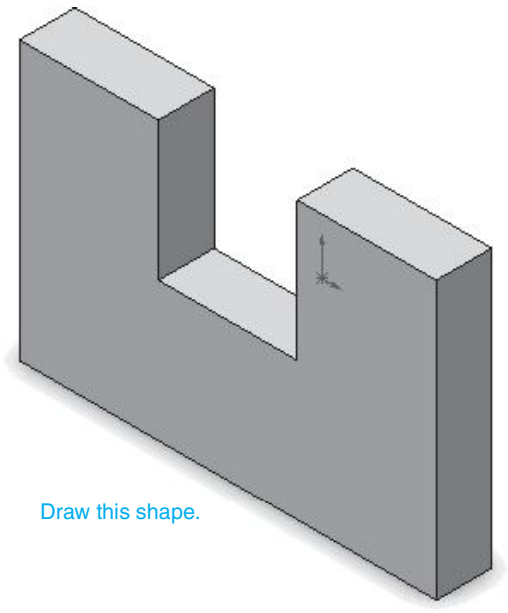


Figure 1-22



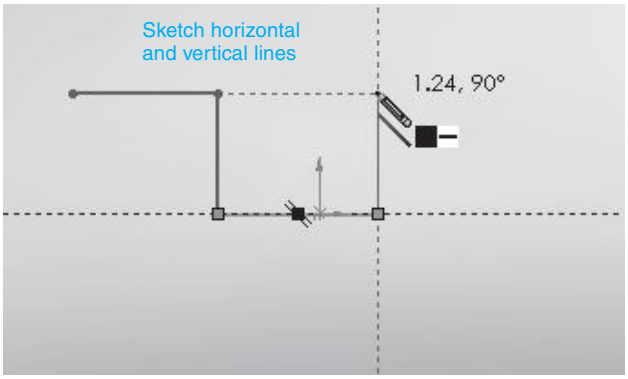
The finished rectangular box

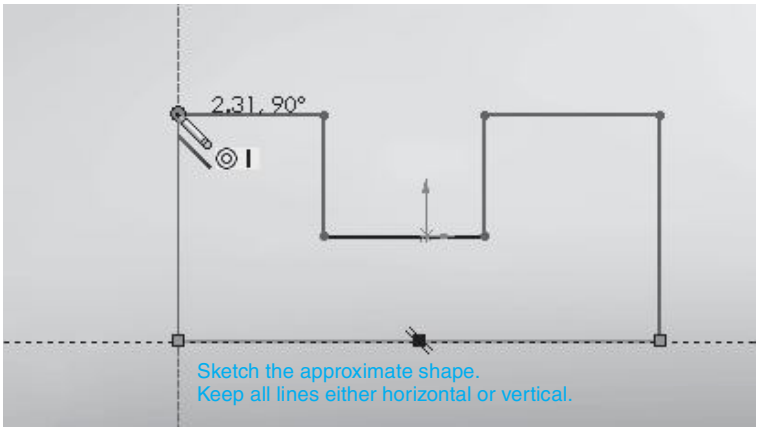
Figure 1-23

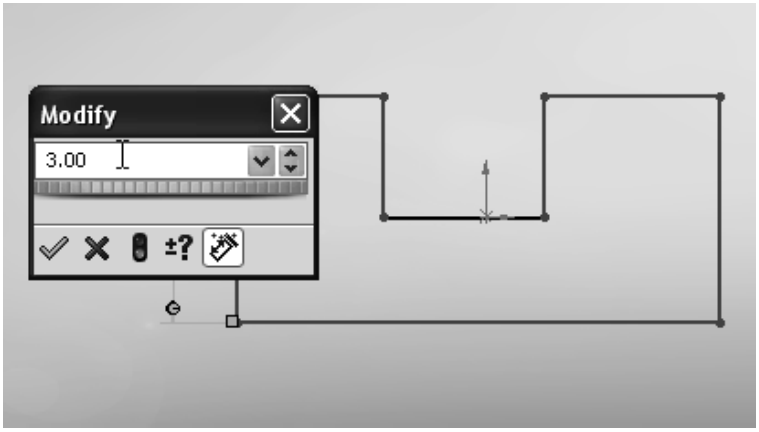


Draw this shape.

Figure 1-24


Sketch horizontal
and vertical lines
1.24, 90°


2.31, 90°
Sketch the approximate shape.
Keep all lines either horizontal or vertical.


Modify
3.00

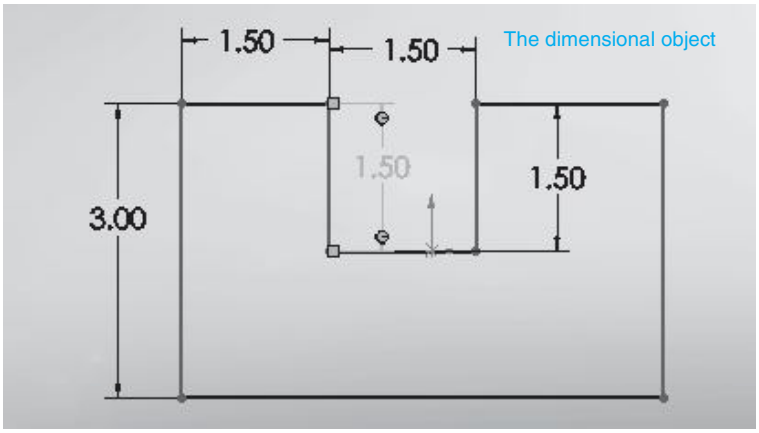

1.50
1.50
The dimensional object
1.50
1.50
3.00

Figure 1-25

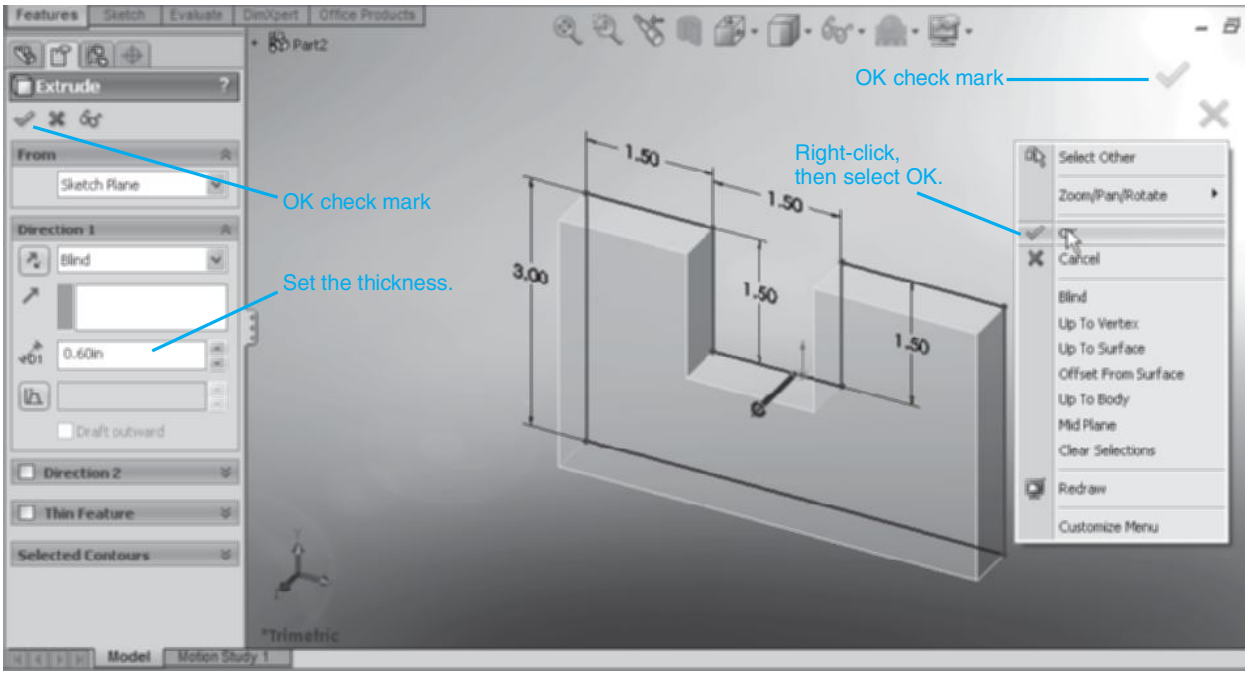


Figure 1-26

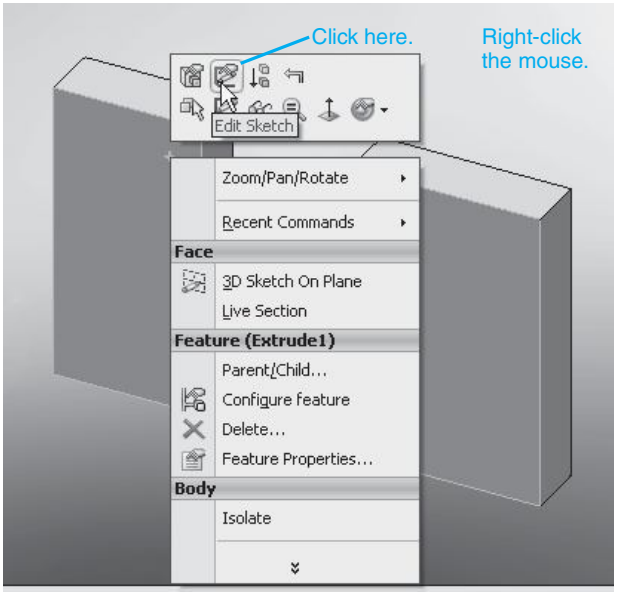


Figure 1-27

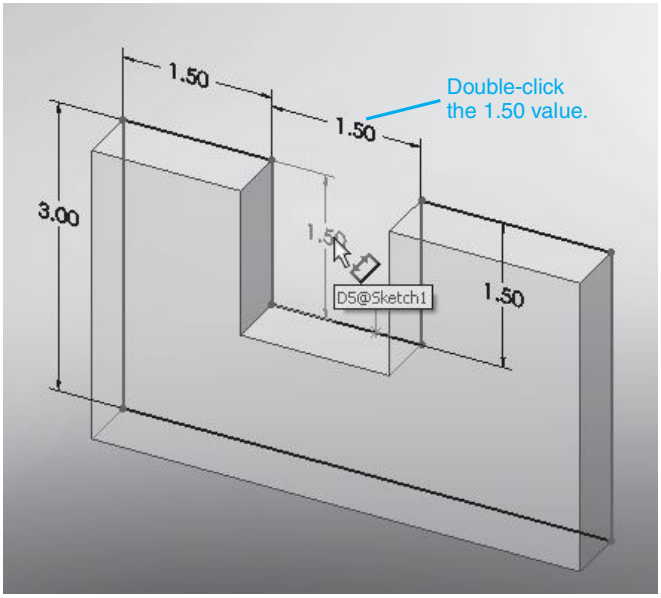


Figure 1-28

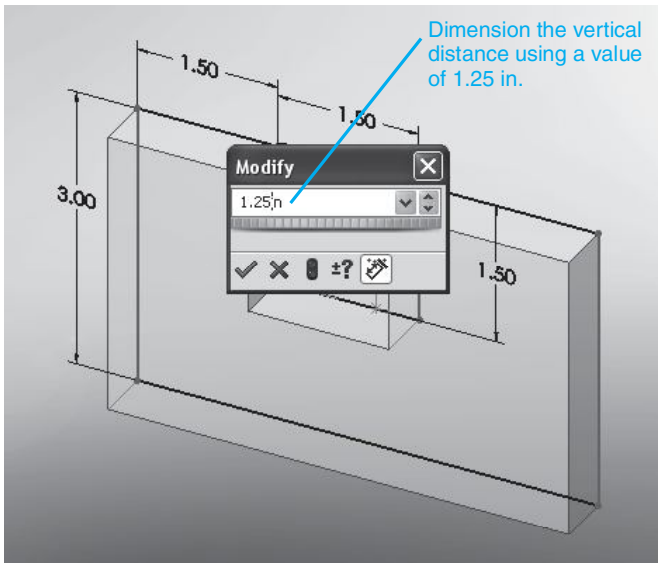


Figure 1-29

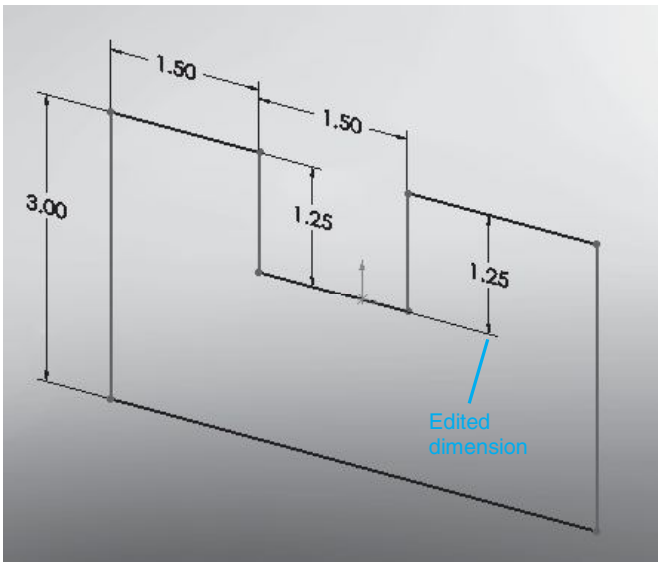


Figure 1-31

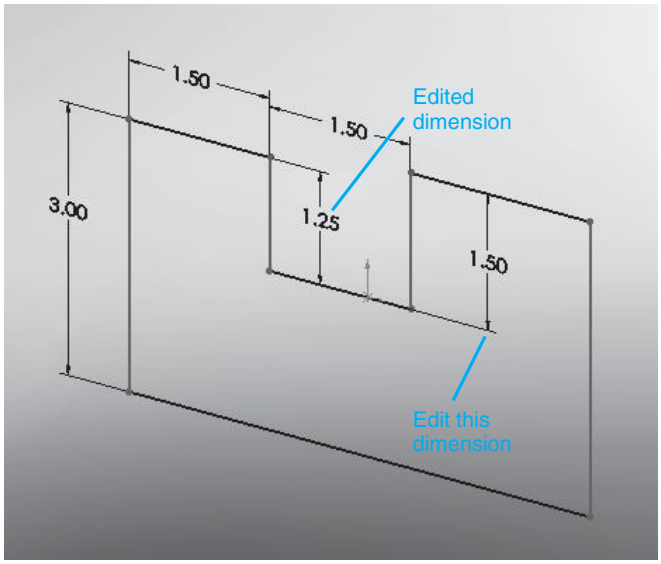


Figure 1-30

The ends of the lines are defined by colored circles. Circles will change to red and grow larger when they are selected. See Figure 1-52.

4. Move the cursor along the approximate path of the arc down to the left endpoint of the lower 80-mm line. Click the endpoint.
5. Right-click the mouse and click the **Select** option or click the check mark in the **Arc Properties Manager**.

The Centerpoint Arc Tool

1. Click the **Centerpoint Arc** tool.
2. Locate the center point for the arc.

See Figure 1-53. The center point for the arc can be located by moving the cursor to the approximate midpoint between the ends of the two parallel lines. Dotted lines will project from the left arc's center point and the line's endpoint when the cursor is aligned with the points.

3. Click the center point.
4. Move the cursor to the endpoint of the lower 80-mm line and click the endpoint.
5. Move the cursor upward and click the right endpoint of the upper 80-mm line.

See Figure 1-54.

6. Right-click the mouse and click the **Select** option or click the check mark in the **Arc Properties Manager**.

1-11 EXTRUDING AN OBJECT

1. Click the arrow to the right of the axis orientation icon at the lower left of the screen.
2. Click the **Features** group on the **Command Manager** and select the **Extruded Boss/Base** tool.

See Figure 1-55.

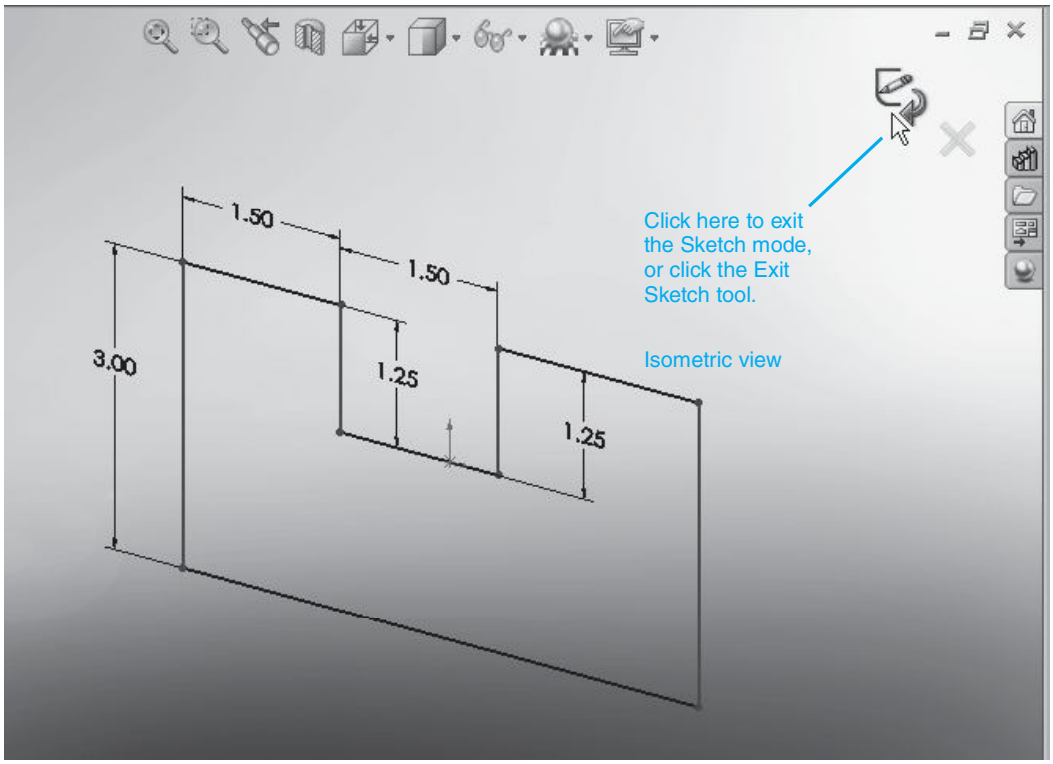


Figure 1-32

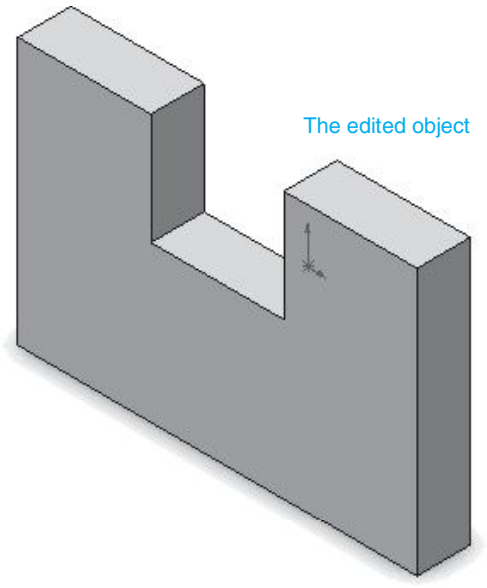


Figure 1-33

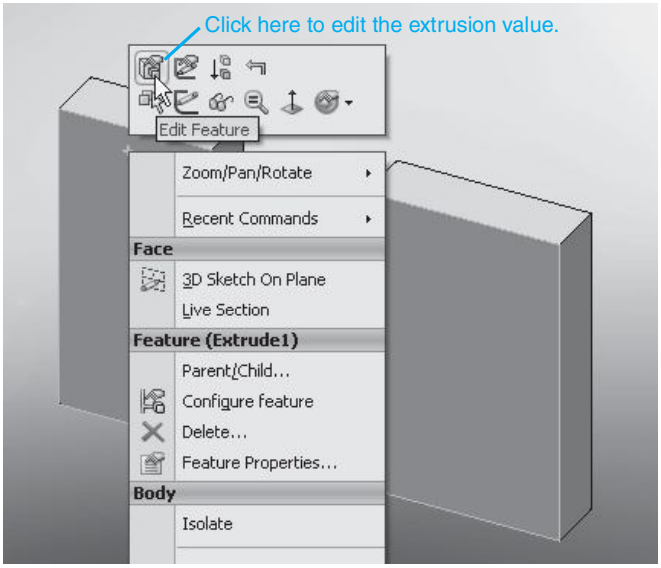


Figure 1-34

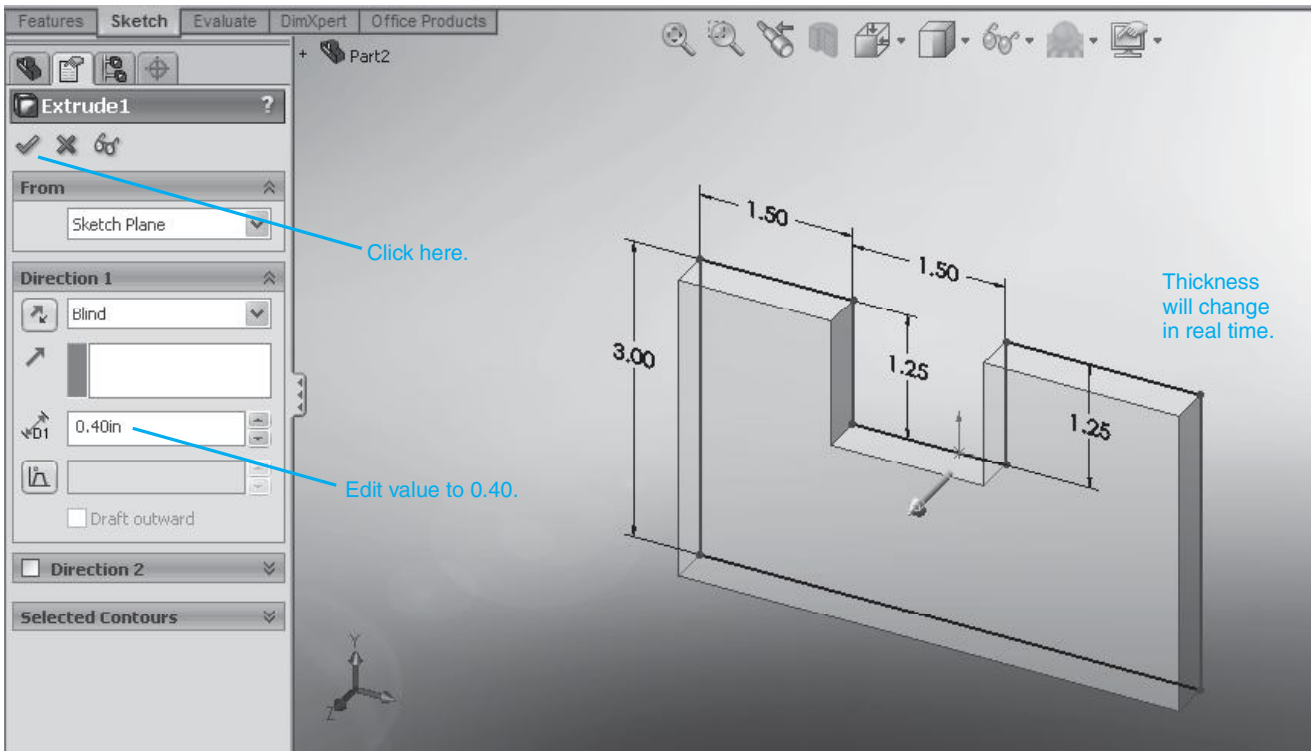


Figure 1-35

3. Set the depth of the extrusion as **20 mm**.
4. Click the OK check mark in the **Extrude Properties Manager** to complete the extrusion.

See Figures 1-56 and 1-57.

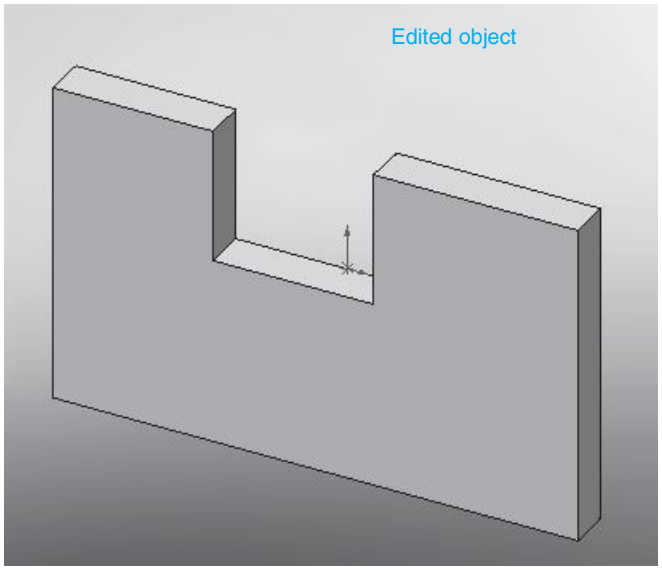


Figure 1-36

To Add Holes to the Object

1. Click the top surface of the object, then right-click the mouse.
2. Select the **Sketch** tool.

See Figure 1-58. This command allows you to create 2D shapes on the top surface.

3. Use the **Circle** tool and sketch a circle. Locate the center point of the circle on the center point of the arc used to define the left end of the object. Size the circle to **Ø20.0 mm** using the **Circle Properties Manager**.

See Figure 1-59.

TIP

The center point for the arc can be found by moving the cursor in the general area of the arc's center point. A circle will appear with a center point when the cursor is located directly over the arc's center point.

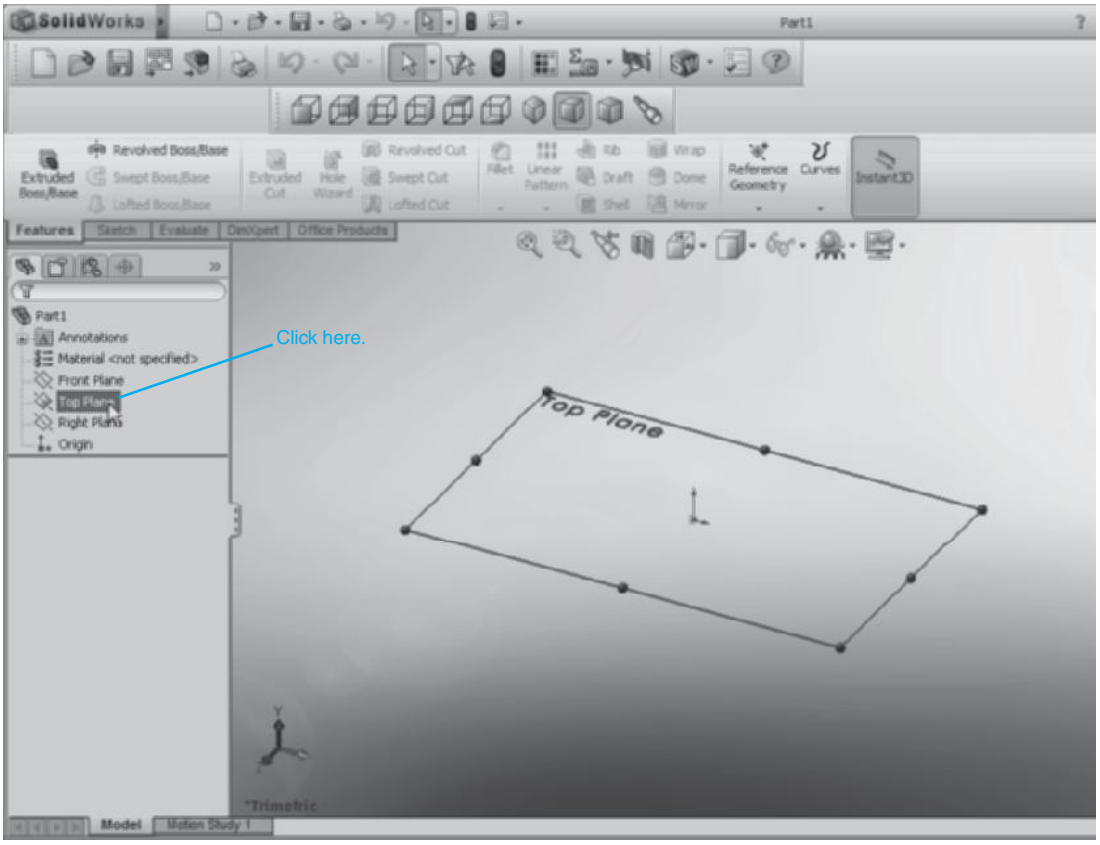


Figure 1-37

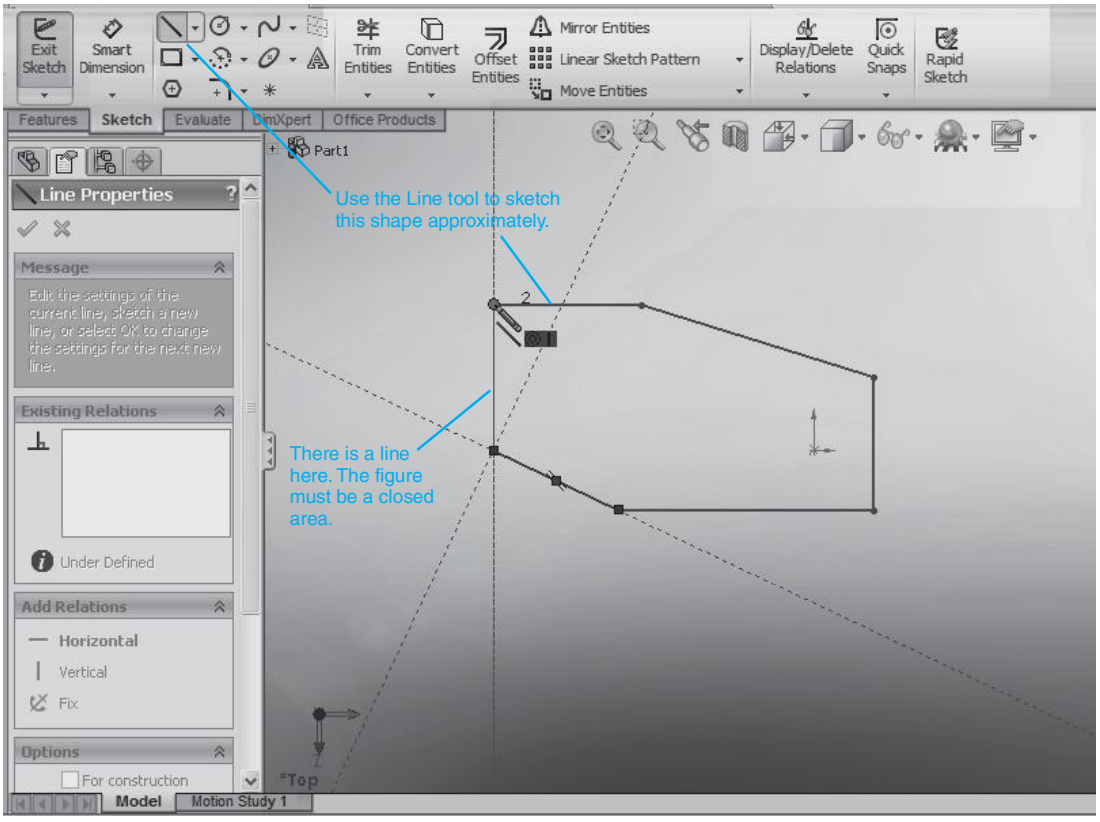


Figure 1-38

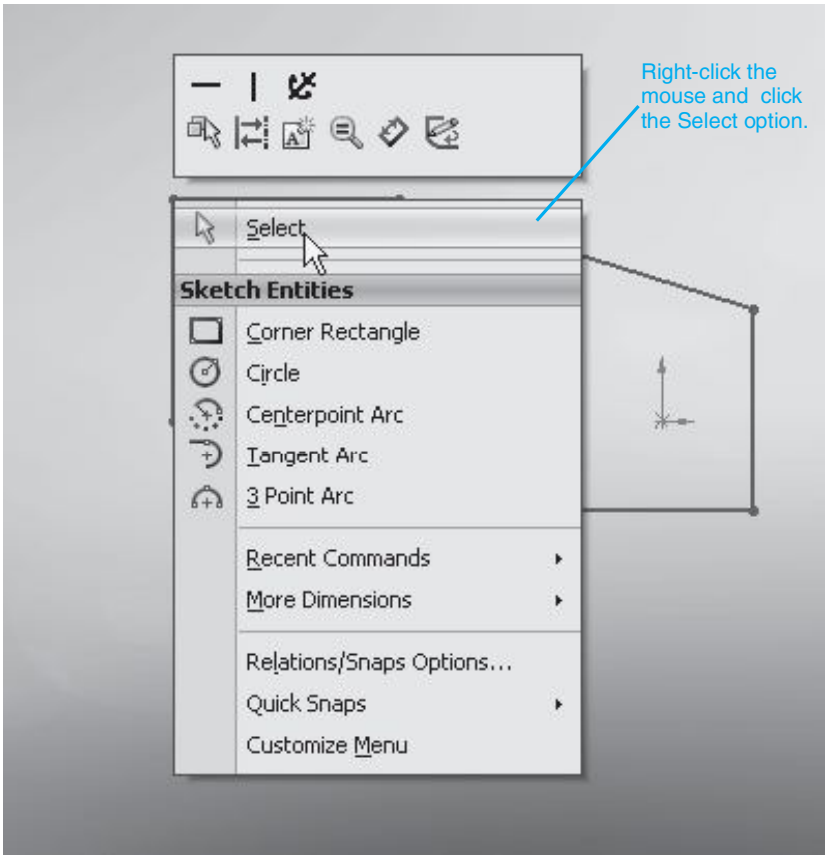


Figure 1-39

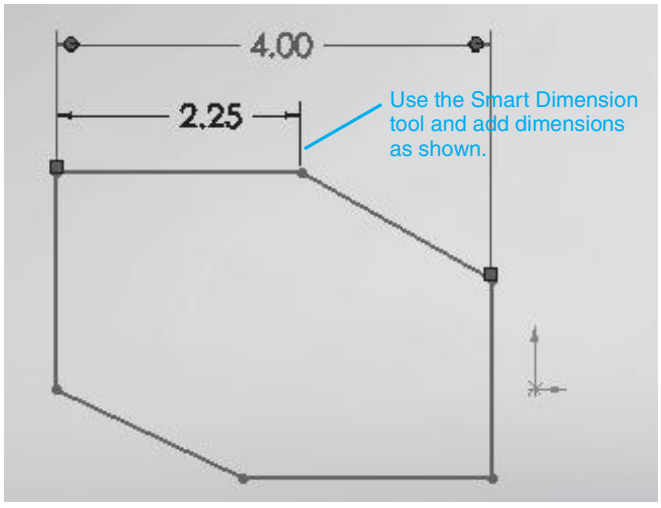


Figure 1-40

4. Locate the center point on the right end of the object for a second Ø20.0-mm circle and draw a circle.

The center point for the second circle can be located using the first center point and the endpoint of the lower 80-mm line. See Figure 1-60.

To Create Holes

1. Click the **Features** group, then the **Extruded Cut** tool.
2. Cut out the circles to form holes by clicking the check mark in the **Cut-Extrude Properties Manager**.

See Figure 1-61.

To Create a Slot

1. Use the **Sketch** tool to create a sketching plane on the top surface of the object.
2. Select the **Rectangle** tool.
3. Sketch a rectangle on the top surface of the object.

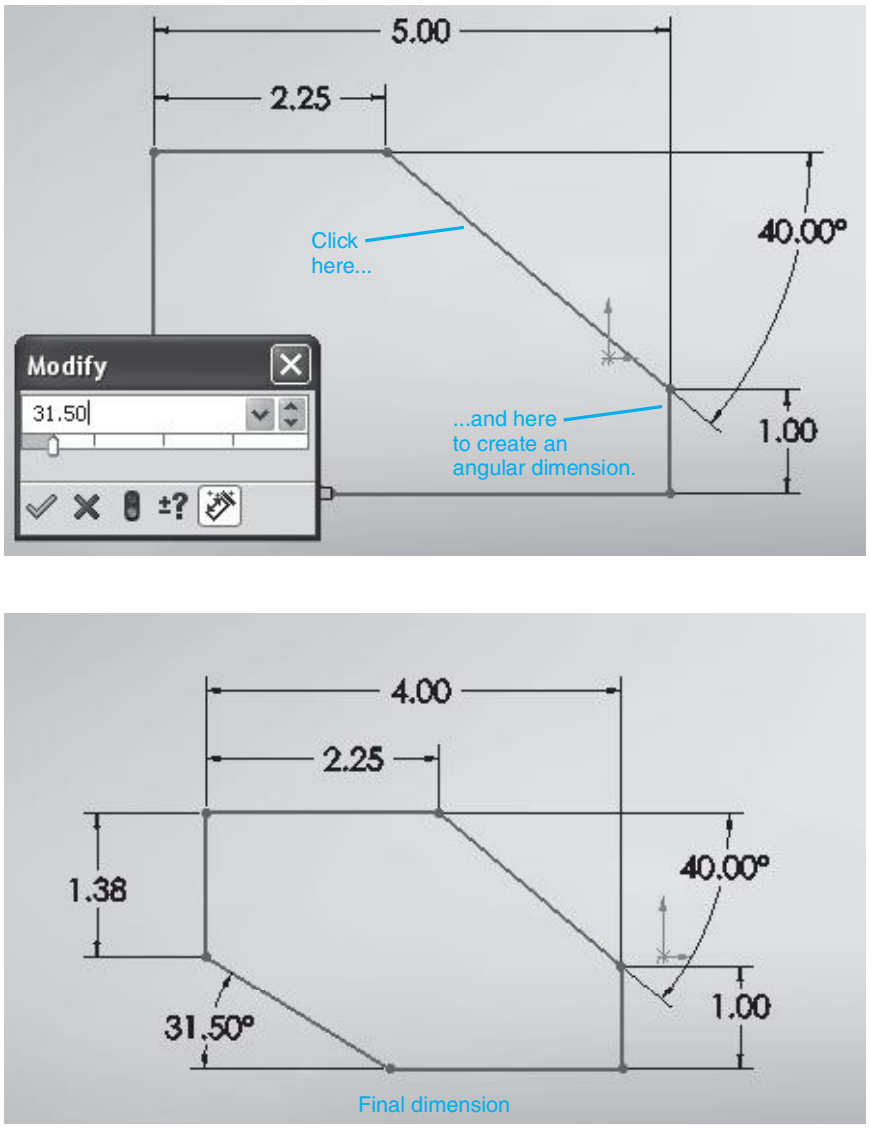


Figure 1-41

TIP

The edges of the part will highlight when touched by the cursor. Make sure that the rectangle goes from one edge of the part to the other.

See Figure 1-62.

4. Use the **Smart Dimension** tool to size and locate the rectangle as indicated in Figure 1-63.

The 30 locating dimension is taken from the edge of the slot to the end of the 80 edge line.

See Figure 1-63.

5. Click the **Features** group and select the **Extruded Cut** tool.
6. Set the slot depth for **10.00mm**.
7. Click the check mark in the **Cut-Extrude Properties Manager**.
8. **Save** or **Exit** the drawing.

Figure 1-64 shows the finished object.

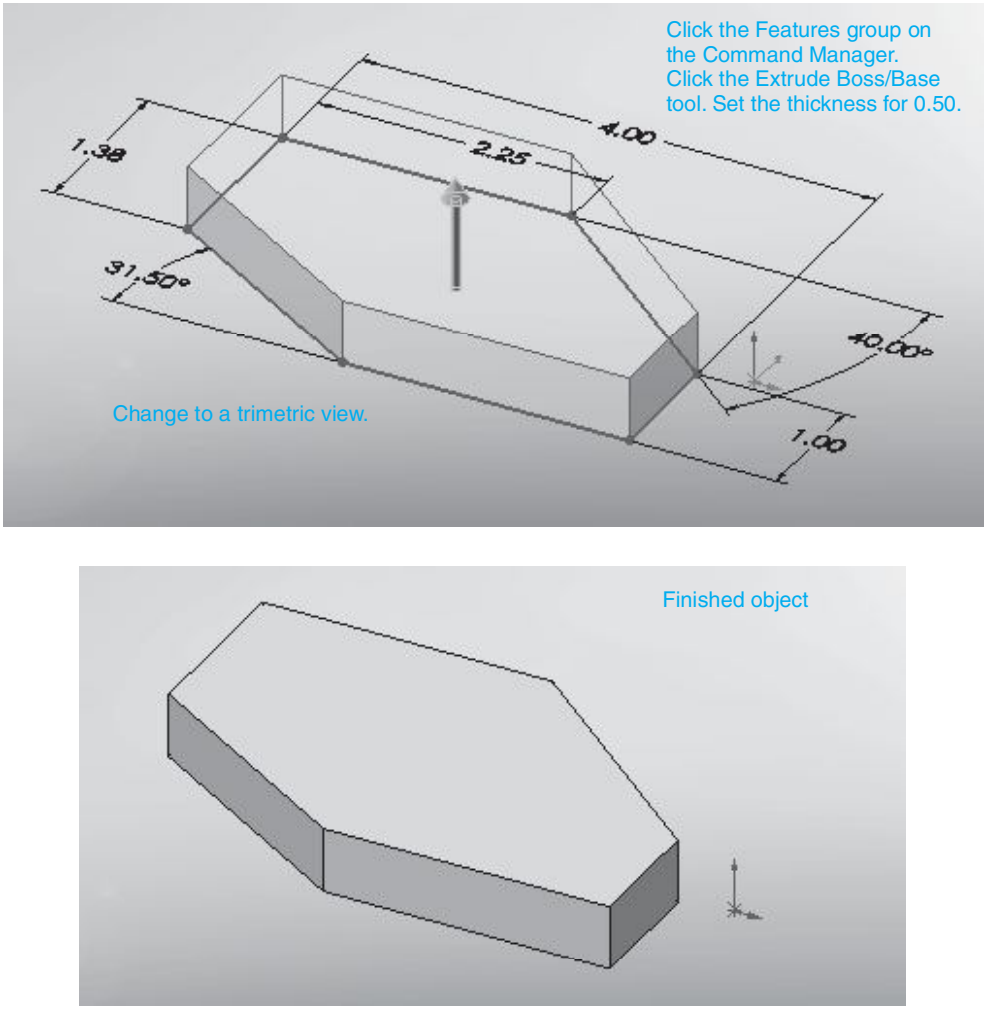


Figure 1-42

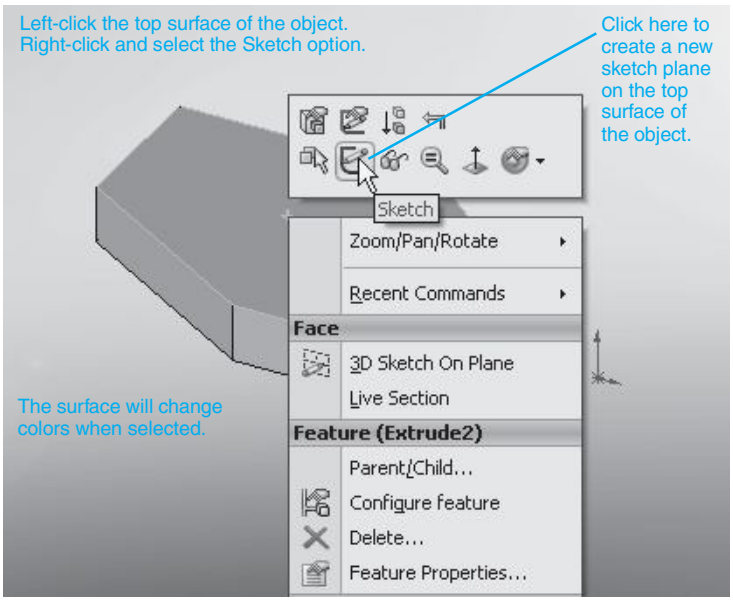


Figure 1-43

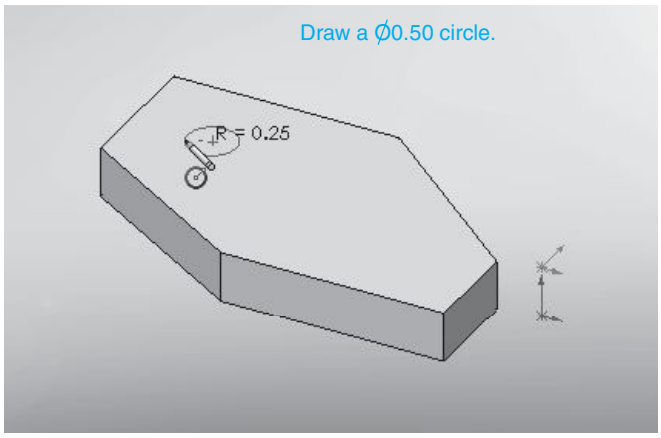


Figure 1-44

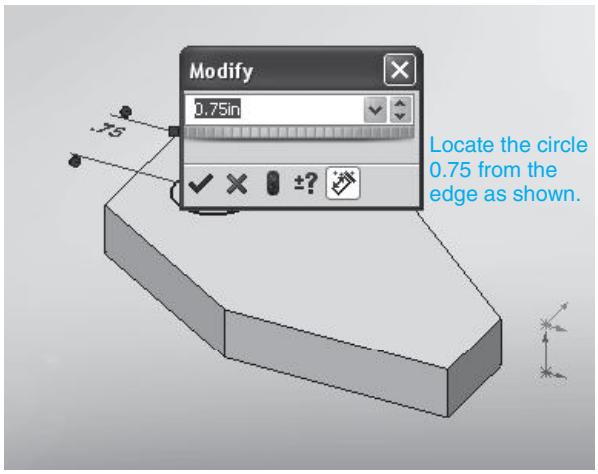


Figure 1-45

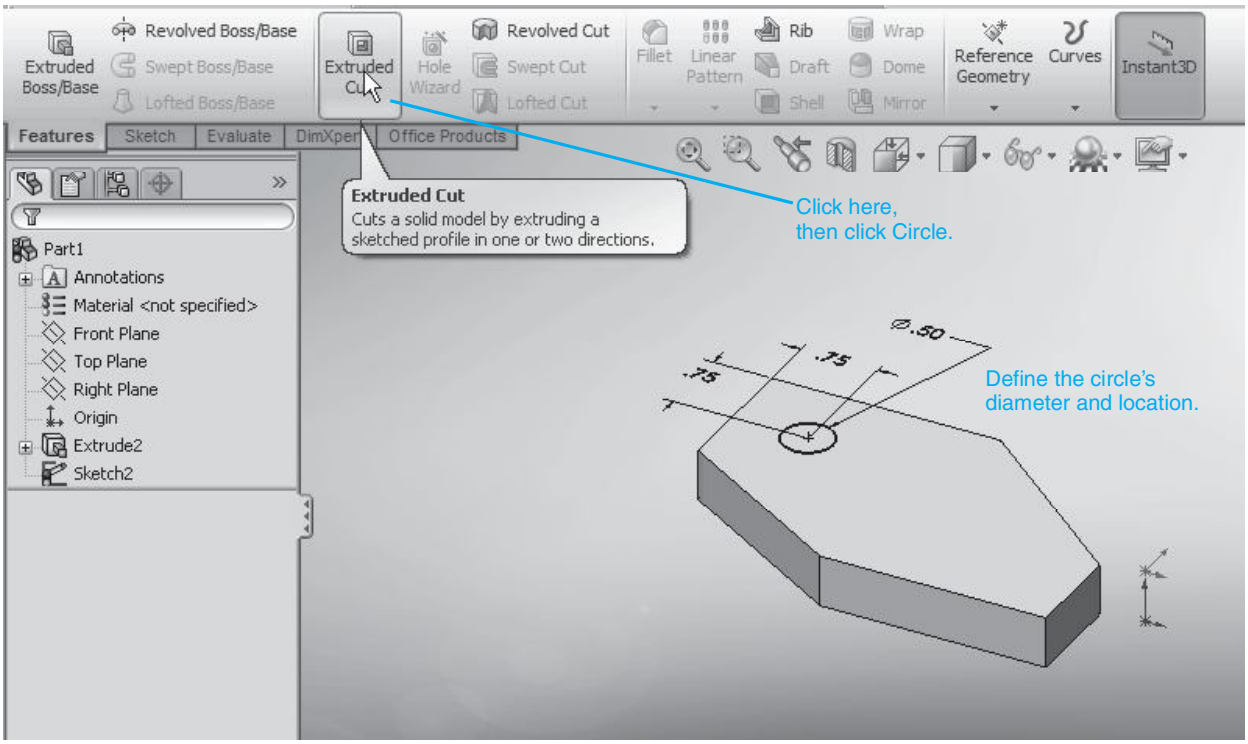


Figure 1-46

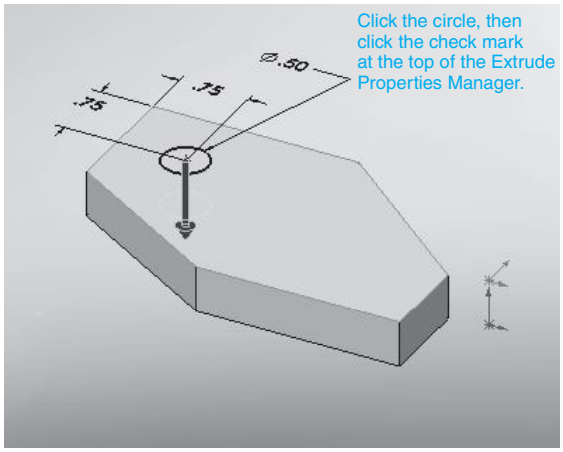


Figure 1-47

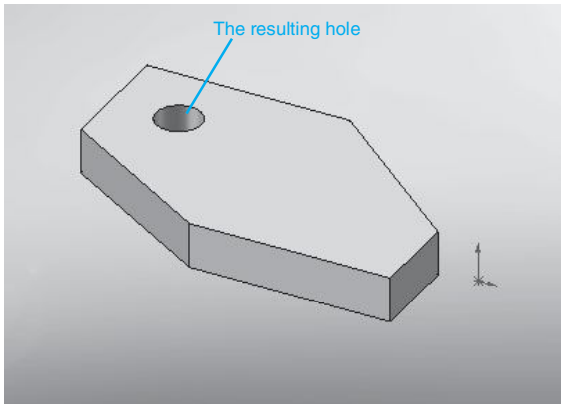


Figure 1-48

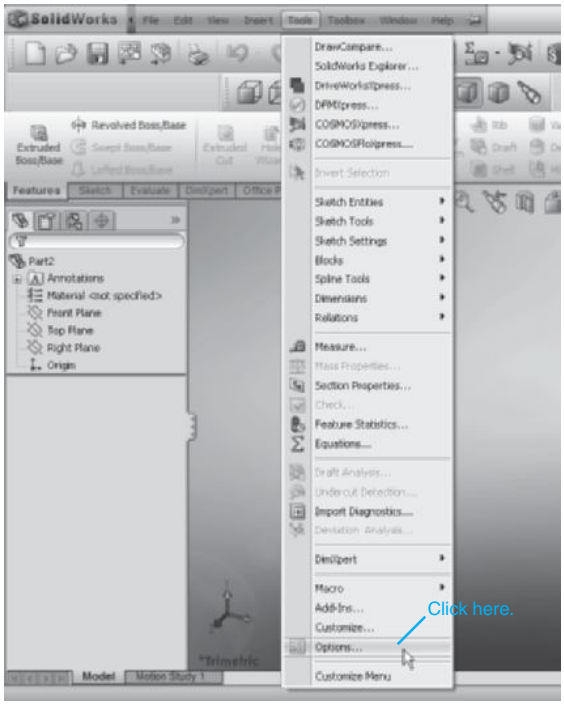


Figure 1-49

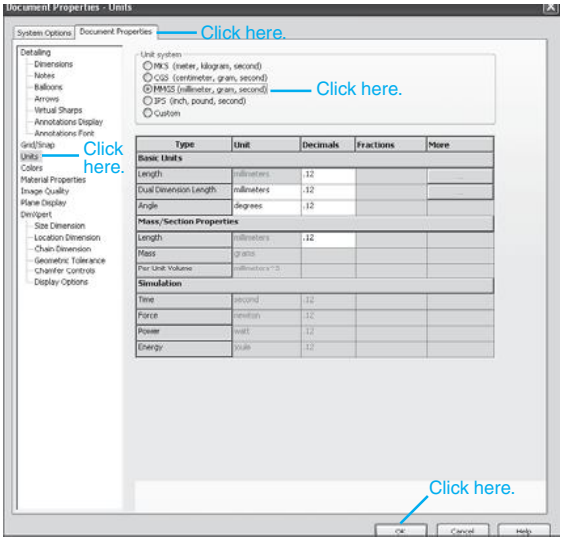


Figure 1-50

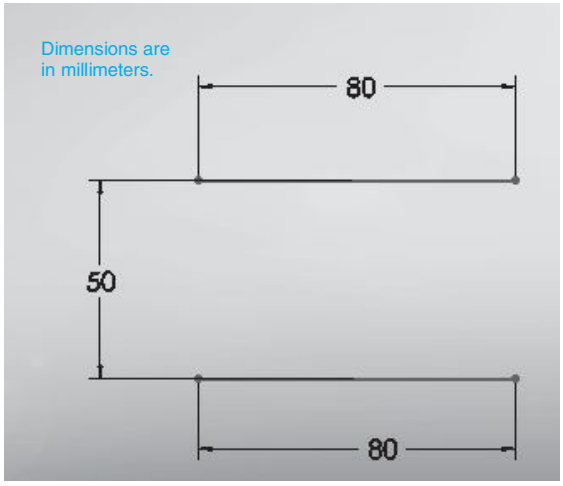


Figure 1-51

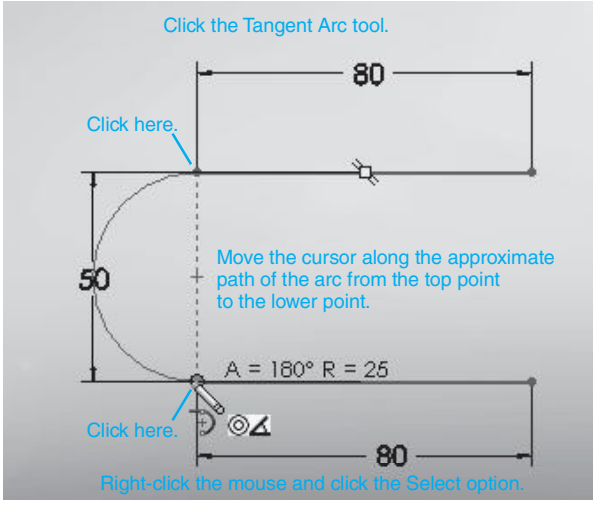


Figure 1-52

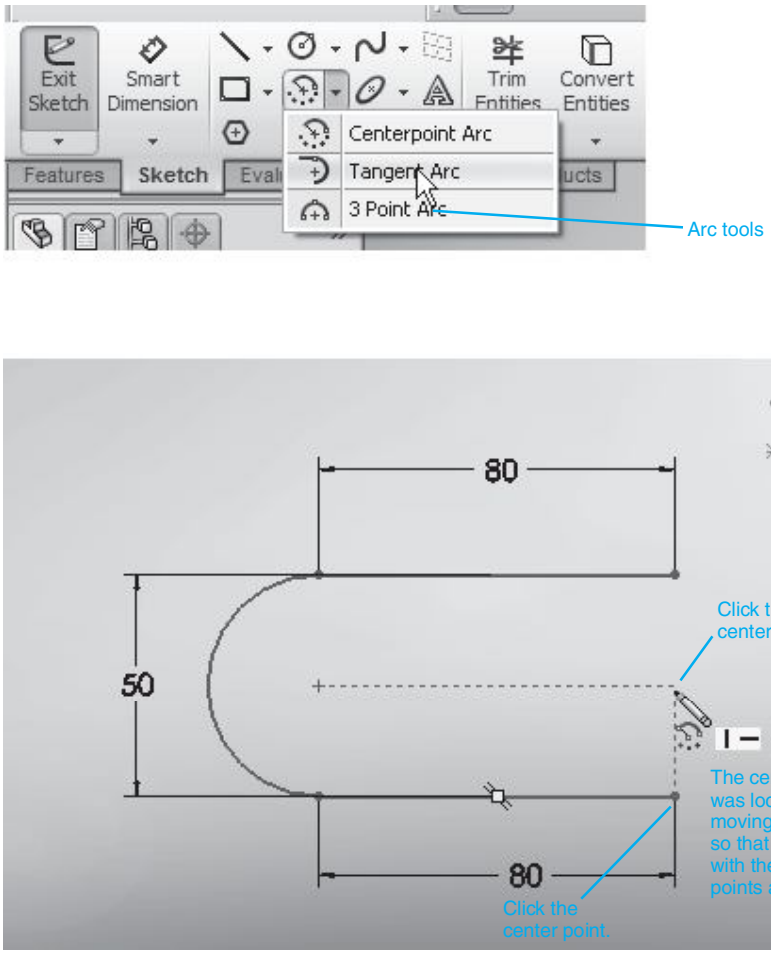


Figure 1-53

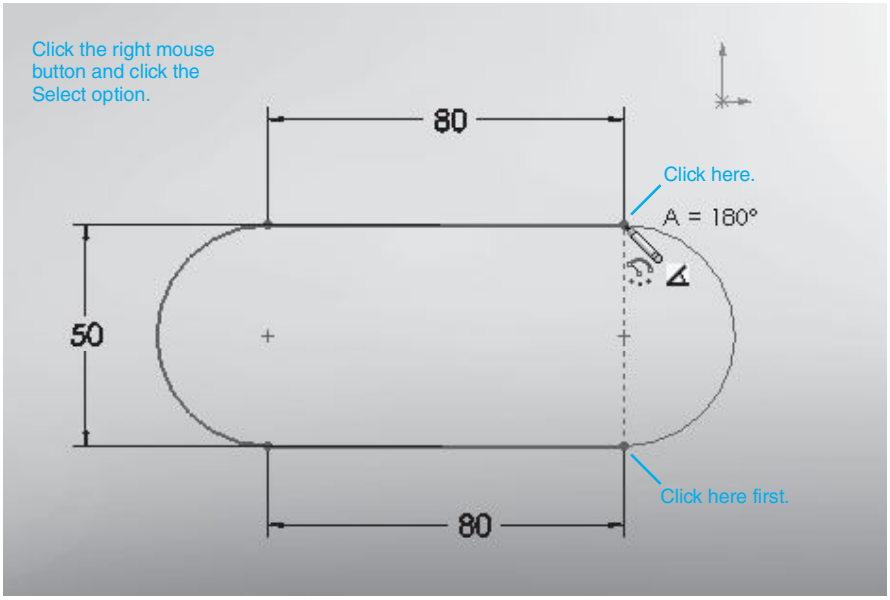


Figure 1-54

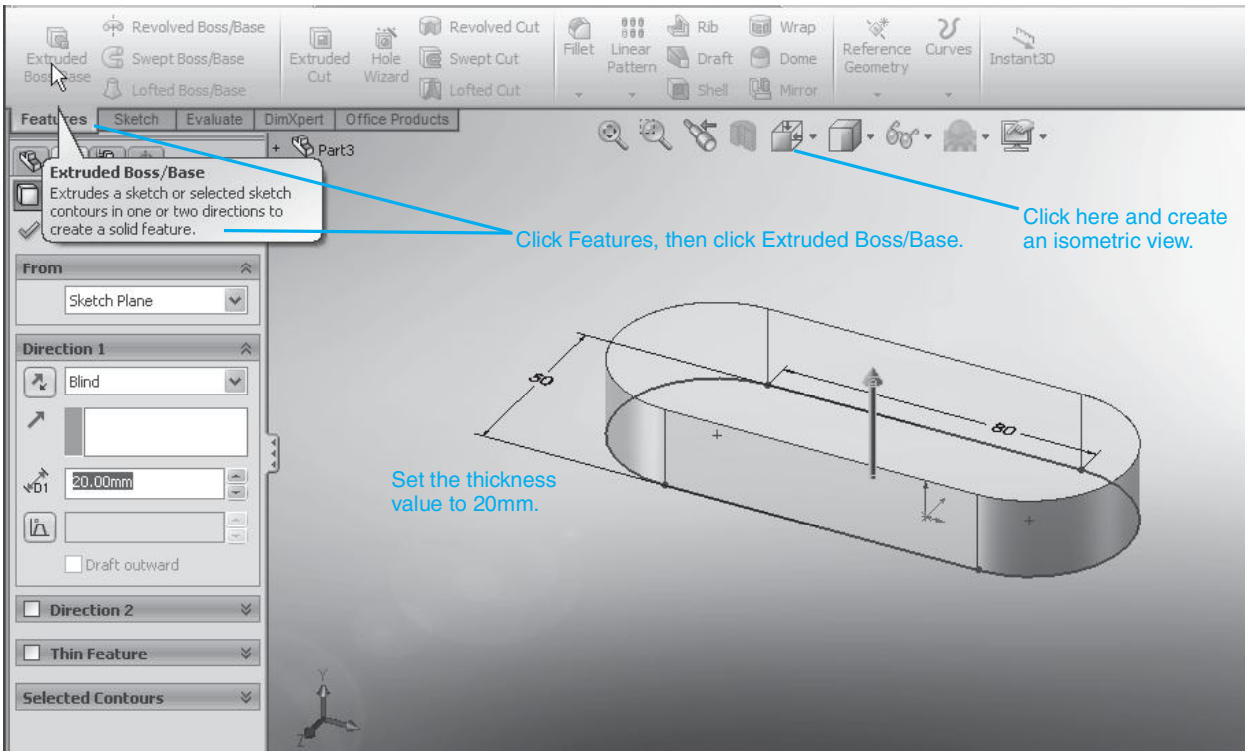


Figure 1-55

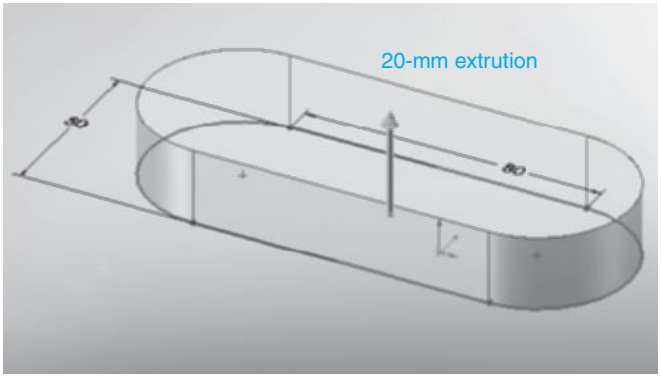


Figure 1-56

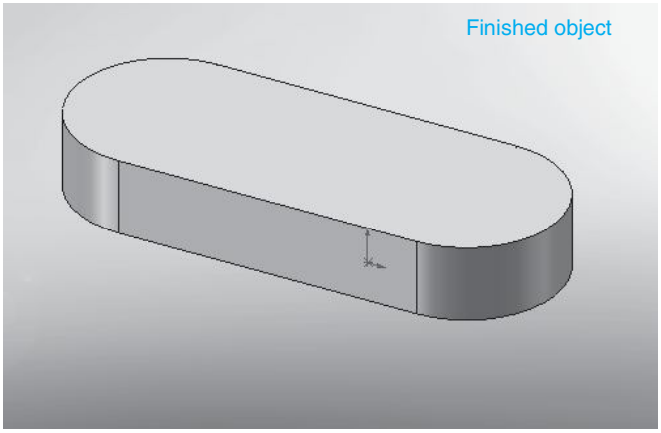


Figure 1-57

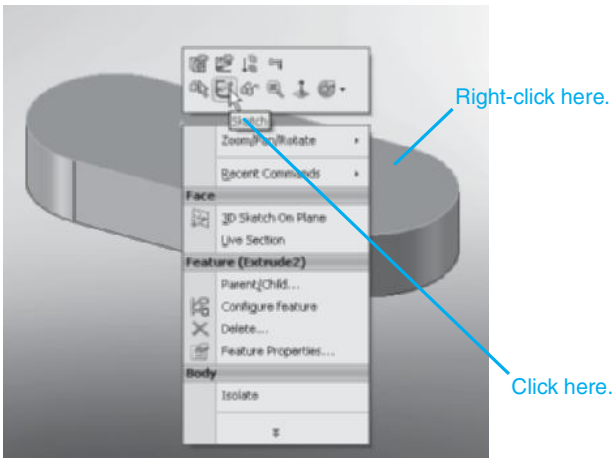
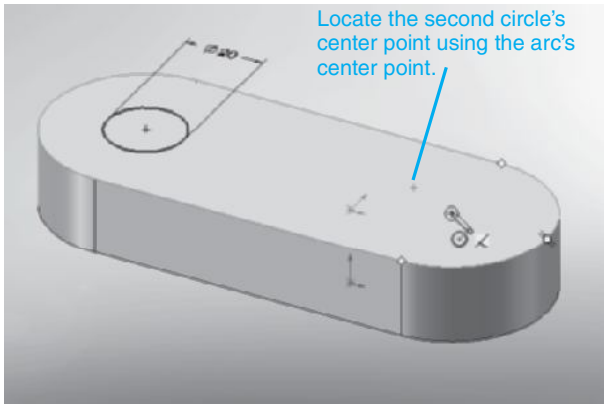


Figure 1-58



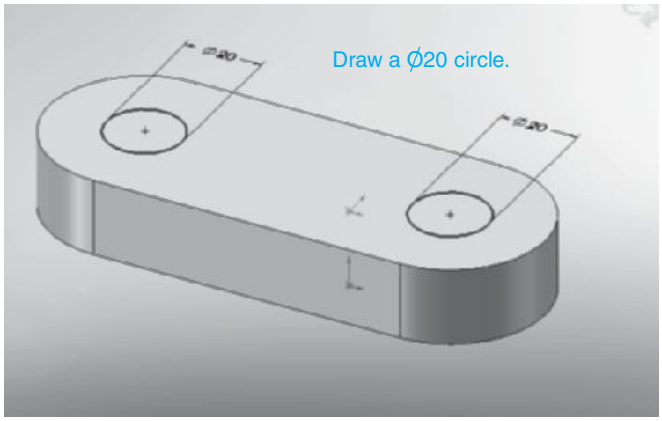


Figure 1-60

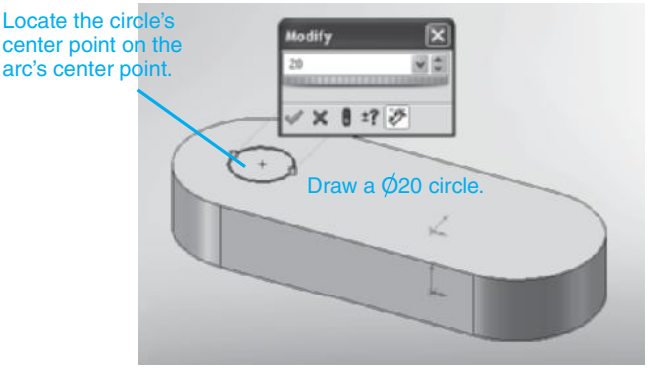


Figure 1-59

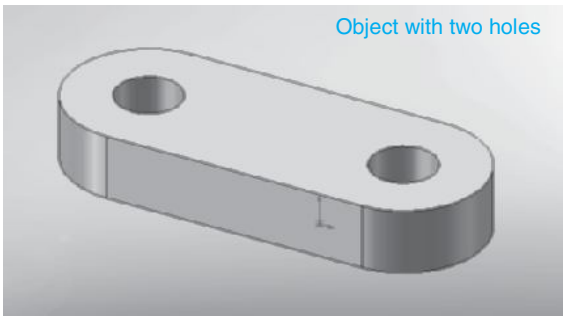


Figure 1-61

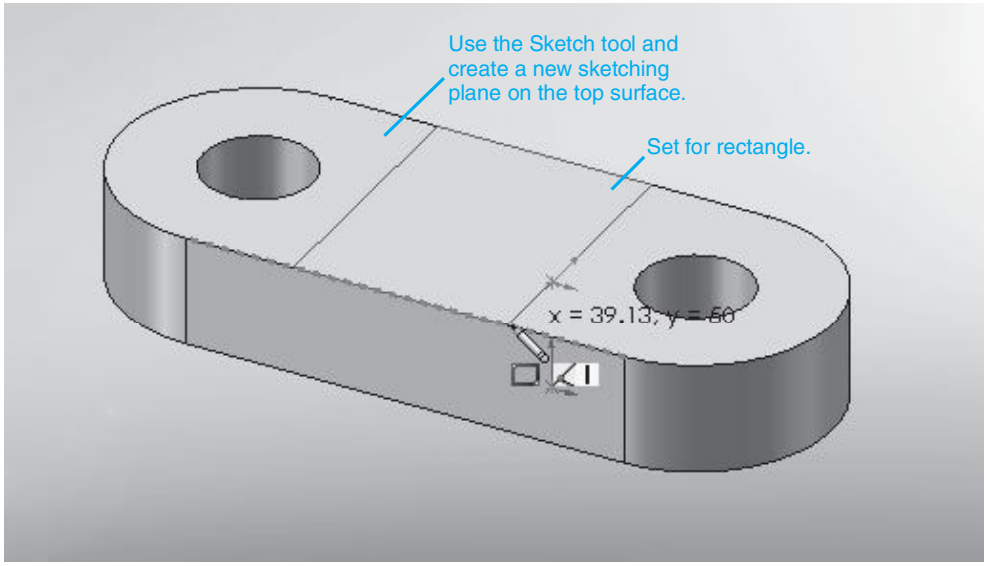


Figure 1-62

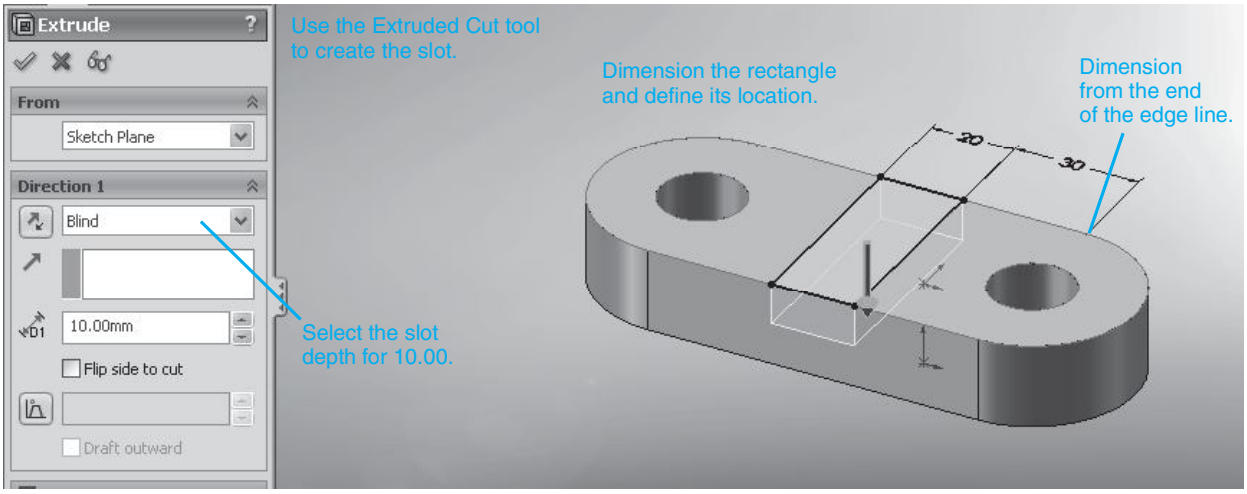


Figure 1-63

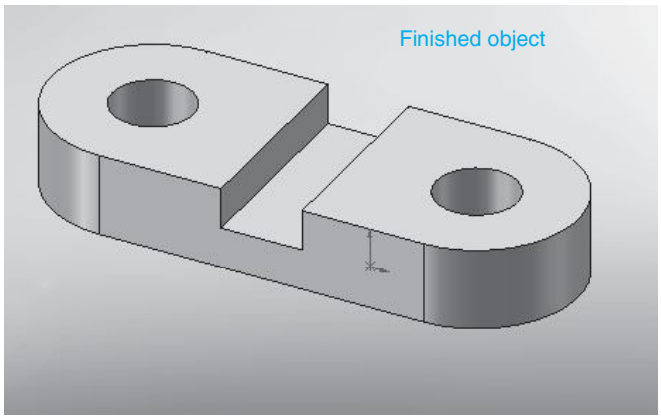


Figure 1-64

1-12 PROJECTS

Sketch the shapes shown in Figures P1-1 through Figure P1-18. Create 3D models using the specified thickness values.

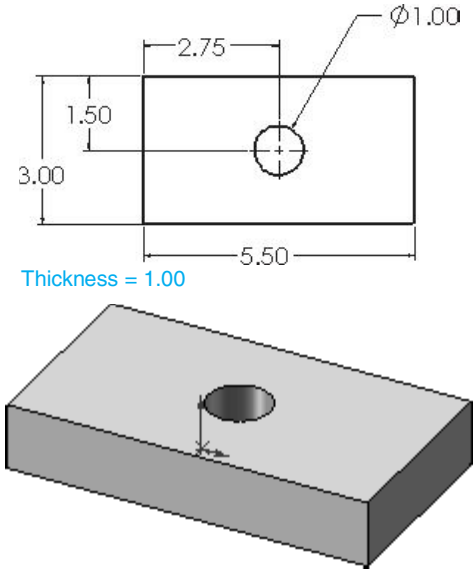


Figure P1-1 INCHES

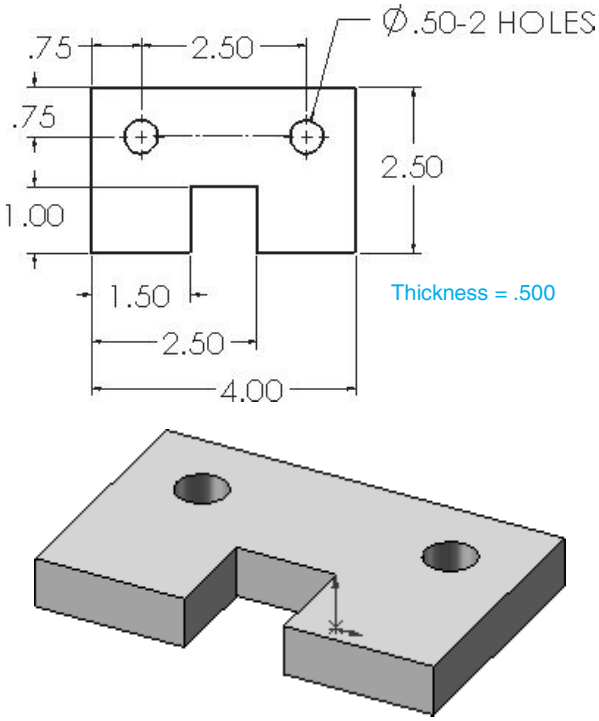


Figure P1-2 INCHES

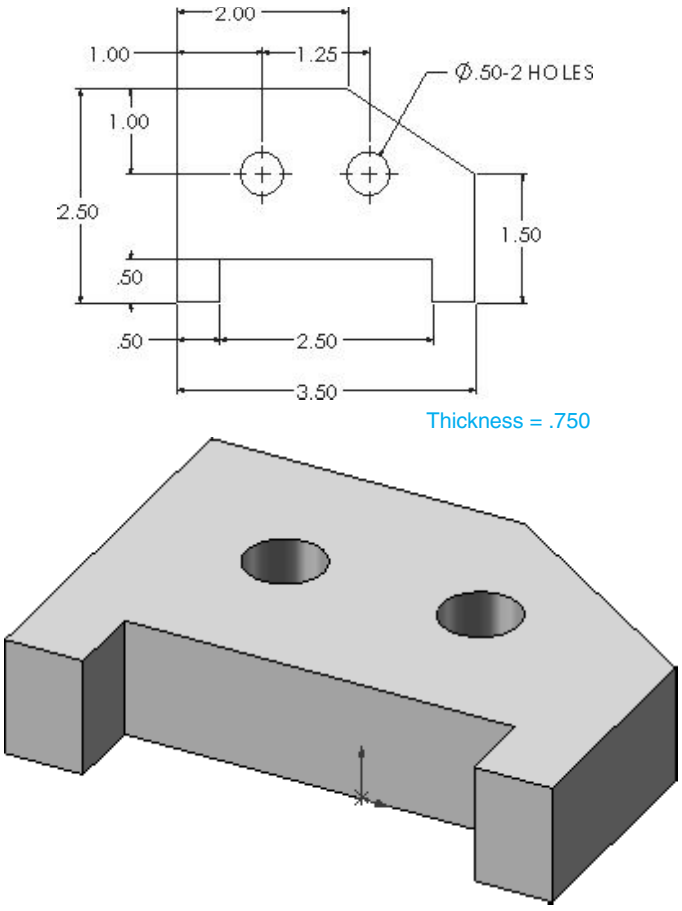


Figure P1-3 INCHES

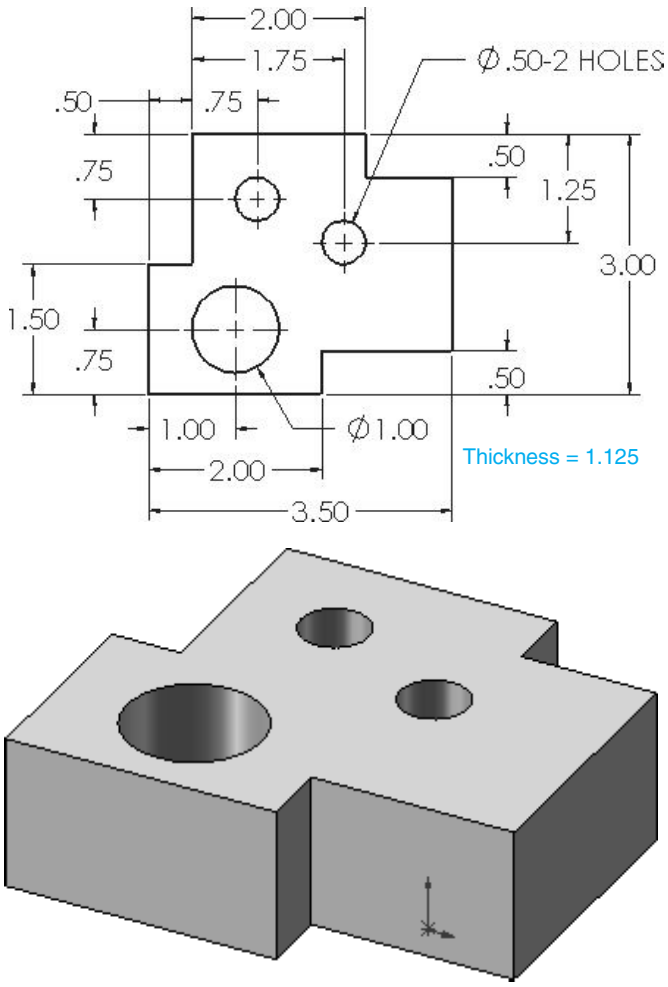


Figure P1-4 INCHES

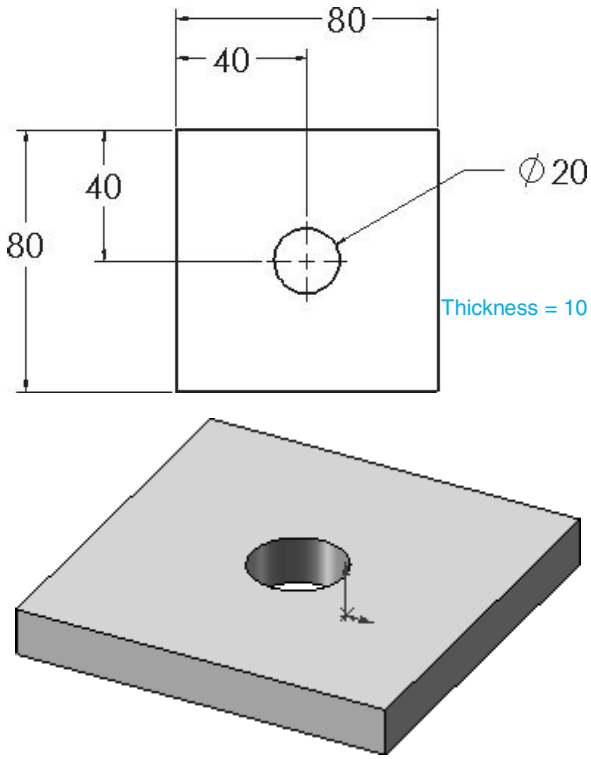


Figure P1-5 MILLIMETERS

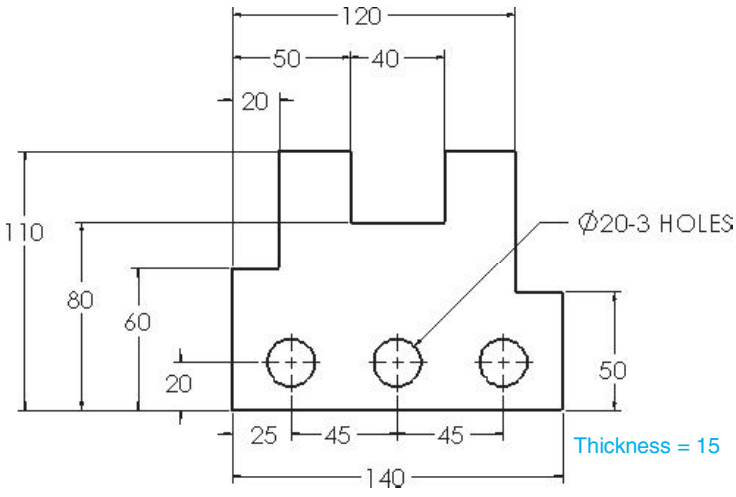


Figure P1-6 MILLIMETERS

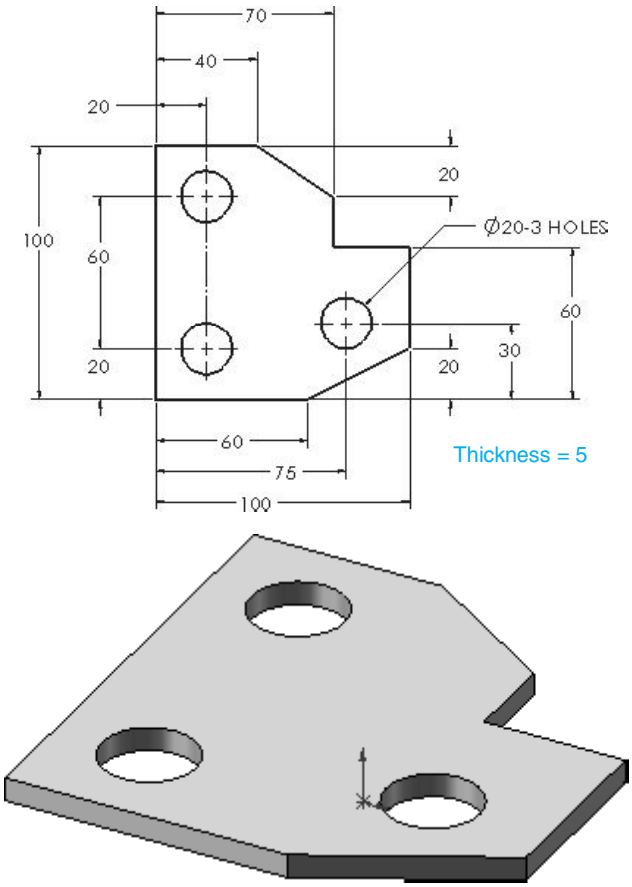


Figure P1-7 MILLIMETERS

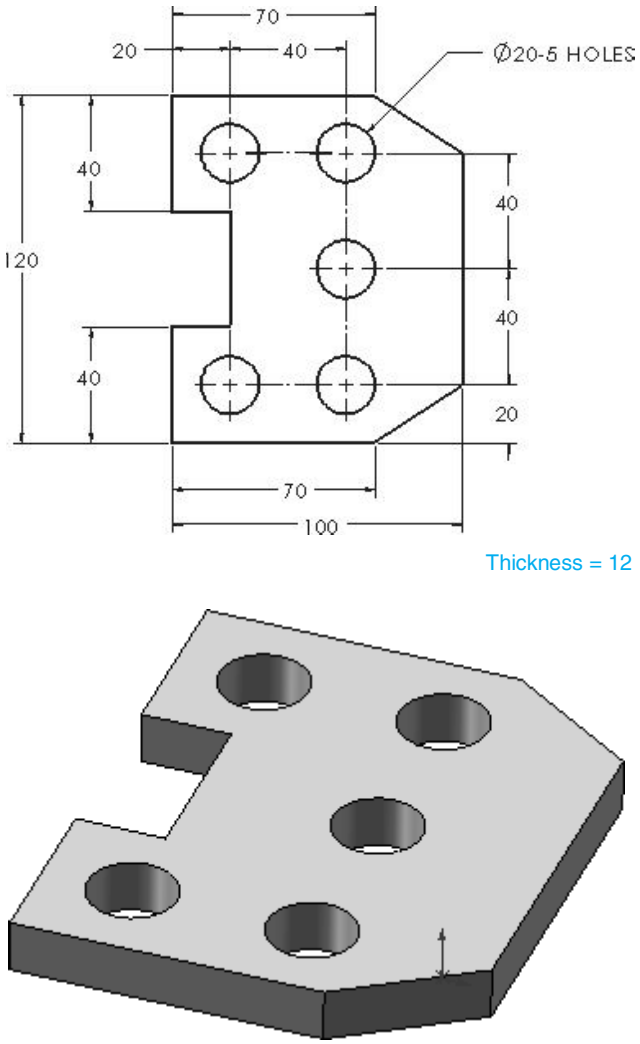


Figure P1-8 MILLIMETERS

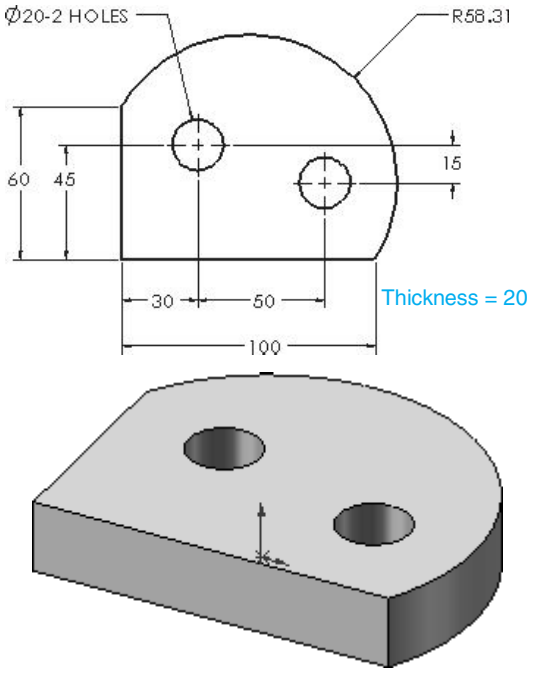


Figure P1-9 MILLIMETERS

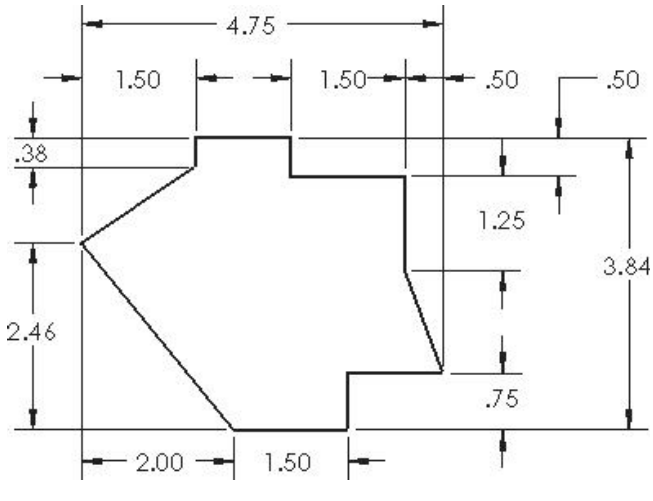


Figure P1-10 INCHES

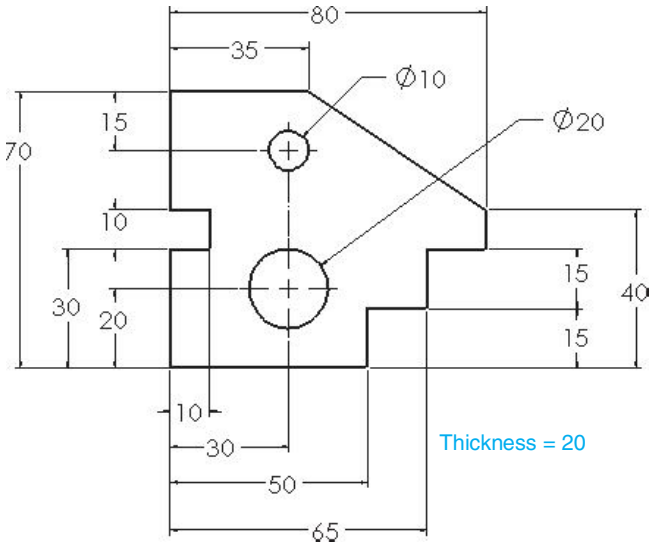


Figure P1-11 MILLIMETERS

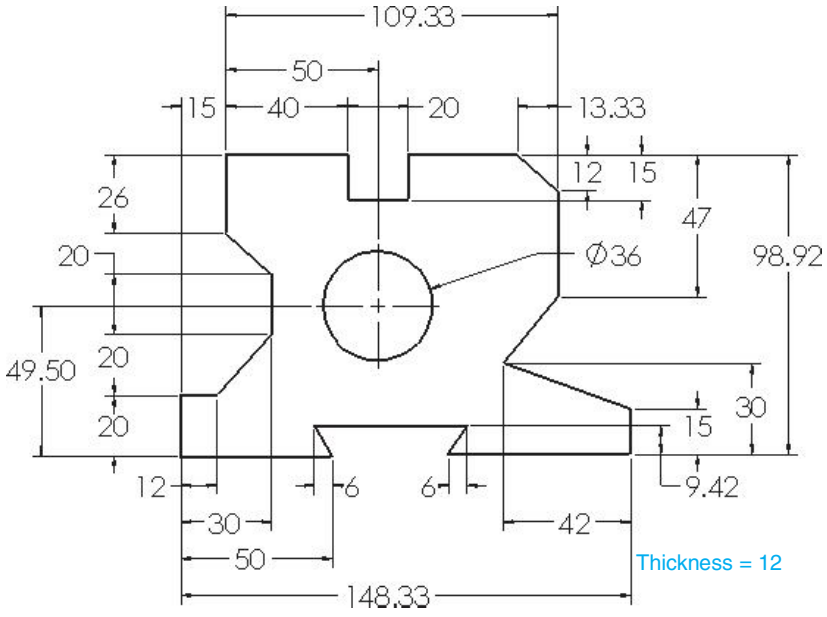


Figure P1-12 MILLIMETERS

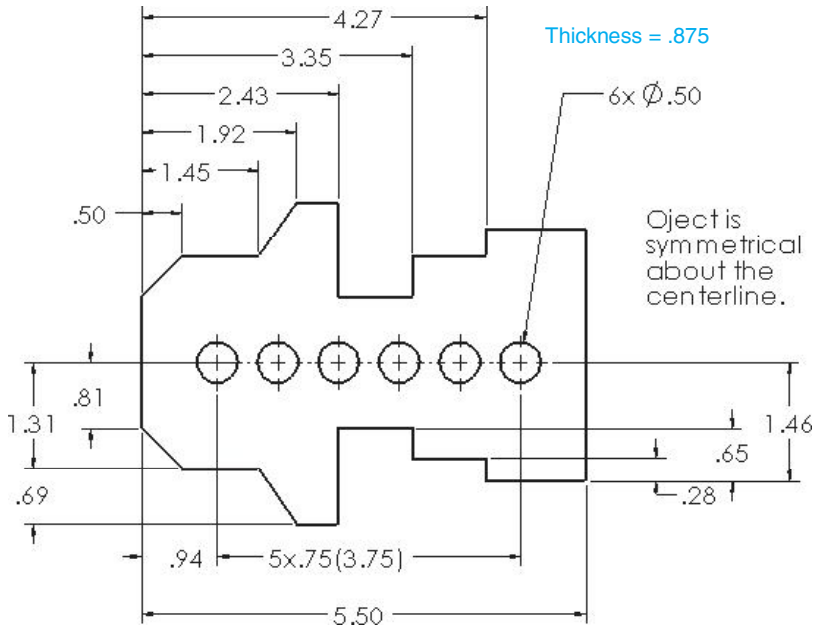


Figure P1-13 INCHES

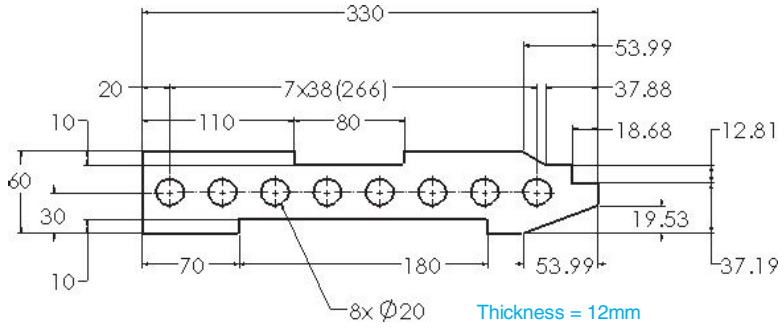
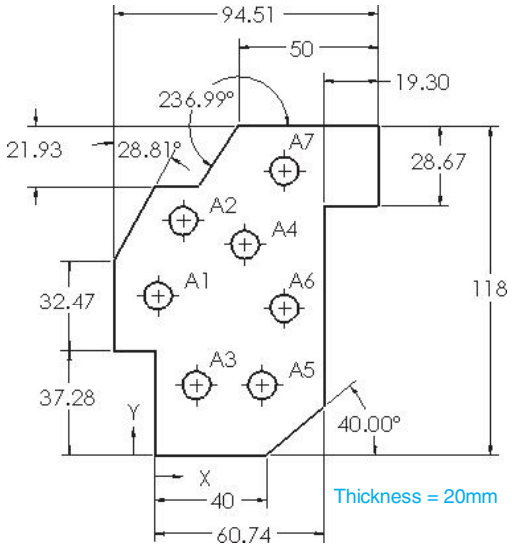


Figure P1-14 MILLIMETERS



TAG	X LOC	Y LOC	SIZE
A1	1.22	57.14	Ø10
A2	10.27	84.04	Ø10
A3	15	25	Ø10
A4	32.38	75.51	Ø10
A5	38.51	25	Ø10
A6	46.50	52.61	Ø10
A7	46.50	101.88	Ø10

Figure P1-15 MILLIMETERS

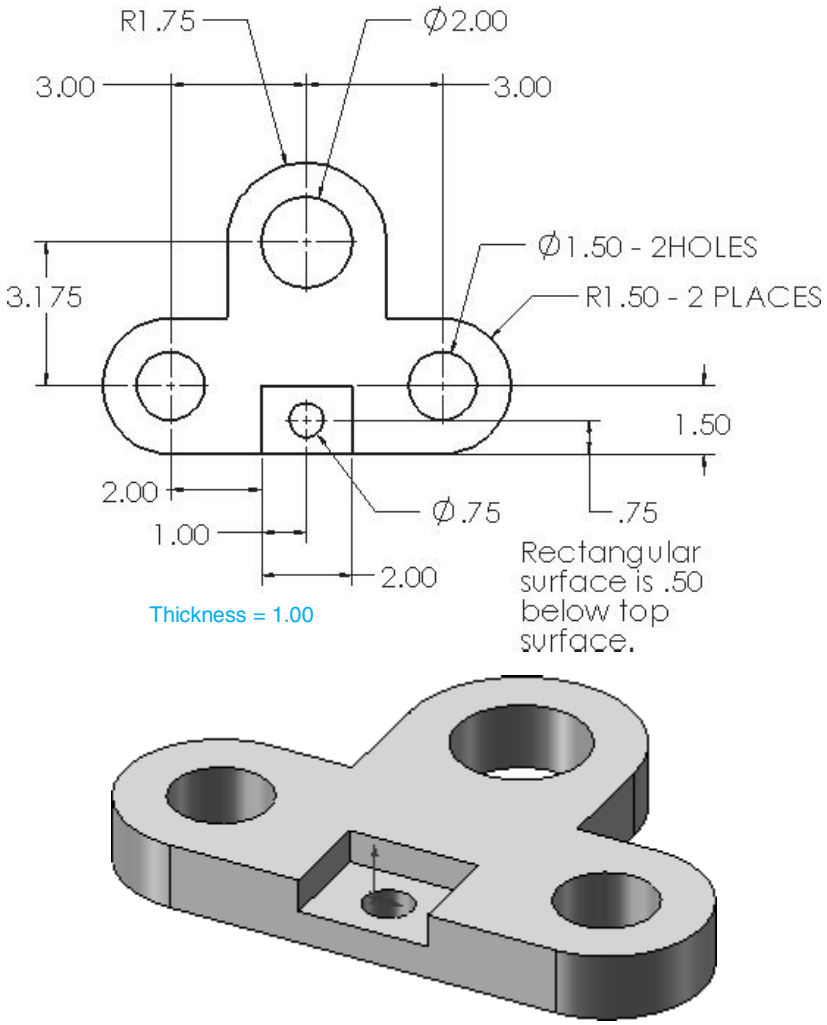


Figure P1-16 INCHES

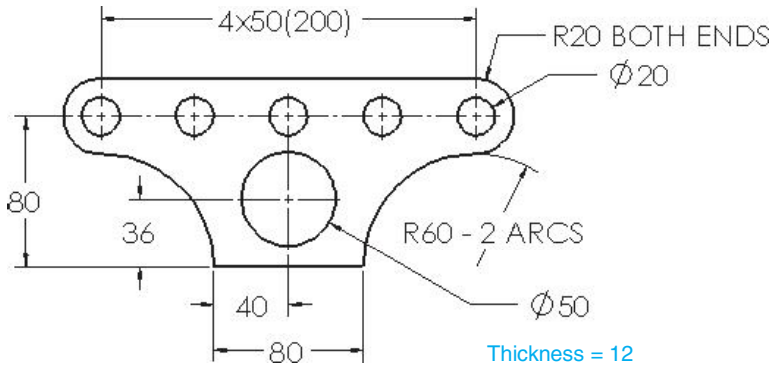


Figure P1-17 MILLIMETERS

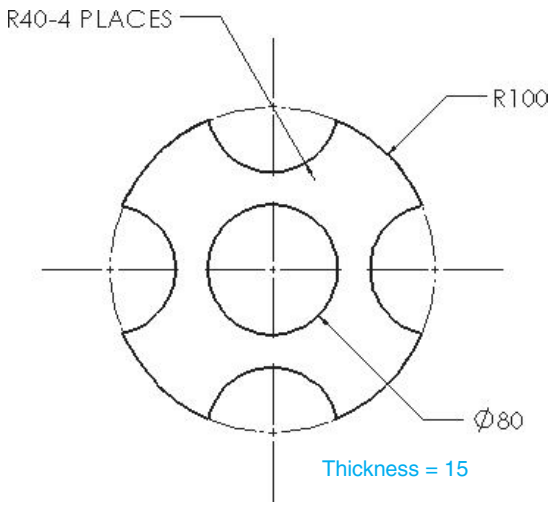


Figure P1-18 MILLIMETERS

CHAPTER 2

Sketch Entities and Tools

Objectives

- Learn about the **Sketch Entities** tools.
- Learn about the **Sketch Tools.**
- Use the **Sketch Tools** together to create shapes and parts.

2-1 INTRODUCTION

Figure 2-1 shows the **Sketch Entities** toolbar, and Figure 2-2 shows part of the **Sketch** toolbar. The **Sketch Entities** toolbar is accessed by clicking the **Tools** heading at the top of the screen. The **Sketch** tool is already on the **Part** document screen.

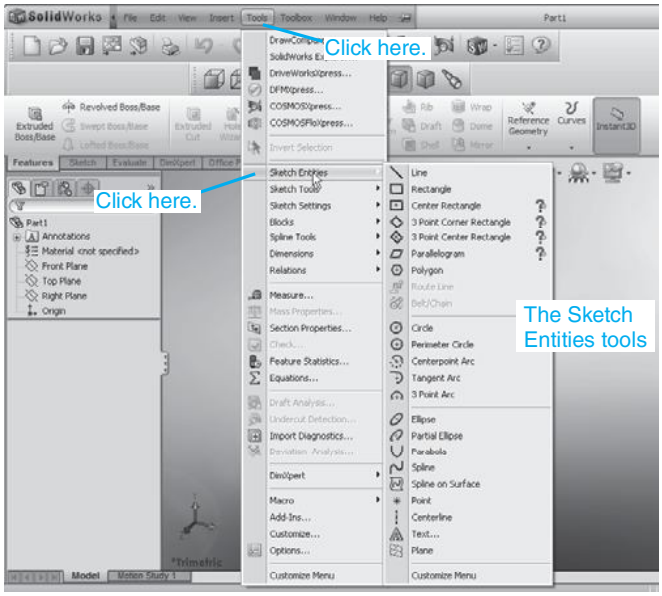


Figure 2-1

2-2 3 POINT ARC

Figure 2-3 shows three randomly located points. They were created using the **Point** tool.

1. Start a new **Part** document, click the **Sketch** group on the **Command Manager,** and select **Top Plane** from the **Features Manager.**