C H A P T E R 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.

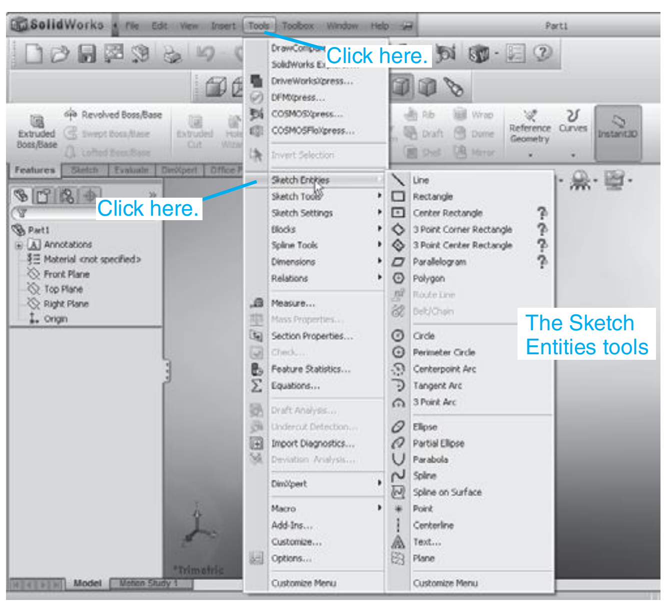


Figure 2-1

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.

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.**

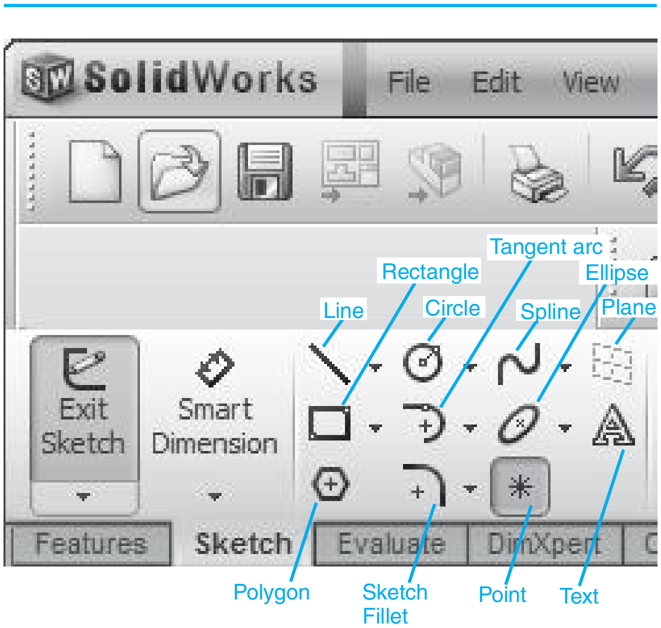


Figure 2-2

2. Use the **Point** tool and randomly locate three points approximately as shown.
3. Click the **3 Point Arc** tool and select three points to define an arc.

> ***Note:***
> An alternative method for creating a 3 point arc is to start a new **Part** document, select the **Sketch** group **Manager**, select the **Front** view, then click the **3 Point Arc** tool on the **Command Manager.** This tool will simultaneously construct an arc as the three points are selected.

> **TIP**
> Try clicking the points in different sequences and seeing the different arcs that are created.

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

> ***Note:***
> The **Arc Properties Manager** on the left side of the screen can be used to edit the location and size of the arc.

2-3 SKETCH FILLET AND UNDO TOOLS

Figure 2-4 shows a 2.50 × 5.00-in. rectangle. It was created using the **Rectangle** tool and sized using the **Smart Dimension** tool. See Section 1-4.

1. Start a new **Part** document, select the **Sketch** tool, and click the **Top Plane** option.
2. Use the **Rectangle** tool and create a **2.50 × 5.00-in.** rectangle. Use the **Smart Dimension** tool to size the rectangle.
3. Click the **Sketch Fillet** tool on the **Sketch** group on the **Command Manager.**

The **Sketch Fillet** options will appear in the **Sketch Fillet Properties Manager** on the left side of the drawing screen.

4. Set the radius value for the fillet for **0.50in.**

See Figure 2-5.

> **TIP**
> The scroll arrows to the right of the radius value box can be used to change the radius value, or a new value may be typed in.

5. Click the left vertical line, then click the top horizontal line.

A preview of the fillet will appear between the two lines.

6. Add an **R = 0.50in.** fillet to the upper right corner of the rectangle by selecting the top horizontal line and the right vertical line.
7. Reset the **Fillet Parameters** to **0.25in.** and add fillets to the two bottom corners of the rectangle.
8. Click the check mark in the **Sketch Fillet Properties Manager.**

See Figure 2-6.

9. Click the **Undo** tool and remove the four fillets.
10. Click the **Sketch Fillet** tool, define the radius as 1.24, and create four fillets as shown.
11. Close **(Save)** the document.

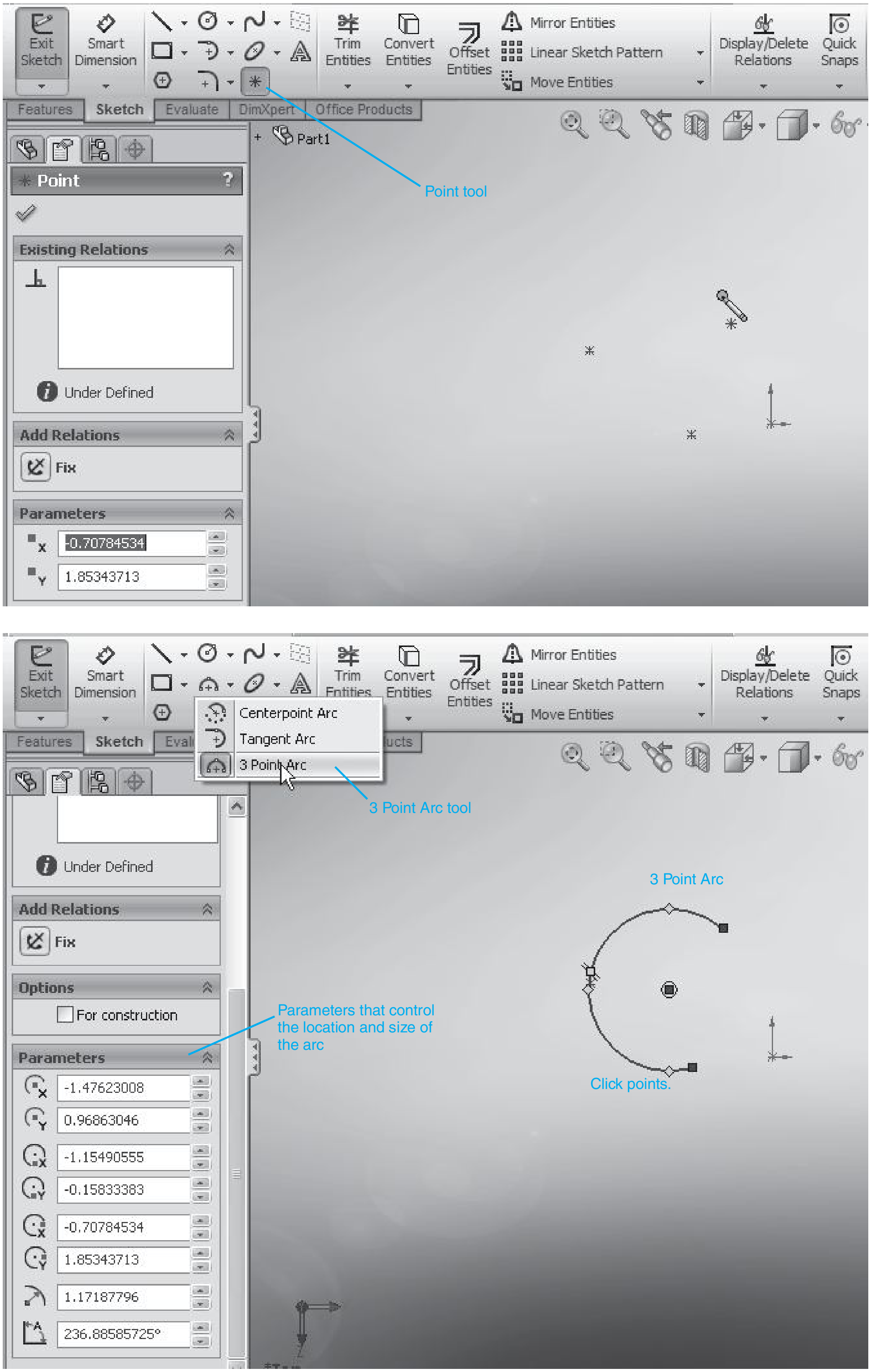

Point tool
Point
Existing Relations
Under Defined
Add Relations
Fix
Parameters
-0.70784534
1.85343713
Centerpoint Arc
Tangent Arc
3 Point Arc
3 Point Arc tool
3 Point Arc
Options
For construction
Parameters that control the location and size of the arc
Click points.
-1.47623008
0.96863046
-1.15490555
-0.15833383
-0.70784534
1.85343713
1.17187796
236.88585725°

Figure 2-3

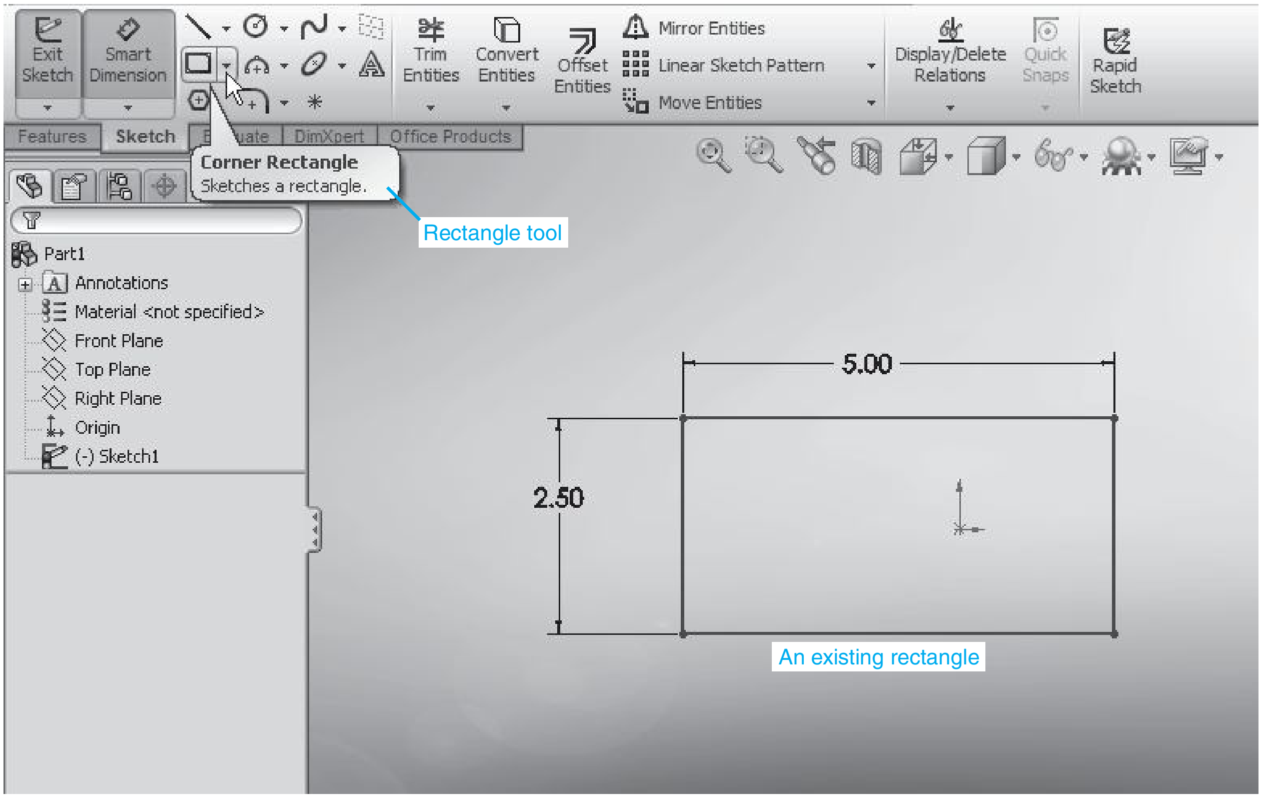


Figure 2-4

Figure 2-7 shows the original 2.50 × 5.00-in. rectangle modified using the **Sketch** tool to create a rounded shape with radii of 1.24.

2-4 SPLINE

Figure 2-8 shows a spline.

1. Start a new **Part** document, click the **Sketch** tool, and click the **Top Plane** option.
2. Click the **Spline** tool.
3. Select a starting point for the spline and click the point.
4. Select other points and extend the spline.
5. When the spline is complete, right-click the mouse and click the **Select** option or select the check mark in the **Spline Properties Manager.**

A spline may be edited by moving any one of its defining points.

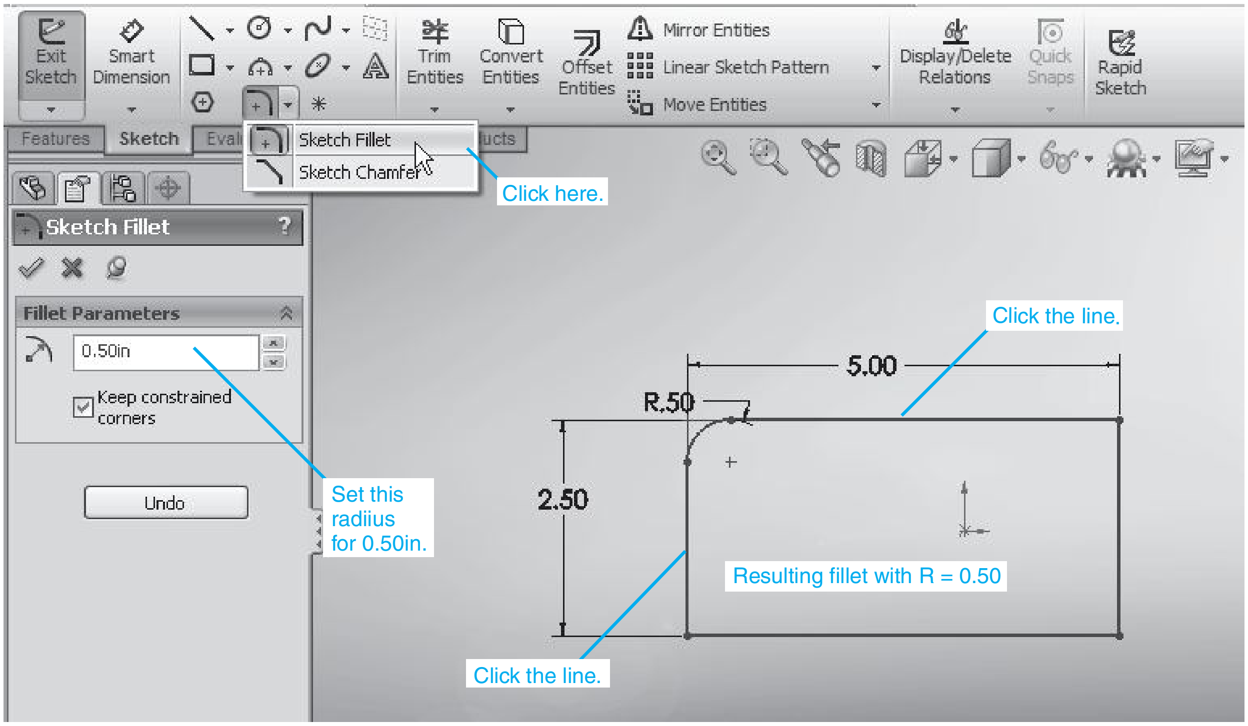


Figure 2-5

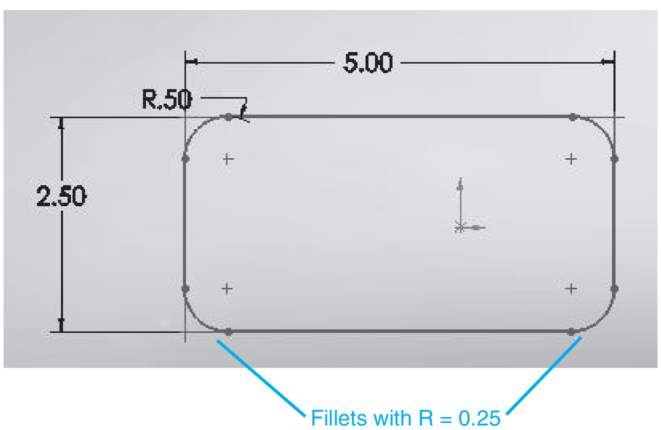


Figure 2-6

To Edit a Spline

1. Click and hold one of the defining points and drag the point to a new location and release the mouse button.

The point's parameters will be listed in the **Parameters** box. See Figure 2-8. These values will change as the point is moved. Point values may be entered directly into the **Parameters** box. Click the check mark in the **Point Properties Manager** to apply the entered values to the spines's points.

2-5 POLYGON

The **Polygon** tool is accessed by clicking **Tools** on the main menu (top of the screen), clicking **Sketch Entities** on the drop-down menu, then selecting the **Polygon** tool. See Figure 2-1.

1. Start a new **Part** document, click the **Sketch** tool, and click the **Top Plane** option.
2. Select the **Polygon** tool.
3. Under the **Polygon Properties Manager** define the number of sides as six.
4. Select a center point for the polygon by clicking the left mouse button.
5. Drag the cursor away from the center point to create the polygon.

A set of angular coordinate values will appear next to the cursor as it is moved. The values will define the distance from the center point to the edge line.

The example shown was created by dragging the cursor horizontally to the right.

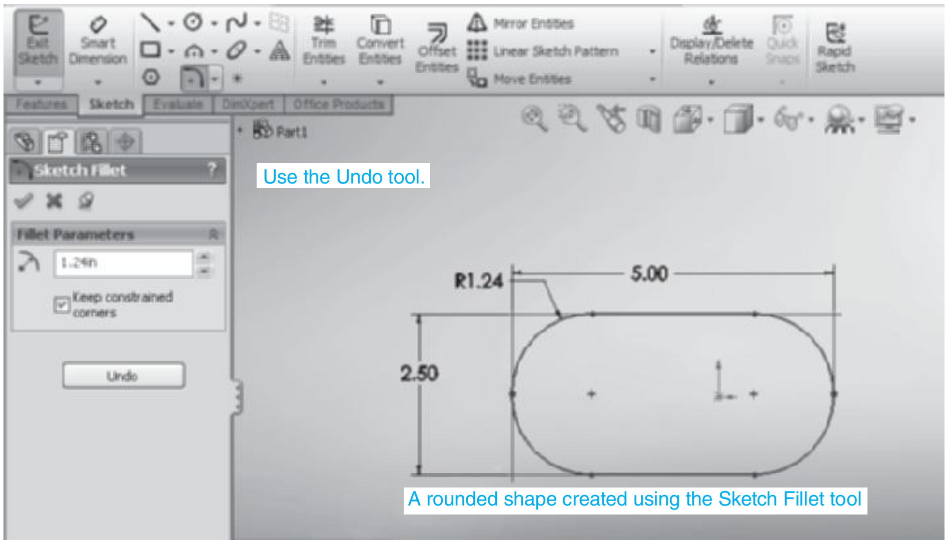


Figure 2-7

> **TIP**
>
> SolidWorks defines a horizontal line to the right as 0°. The counterclockwise direction is the positive direction.

6. Use either the parameter values or the **Smart Dimension** tool to size the polygon.
7. Click the OK check mark to complete the polygon construction.

The 3.00 in. dimension shown in Figure 2-9 is the distance *across the flats* of the hexagon. The distance along one of the sides is called the *edge distance,* and the distance across the hexagon from one corner to another is called the *corner distance.*

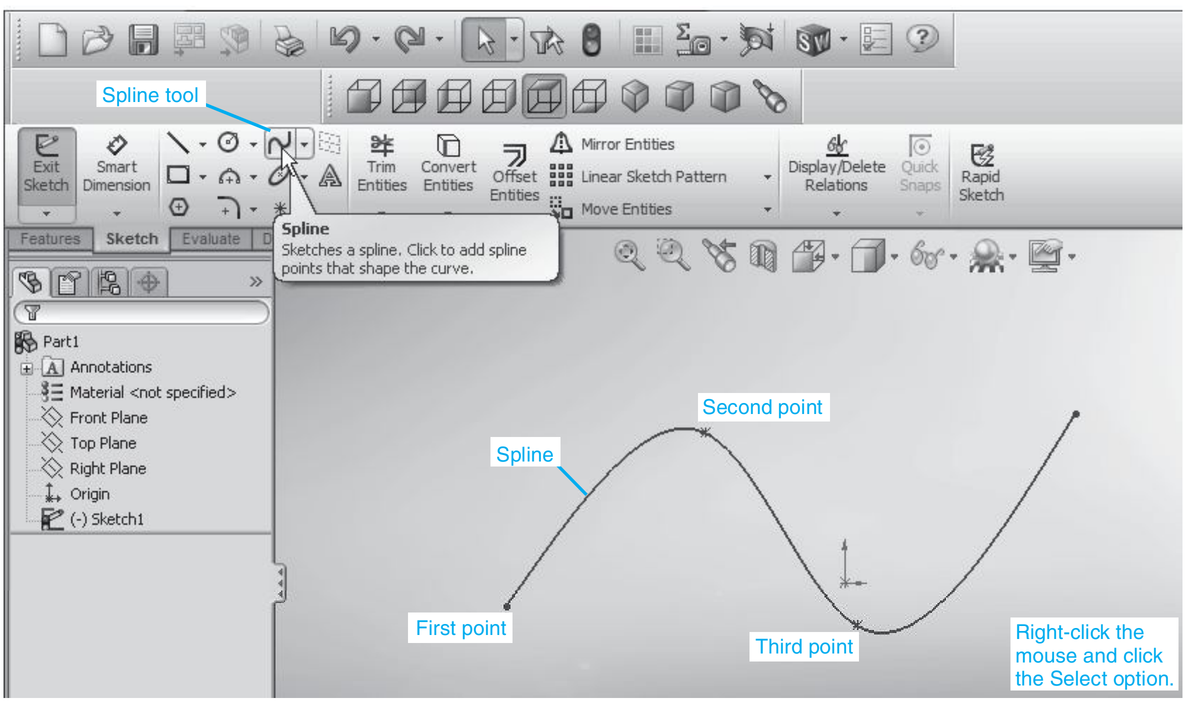


Figure 2-8

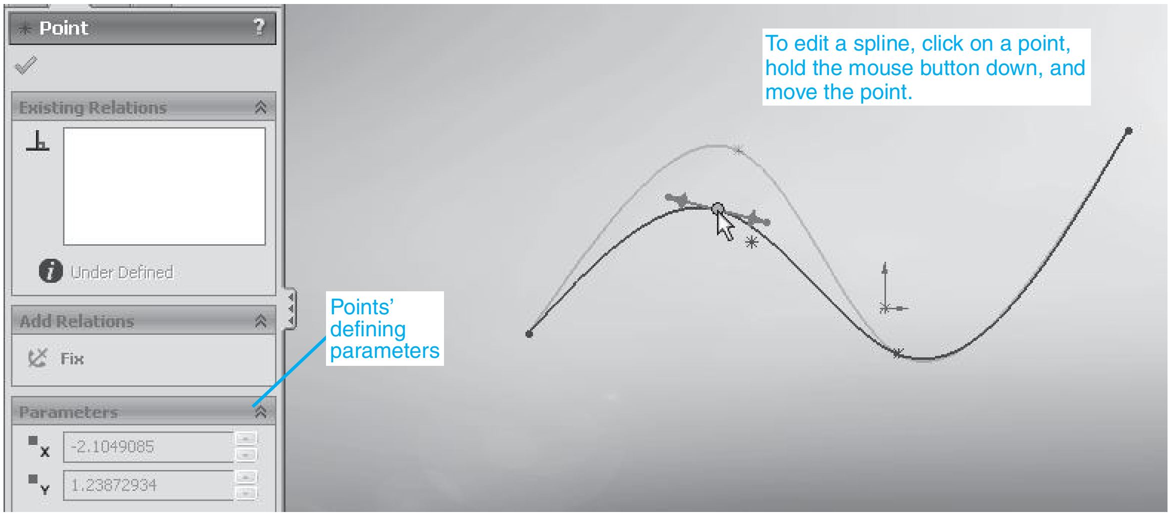


Figure 2-8 *(continued)*

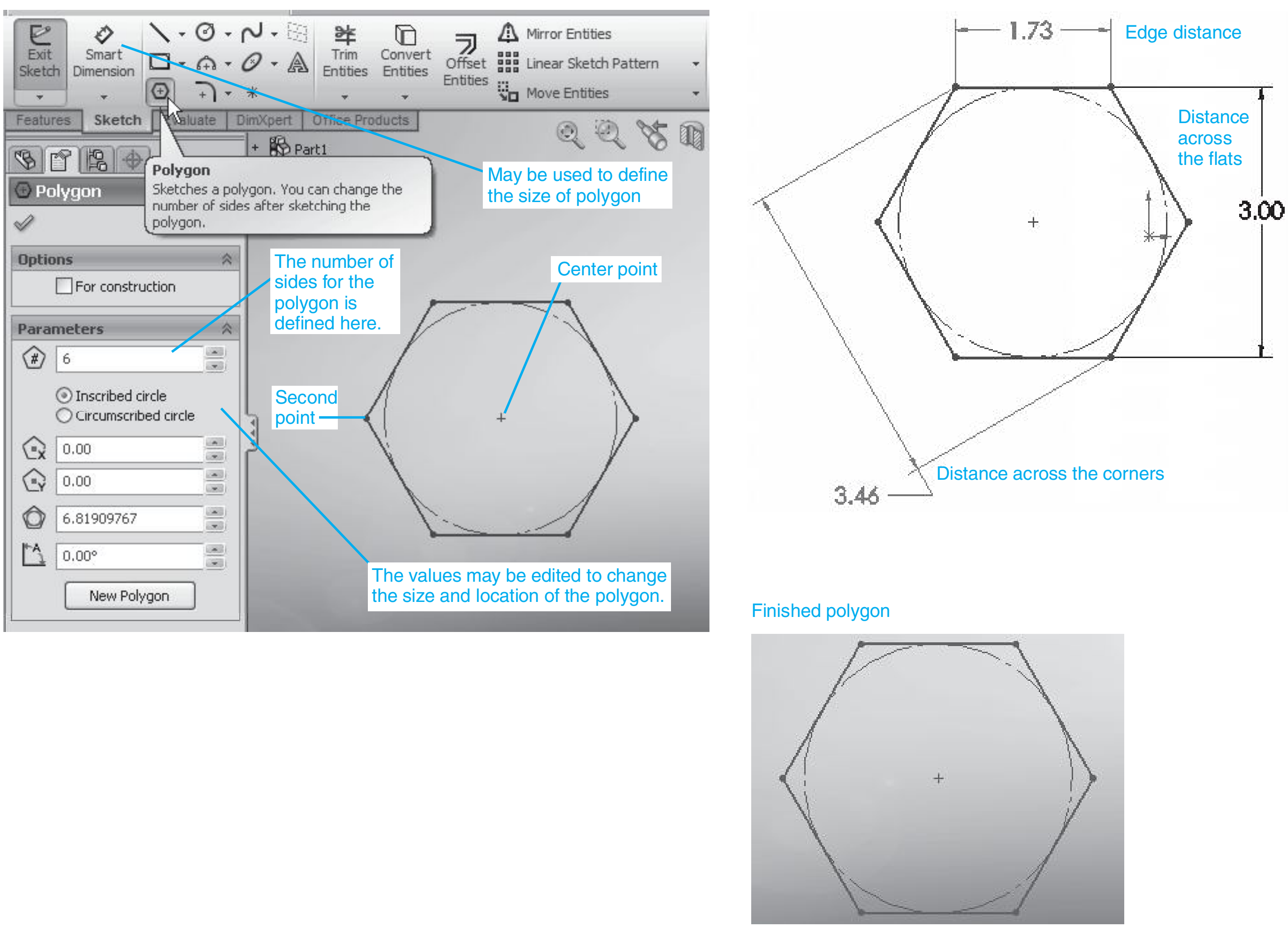


Figure 2-9

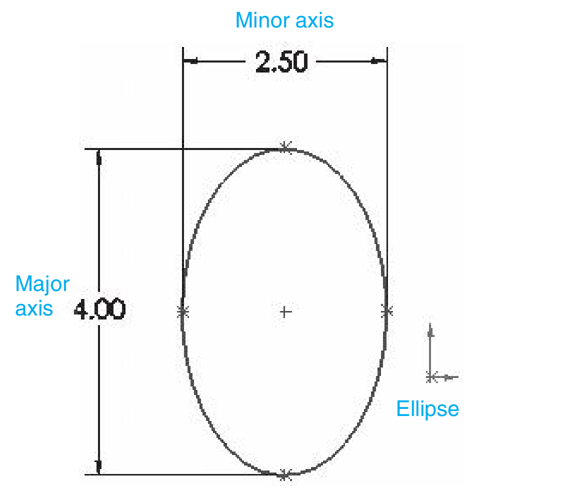


Figure 2-10

2-6 ELLIPSE

The **Ellipse** tool is located on the **Sketch** toolbar. See Figure 2-2.

Ellipses are defined by their major and minor axes. See Figure 2-10.

1. Start a new **Part** document, click the **Sketch** tool, and click the **Top Plane** option.
2. Access the **Ellipse** tool from the **Sketch** toolbar.
3. Locate a center point for the ellipse and drag the cursor horizontally away from the center point.

Values for the major and minor axes will appear as the cursor is moved. The initial values will be equal, as the first part of the ellipse construction is a circle. See Figure 2-11.

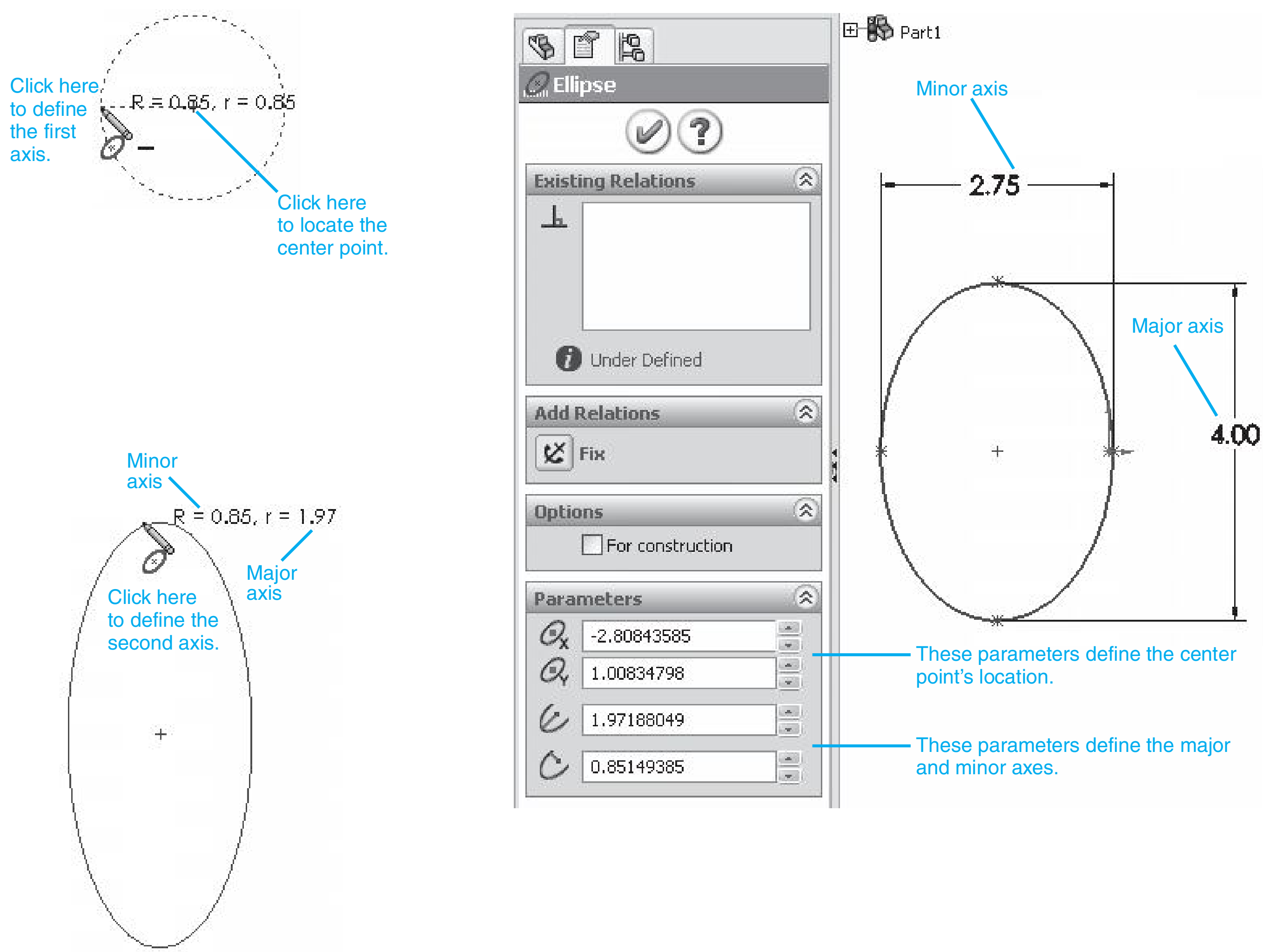


Figure 2-11

4. Locate the first point for the ellipse; click the mouse.
5. Locate the second point along the second axis; click the mouse.

The finished size of the ellipse may be defined using either the parameter values in the **Ellipse Properties Manager** or the **Smart Dimension** tool.

6. Define the major and minor axes for the ellipse.
7. Click the OK check mark.

2-7 PARABOLA

A ***parabola*** is the loci of points such that the distance between a fixed point, the ***focus,*** and a fixed line, the ***directrix***, are always equal. See Figure 2-12.

The **Parabola** tool is a flyout from the **Ellipse** tool on the **Sketch** toolbar.

1. Start a new **Parts** document, click the **Sketch** tool, and select the **Top Plane** option.
2. Access the **Parabola** tool from the **Sketch** toolbar.
3. Select a location for the focus point.

In this example the 0,0,0 coordinate point, or origin, was selected as the focus point. See Figure 2-12. The directrix was added to the illustration to help you understand how the parabolic shape is generated. The directrix will not appear during the SolidWorks construction.

4. Select a point away from the locus; click the mouse.
5. Select the left endpoint for the parabola.
6. Select the other endpoint for the parabola.

The **Parameters** section of the **Parabola Properties Manager** can be used to change the location of the focus point and the orientation of the parabola. The parabola may also be sized using the **Smart Dimension** tool.

7. Click the OK check mark.

2-8 OFFSET

The **Offset** tool is used to draw entities parallel to existing entities. Figure 2-13 shows an existing line. The **Offset** tool is used to draw a line parallel to the exisiting line and of equal length.

1. Start a new **Part** document, click the **Sketch** group on the **Command Manager,** and click the **Top Plane** option.
2. Draw a random line on the screen using the **Line** tool.
3. Access the **Offset Entities** tool.

The **Offset** tool is located on the **Sketch** toolbar.

> ***Note:***
> You can also access the **Offset** tool by clicking the **Offset Entities** tool directly from the **Sketch** group.

4. Define the distance between the existing line and the offset line by entering the distance into the **Offset Entities Properties Manager.**

> ***Note:***
> As the arrows to the right of the defining offset value box are clicked, the offset line will move in real time to reflect the increase or decrease in the offset distances.

An arrow will appear on the existing line indicating the default direction of the offset. You can change the direction of the offset by moving the mouse to either side of the line or by checking the **Reverse** box in the Offset Entities **Properties Manager.**

5. Click the side of the line where the offset line is to be located.
6. Click the OK check mark.

Entities other than lines may be offset. Figure 2-14 shows an offset circle and an offset rectangle.

2-9 TRIM

The **Trim** tool is used to remove unwanted entities from existing sketches.

Figure 2-15 shows an existing configuration consisting of a circle, a rectangle, and a line. The **Trim** tool will be used to remove a segment of the line from within the circle and the rectangle.

1. Start a new **Part** document, click the **Sketch** group, and select the **Top Plane** option.
2. Draw a line, a circle, and a rectangle approximately as shown. Exact sizes are not required.
3. Access the **Trim** tool.

The **Trim** tool can be accessed by using the **Trim** tool directly from the **Sketch** group.

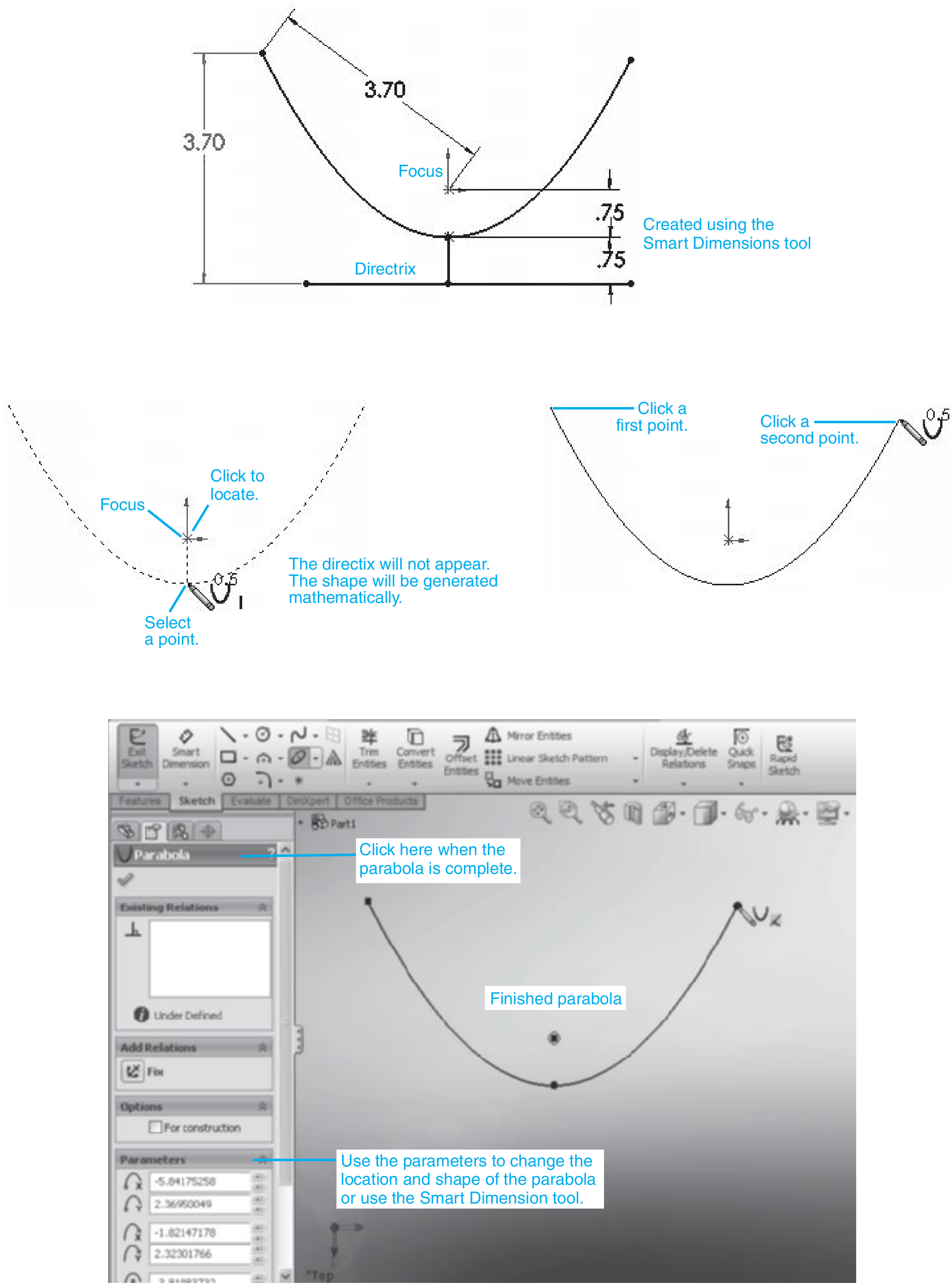

3.70
3.70
Focus
.75
.75
Created using the
Smart Dimensions tool
Directrix
Focus
Click to
locate.
Select
a point.
The directix will not appear.
The shape will be generated
mathematically.
Click a
first point.
Click a
second point.
Parabola
Existing Relations
Under Defined
Add Relations
Fix
Options
For construction
Parameters
-5.84175258
2.36950049
-1.82147178
2.32301766
Click here when the
parabola is complete.
Finished parabola
Use the parameters to change the
location and shape of the parabola
or use the Smart Dimension tool.

Figure 2-12

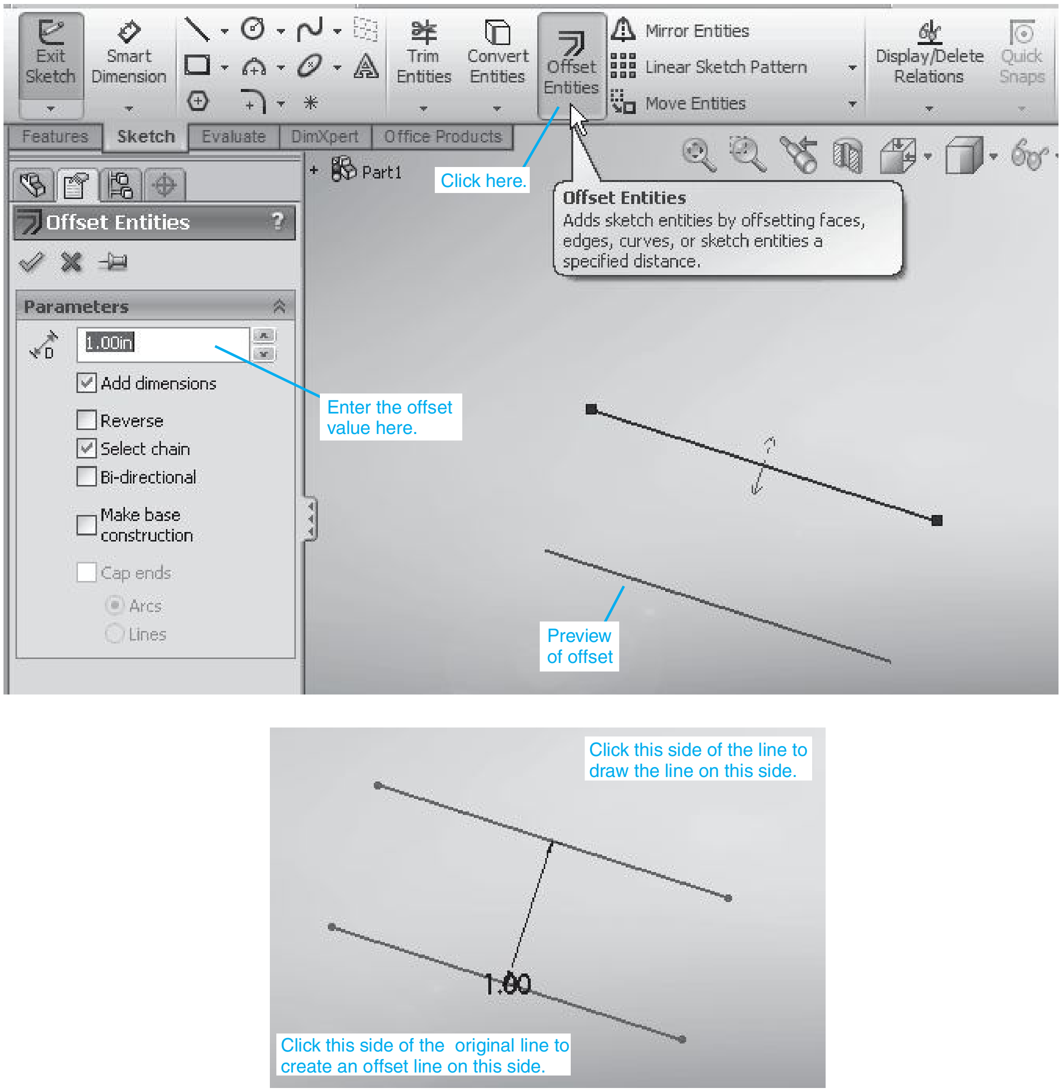
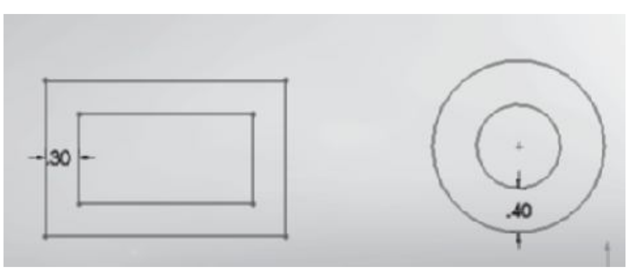


Figure 2-13



An offset rectangle An offset circle

Figure 2-14

4. Move the cursor to the line segment within the circle; click the line.

The segment will change colors when selected. The line segment will be removed when clicked.

5. Select the line segment within the rectangle; click the segment.
6. Click the OK check mark.

2-10 EXTEND

The **Extend** tool is used to extend existing lines and entities to new lengths or to other sketch entities.

Figure 2-16 shows a 1.50 × 4.00-in. rectangle. This example shows how to extend the rectangle so that it measures 1.5 × 5.5 in.

1. Start a new **Part** document, click the **Sketch** group, and select the **Top Plane** option.
2. Draw a **1.5 × 4.00-in.** rectangle.

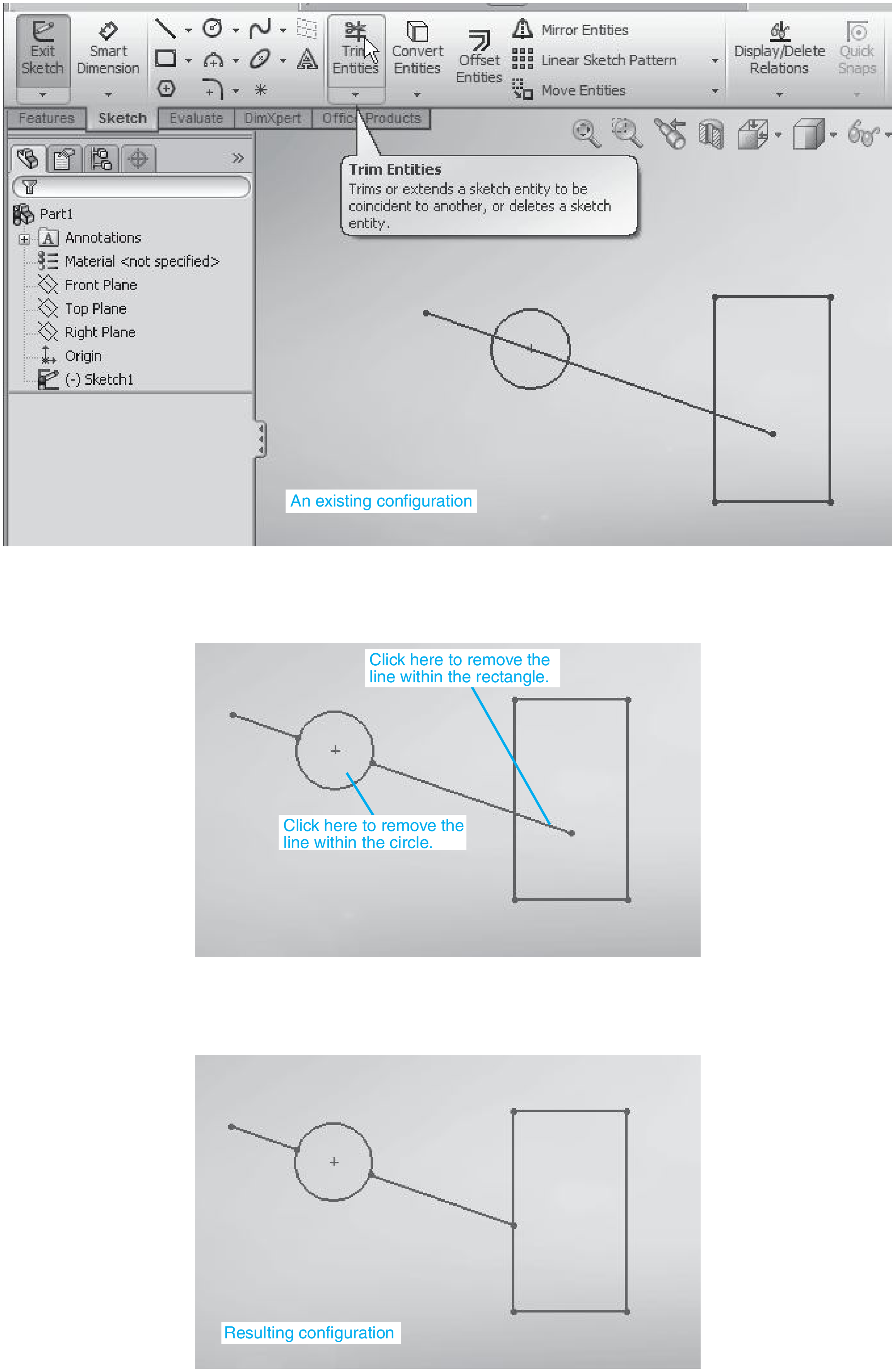

Exit Sketch
Smart Dimension
Trim Entities
Convert Entities
Offset Entities
Mirror Entities
Linear Sketch Pattern
Move Entities
Display/Delete Relations
Quick Snaps
Features
Sketch
Evaluate
DimXpert
Office Products
Trim Entities
Trims or extends a sketch entity to be coincident to another, or deletes a sketch entity.
Part1
Annotations
Material <not specified>
Front Plane
Top Plane
Right Plane
Origin
(-) Sketch1
An existing configuration
Click here to remove the line within the rectangle.
Click here to remove the line within the circle.
Resulting configuration

Figure 2-15

3. Draw a line parallel to the right vertical line of the rectangle. Locate the line **5.5 in.** from the left vertical line of the rectangle.
4. Access the **Extend** tool.

The **Extend** tool is a flyout from the **Trim Entities** tool located on the **Sketch** toolbar. The cursor will include the **Extend** icon as long as the **Extend** tool is active.

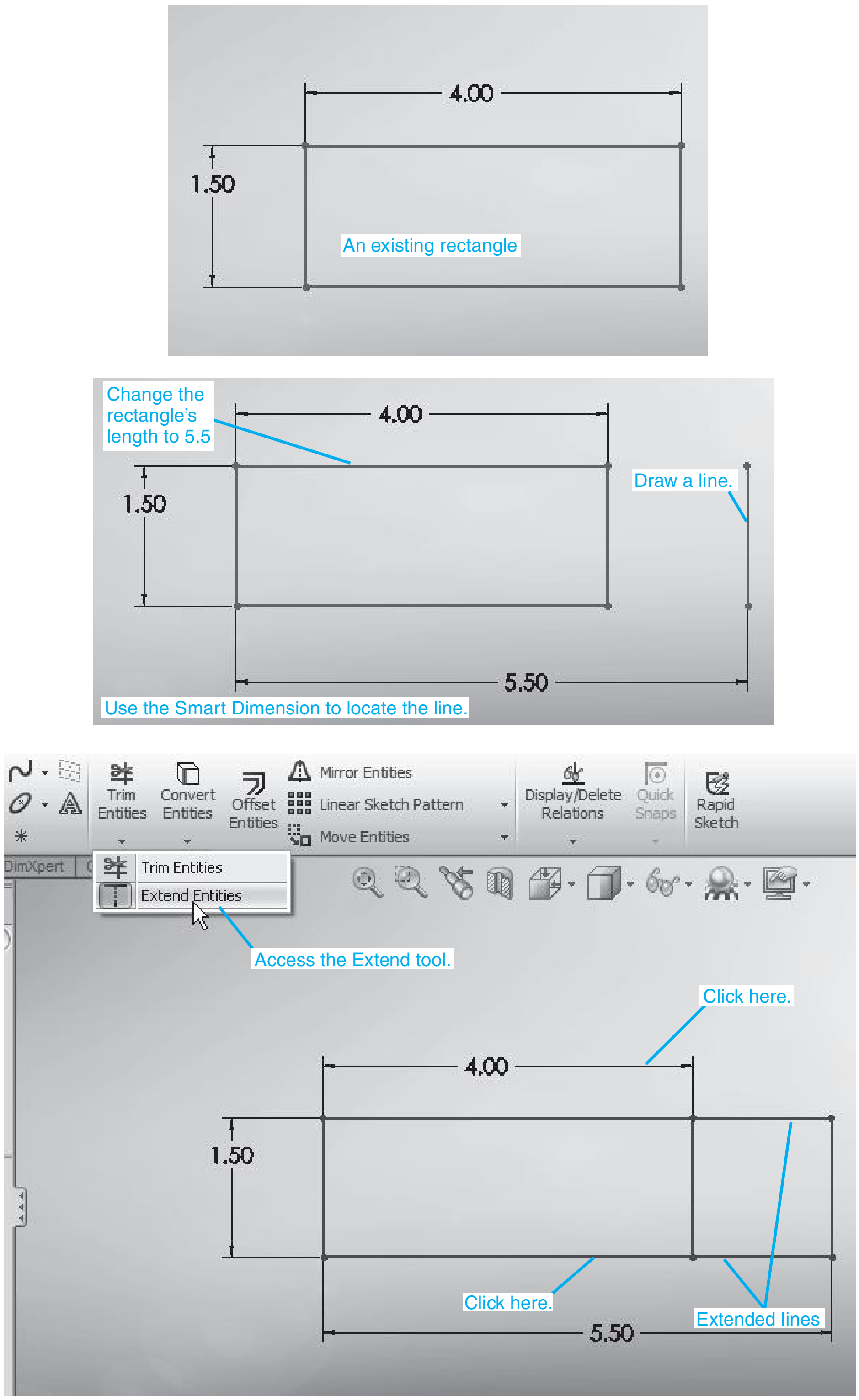


Figure 2-16

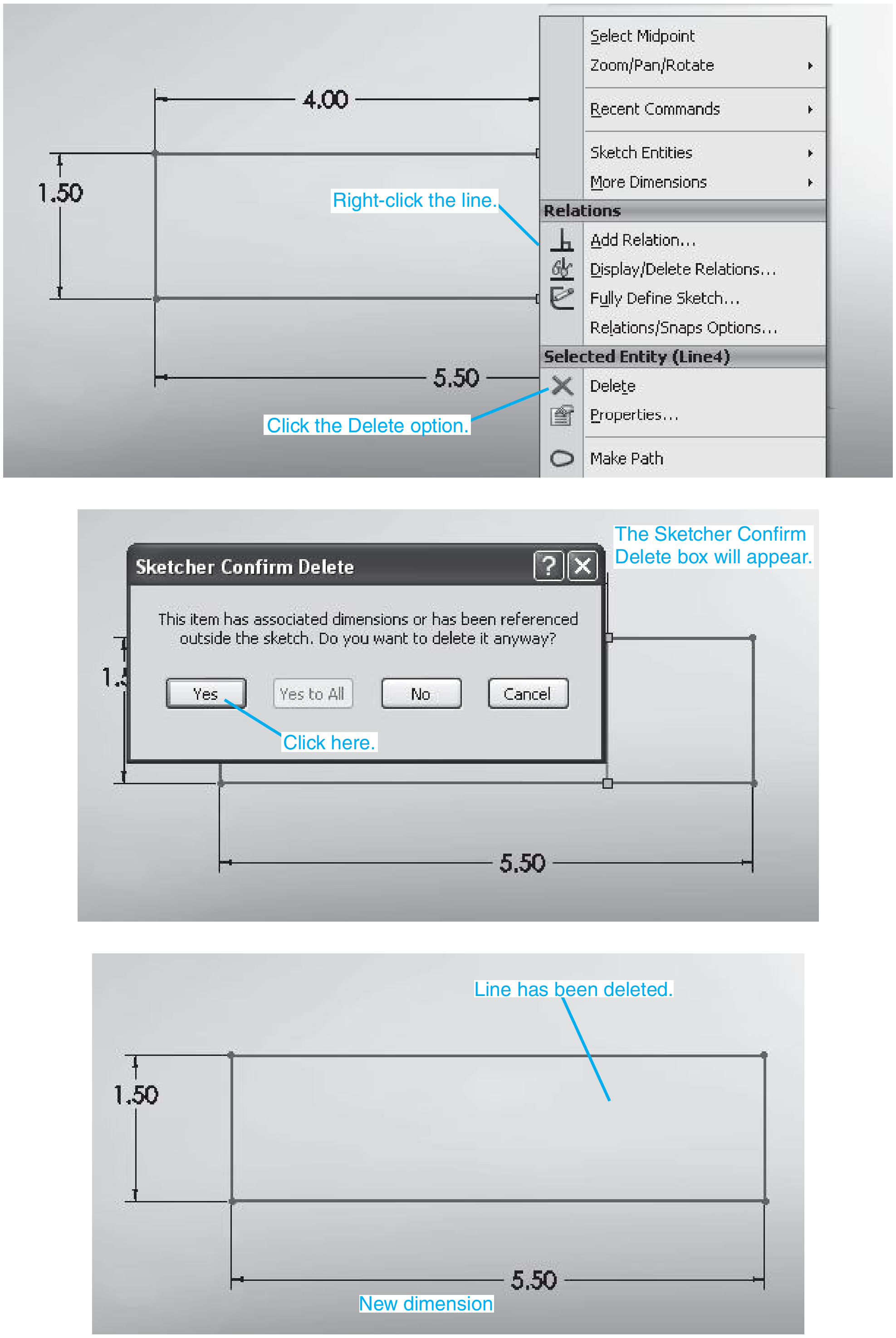


Figure 2-16 ***(continued)***

5. Click the top horizontal line in the rectangle.

   The extended line will appear automatically.

6. Click the lower horizontal line in the rectangle.
7. Right-click the mouse and click the **Select** option.
8. Right-click the vertical line located 4.00 in. from the left vertical line.

   A list of options will appear.

9. Click the **Delete** option.

The **Sketcher Confirm Delete** box will appear.

10. Select the **Yes** button.
11. Click the OK check mark.

2-11 SPLIT ENTITIES

The **Split Entities** tool is used to trim away internal segments of an existing entity or to split an entity into two or more new entities by specifying split points.

Figure 2-17 shows the rectangle created in the last section. Remove a 1.00-in. segment from the top horizontal line so that the left end of the segment is 2.00 in. from the left side of the rectangle. If you have already created a 1.50 × 5.50-in. rectangle, proceed to step 3.

1. Start a new **Part** document, click the **Sketch** group, and select the **Top Plane** option.
2. Draw the shape shown in Figure 2-17.
3. Access the **Split Entities** tool.

The **Split Entities** tool is accessed by clicking the **Tools** heading on the main menu, clicking **Sketch Tools,** then selecting the **Split Entities** tool.

4. Click two random points on the top horizontal line.

As the line is clicked, points will appear. Locate the points approximately **2.00 in.** and **3.00 in.** from the left vertical line of the rectangle.

5. Right-click the segment between these two points and select the **Delete** option.

The line segment will disappear.

6. Use the **Smart Dimension** tool to size and locate the opening in the line.
7. Click the OK check mark.

2-12 JOG LINE

The **Jog Line** tool is used to create a rectangular shape (jog) in a line.

Figure 2-18 shows an existing line. This section will show how to add jogs to the line.

1. Start a new **Part** document, click the **Sketch** group, and select the **Top Plane** option.
2. Draw a random horizontal line on the screen.
3. Access the **Jog Line** tool.

The **Jog Line** tool is located on the **Explode Sketch** toolbar. To access the **Explode Sketch** toolbar click the **View** heading at the top of the **Part** document screen, click **Toolbars,** and select the **Explode Sketch** tool.

4. Click a point on the line and move the cursor away from the point as shown.

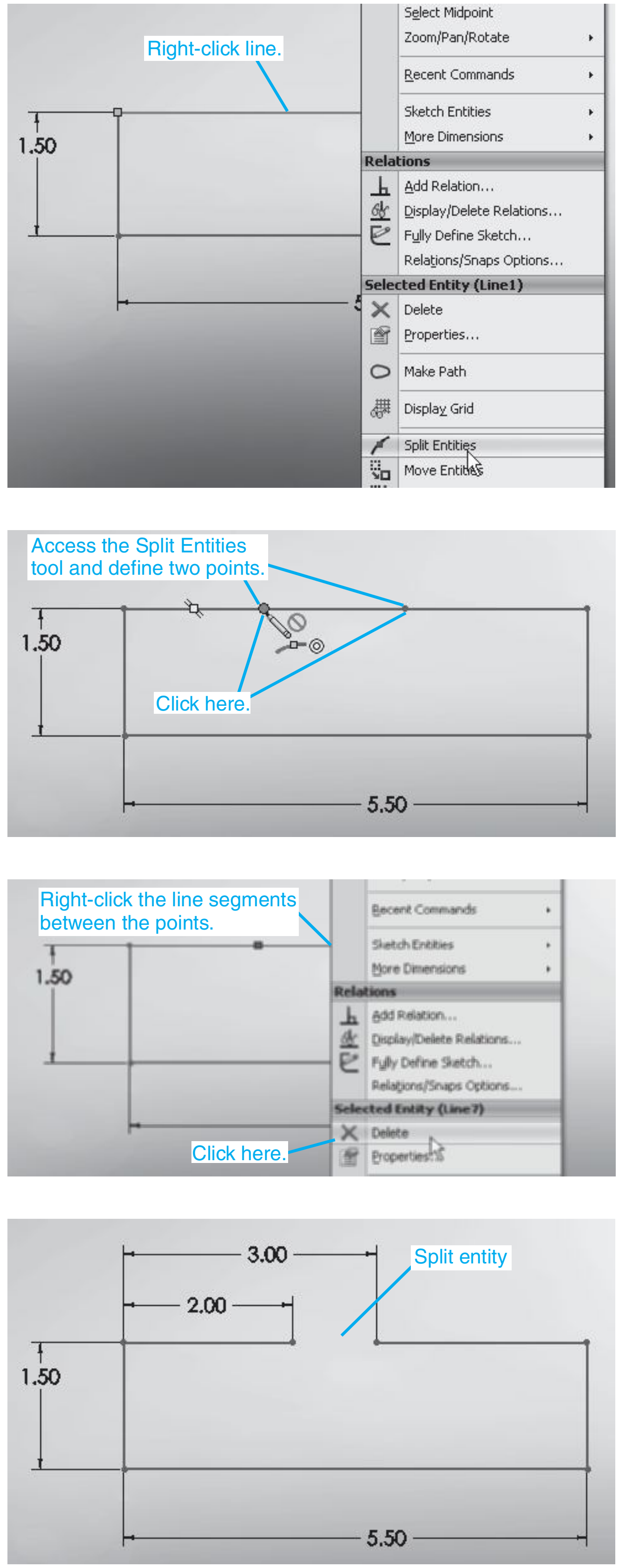


Figure 2-17

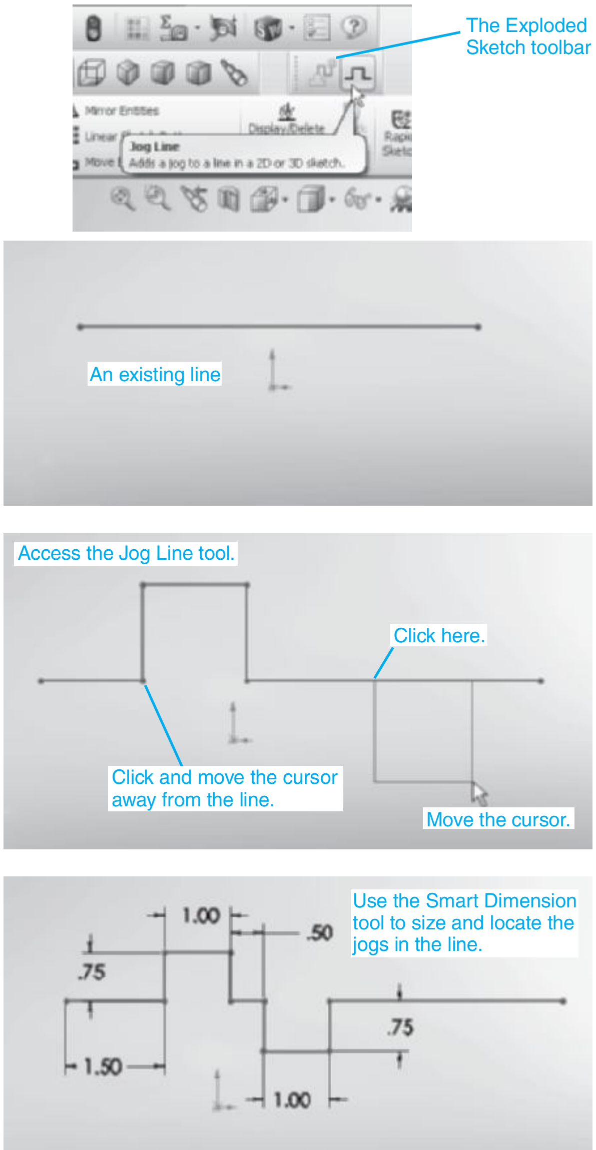


Figure 2-18

This will create a rectangular shape in the direction of the intended jog.

5. Click and move the cursor to create a second jog as shown.
6. Use the **Smart Dimension** tool to size and locate the jogs.
7. Click the OK check mark.

Sample Problem Using the Jog Line Tool

Figure 2-19 shows how the **Jog Line** tool can be used to help shape entities. Two cutouts are added to a 2.00 × 4.00-in. rectangle using the **Jog Line** tool.

Draw the shape shown in Figure 2-19.

1. Draw a **2.00** × **4.00-in.** rectangle in the top plane.
2. Use the **Jog Line** tool to create slots at each end of the object.
3. Use the **Smart Dimension** tool to size and locate the slots.

See Section 1-4.

4. Click the **Features** group on the **Command Manager** and select the **Extruded Boss/Base** tool.
5. Define the thickness by entering a value of **0.50in.** in the **Extrude Properties Manager.**
6. Use the **Fillet** tool in the **Features** group and add 0.50-in. radii fillets by clicking the vertical lines as shown.
7. Click the OK check mark to complete the object.

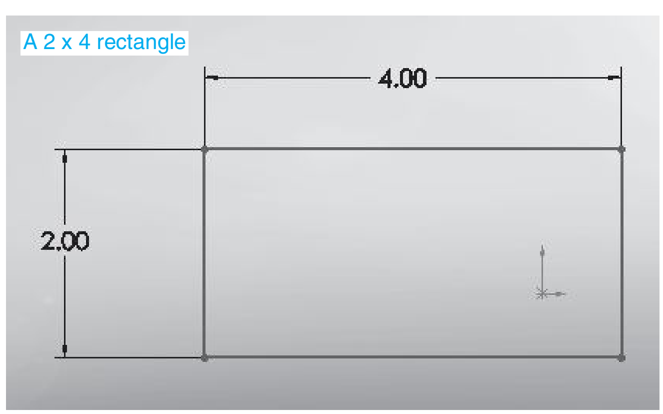


Figure 2-19

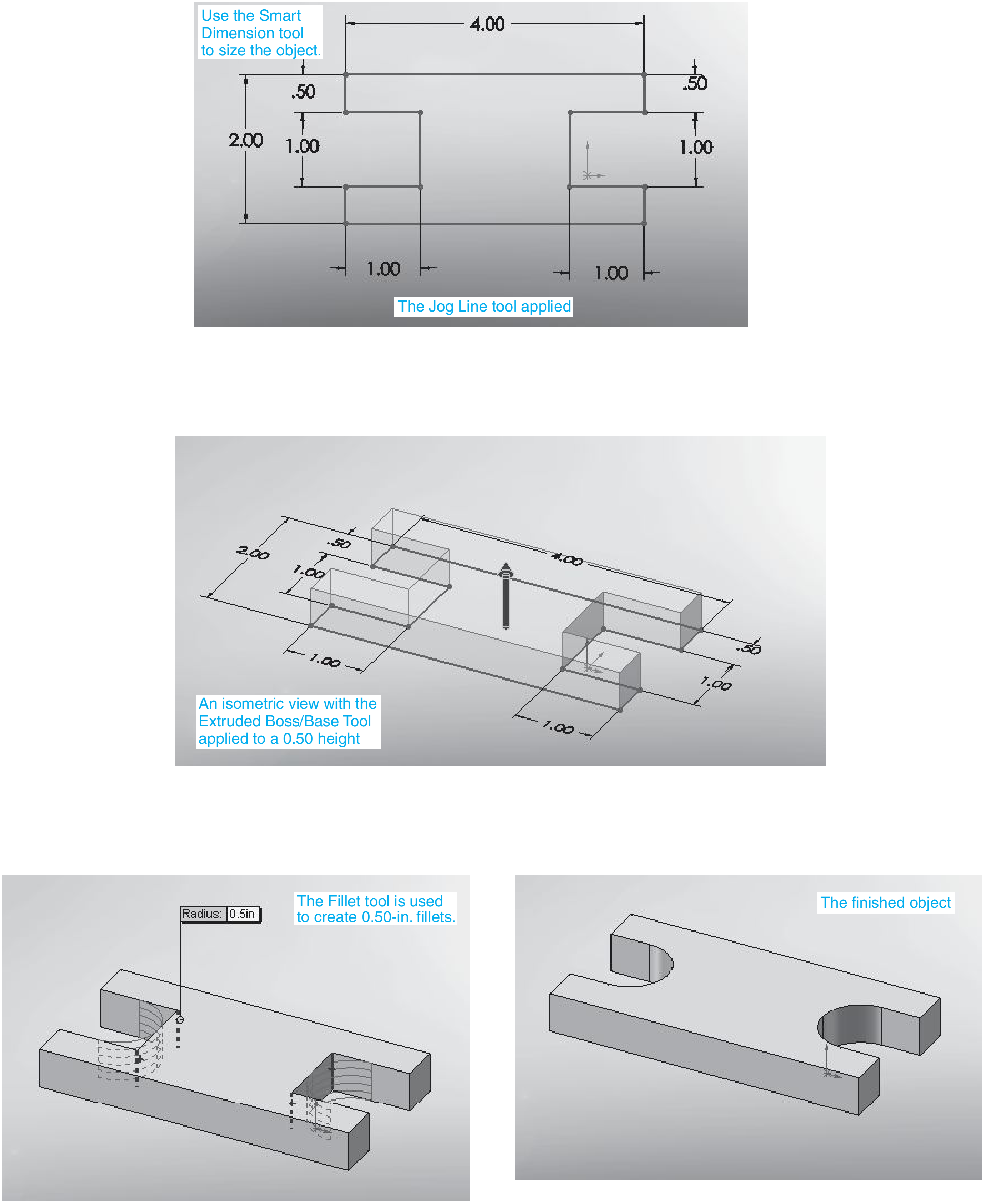


Figure 2-19 *(continued)*

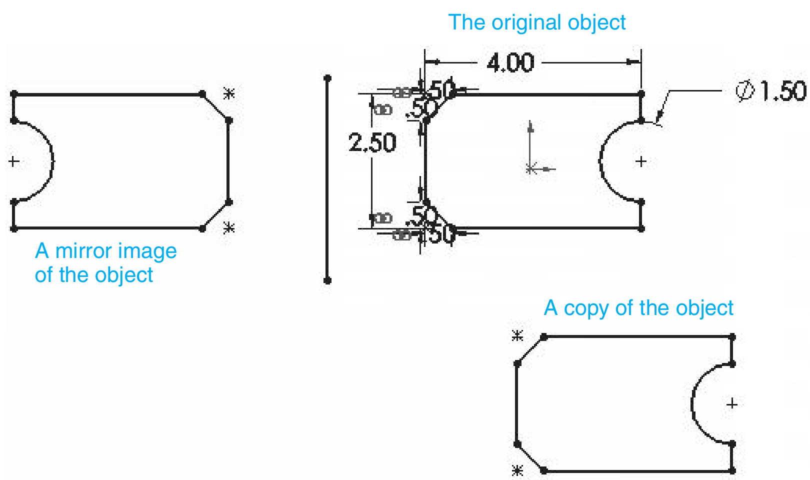


Figure 2-20

2-13 MIRROR ENTITIES

The **Mirror Entities** tool is used to create a mirror image of an entity. A mirror image is different from a copy of an image. Figure 2-20 shows both a mirror image and a copy of the same entity. Note the differences.

1. Start a new **Part** document, click the **Sketch** group, and select the **Top Plane** option.
2. Draw the shape and vertical line shown in Figure 2-21.
3. Access the **Mirror Entities** tool.

The **Mirror Entities** tool is located on the **Sketch** toolbar.

4. Window the object, but do not include the vertical line in the window.

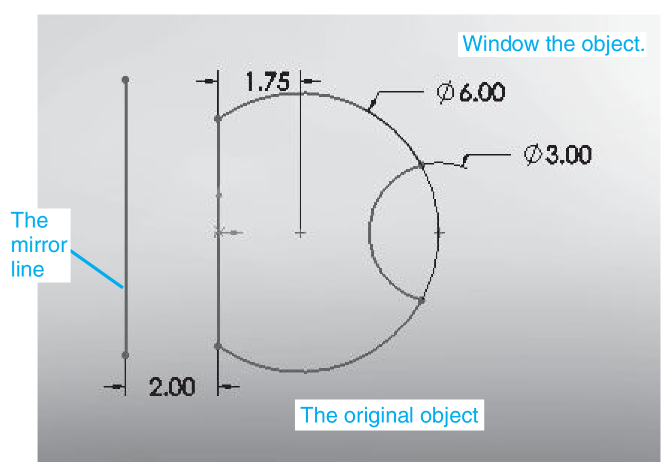
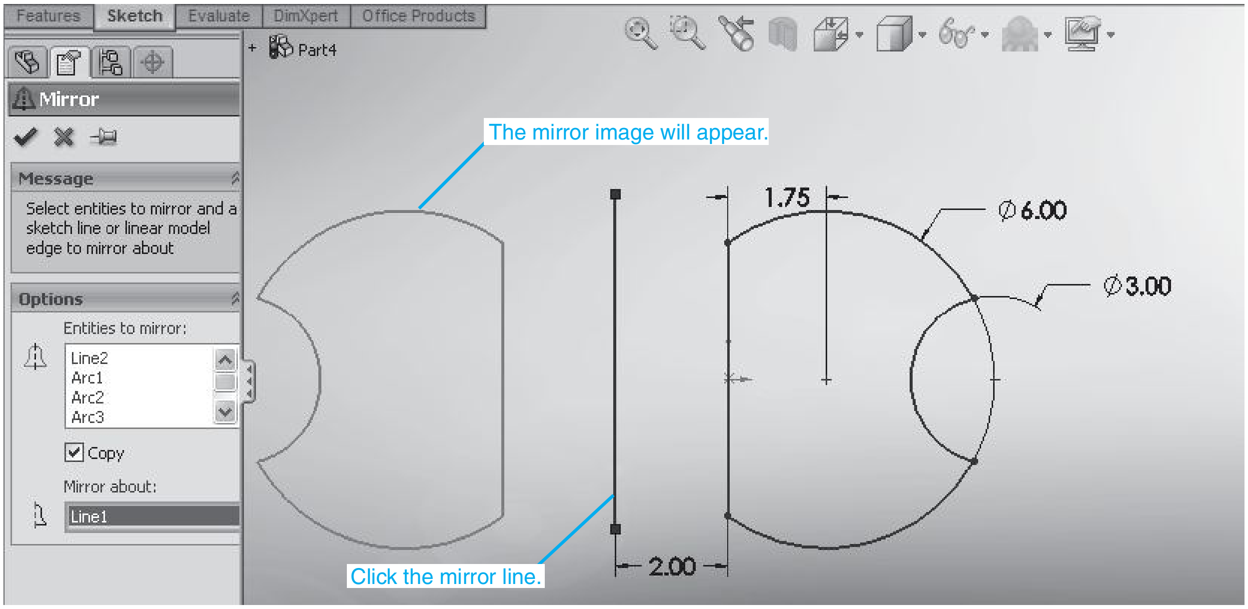
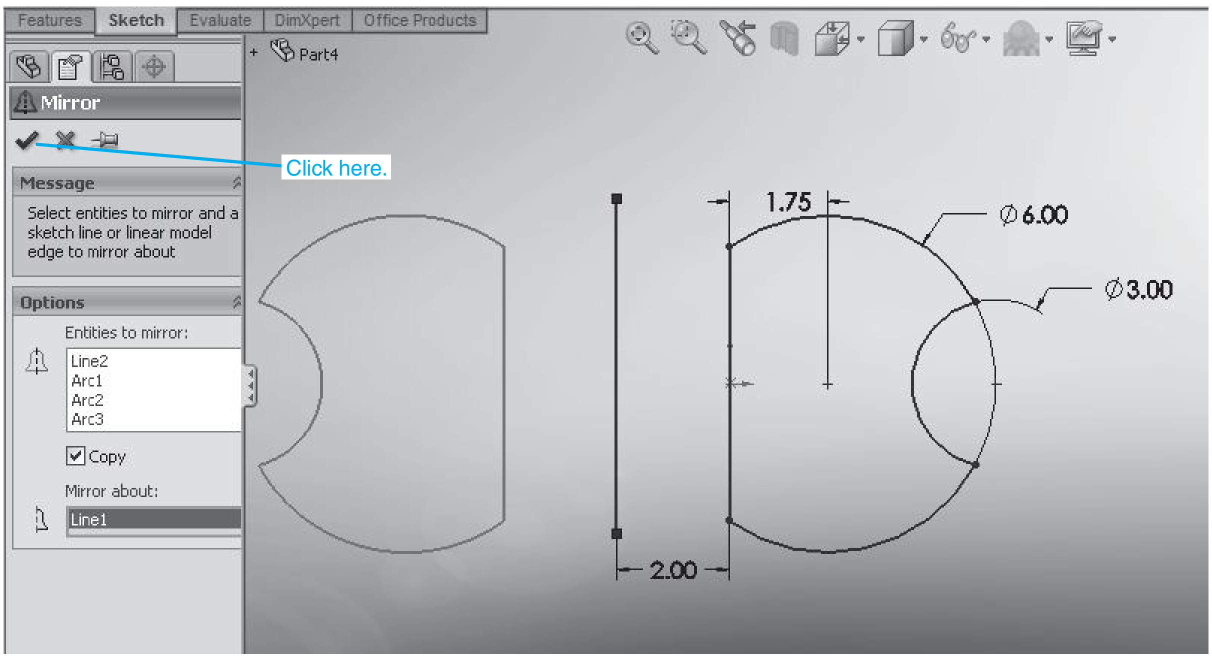


Figure 2-21





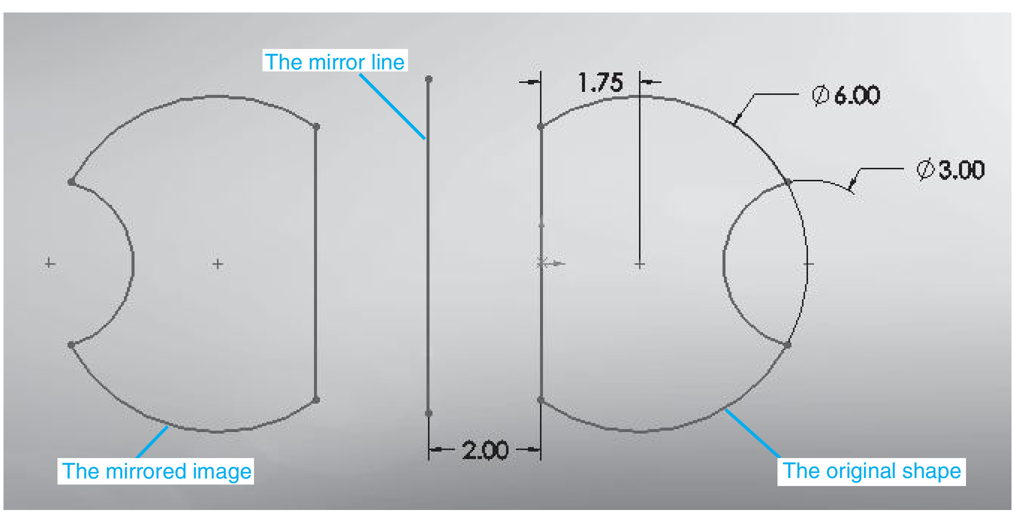


Figure 2-21 *(continued)*

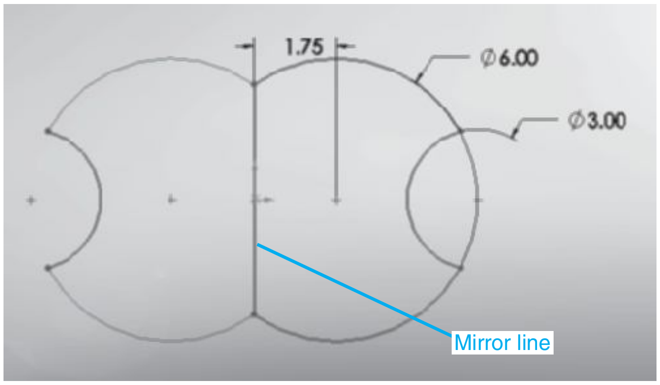


Figure 2-22

A listing of the selected entities will appear in the **Options** rollout box of the **Mirror Properties Manager.**

5. Click the **Mirror about:** box to highlight the selection of the mirror line and select the vertical line.

The vertical line will be used as the mirror line. The mirrored will appear as a preview in yellow. See Figure 2-21.

6. Click the OK check mark, or right-click the mouse and click **OK.**

Lines within an object can be used as mirror lines. Figure 2-22 shows the object in the previous figure mirrored about its own edge line.

2-14 MOVE ENTITIES

The **Move Entities** tool is used to relocate entities. See Figure 2-23.

1. Start a new **part** document, click the **Sketch** group, and select the **Top Plane** option.
2. Draw the shape shown in Figure 2-23.
3. Access the **Move Entities** tool located on the **Sketch** toolbar.

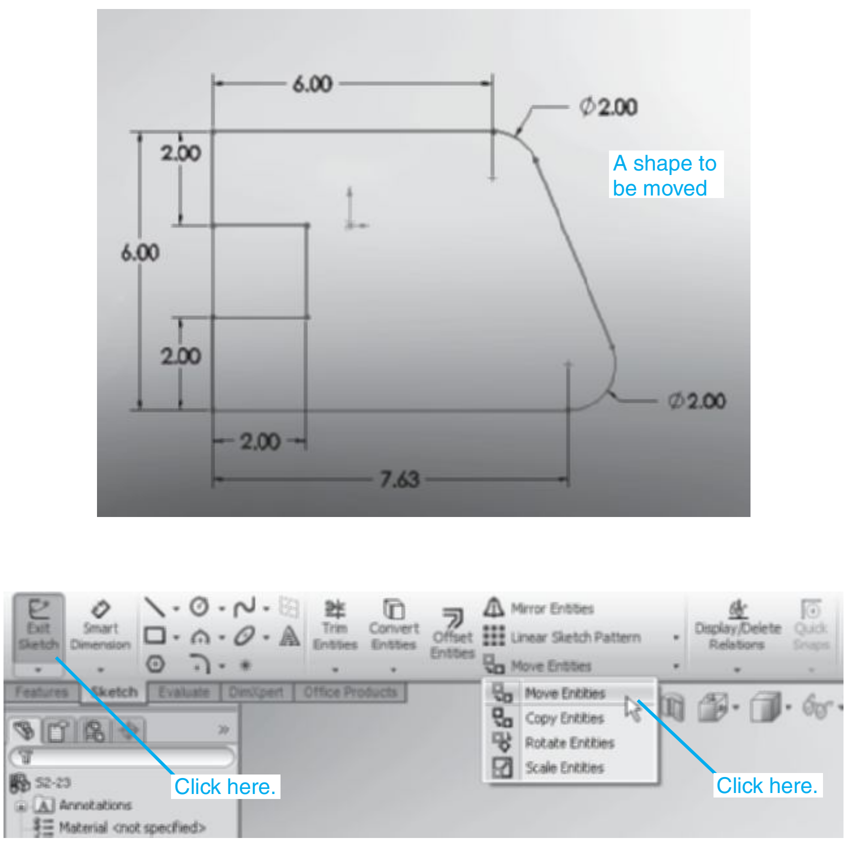


Figure 2-23

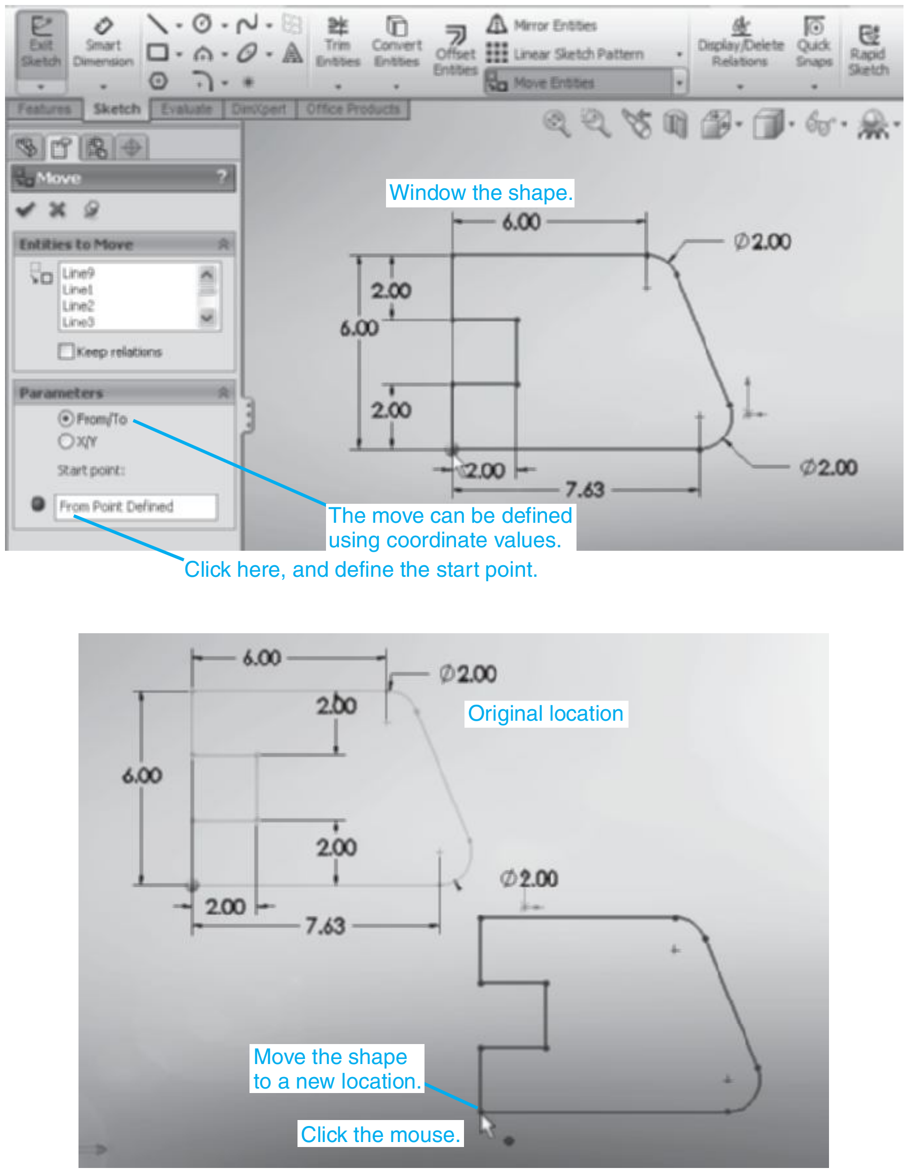


Figure 2-23 *(continued)*

The **Move Entities** tool is accessed by clicking the **Tools** heading in the main menu, clicking **Sketch Tools,** and selecting the **Move Entities** tool. See Figure 2-2.

4. Window the object to be moved.
5. Click the **Start point:** selection area in the **Parameters** rollout box.

All the sketch entities will be displayed in the rollout box.

6. Select a start point by clicking a point. This point will become the base point.

Any point on the screen can be used as a base point. In this example the lower left corner of the object was selected.

> ***Note:***
> The start point does not have to be on the object. Any point on the drawing screen can be used.

> ***Note:***
> A move can also be defined using X,Y coordinate values.

7. Move the base point to a new location and click the mouse.

Notice that the blue outline of the object in its original position will remain, and the moved object will appear in green.

8. Once the object is in the new location, right-click the mouse and select **OK.**

2-15 ROTATE ENTITIES

The **Rotate Entities** tool is used to change the orientation of a sketched entity. See Figure 2-24. Figure 2-24 shows the same object that was used in the previous section on the **Mirror Entities** tool.

1. Access the **Rotate Entities** tool.

The **Rotate Entities** tool is a flyout from the **Move Entities** tool on the **Sketch** toolbar.

2. Window the object.

A listing of all the sketch entities windowed will appear in the rollout box in the **Rotate Entities Properties Manager.**

3. Click the **Center of rotation:** box in the **Rotate Entities Properties Manager.**
4. Select a point of rotation or base point about which the sketch will rotate.

In this example the lower left corner of the object was selected.

5. Drag (click and hold down the left mouse button) the cursor away from object to rotate the object.

The object will rotate as the object is moved. The angle of rotation will appear in the **angle selection** box in the **Rotate Entities Properties Manager.**

6. Click the OK check mark.

Note:
The angle of rotation may also be defined by entering angular values in the **Parameters** box in the **Properties Manager** and clicking the check mark at the top of the **Rotate Entities Properties Manager** box.

2-16 COPY ENTITIES

The **Copy Entities** tool is used to create a duplicate of an entity. See Figure 2-25. The difference between the **Move Entities** tool and the **Copy Entities** tool is that the **Copy Entities** tool retains the original object in its original location and adds a new drawing of the object. The **Move Entities** tool relocates the original object.

1. Access the **Copy Entities** tool.

The **Copy Entities** tool is located on the **Sketch** toolbar.

2. Window the object.
3. Click the **Start point** box in the **Parameters** box to define the base point.
4. Select the base point by clicking on a point.

In this example the lower left corner of the objected was selected as the base point.

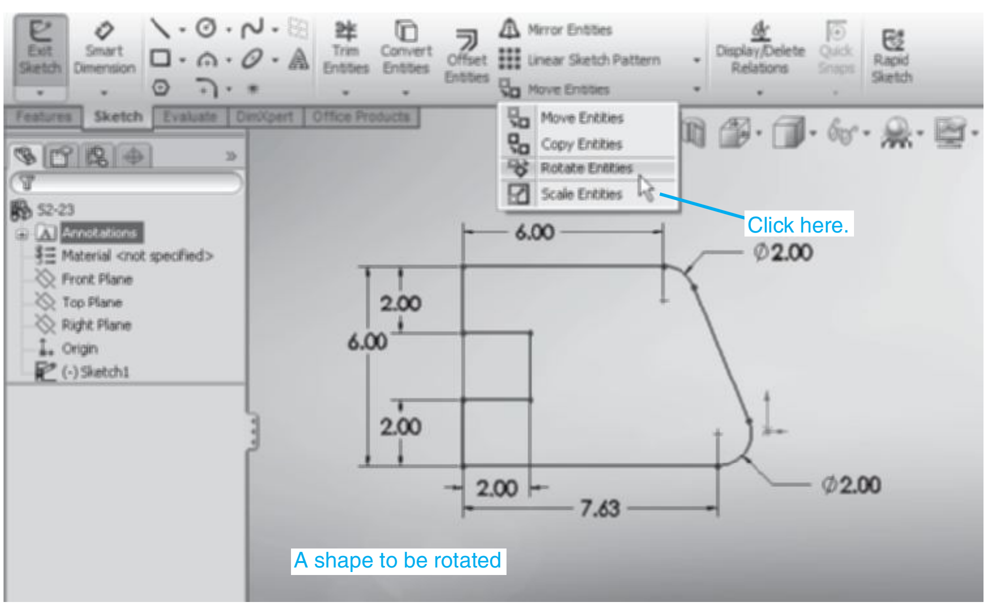


Figure 2-24

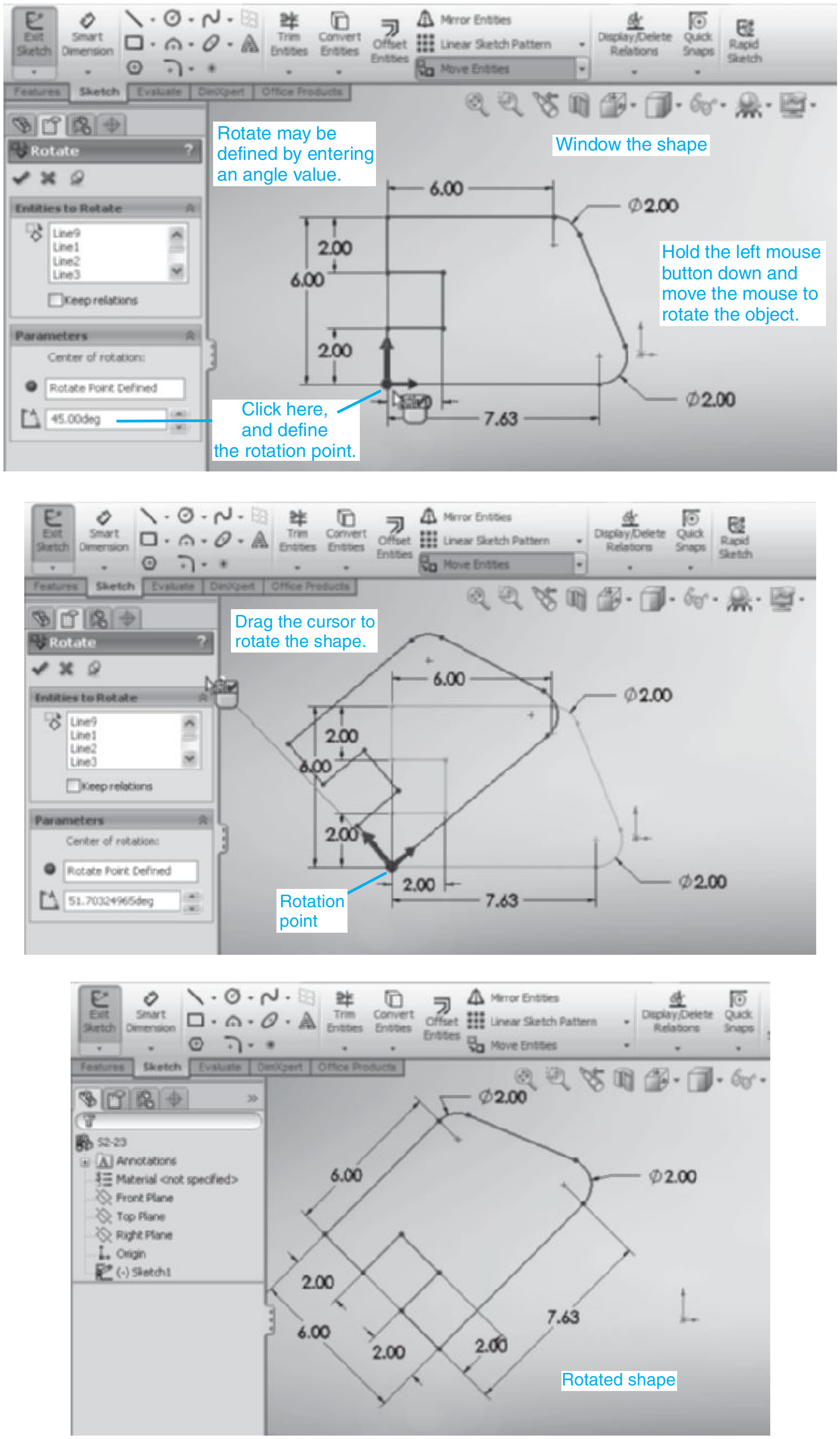


Figure 2-24 *(continued)*

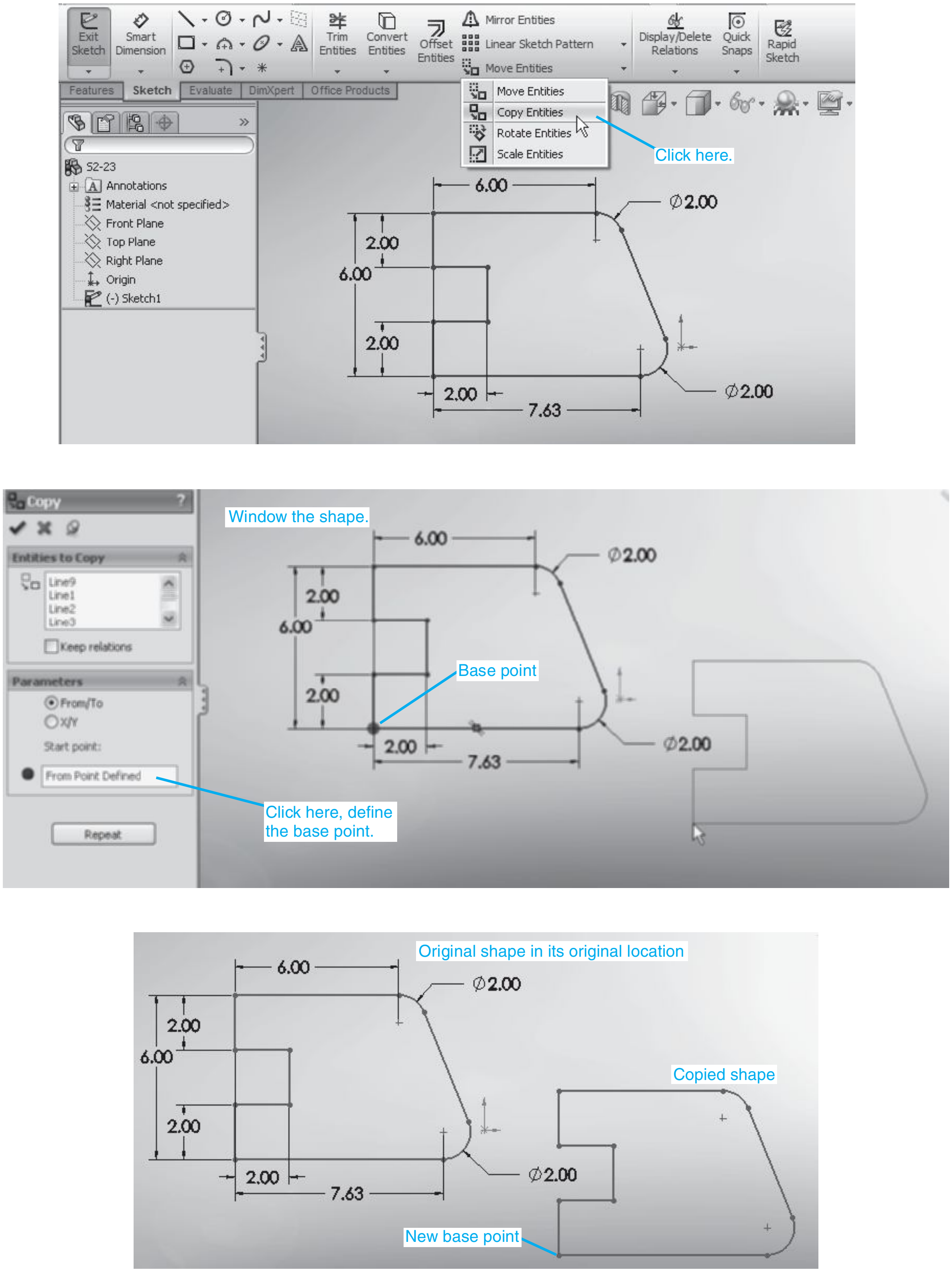


Figure 2-25

5. Move the cursor away from the object.

The preview of the object will be shown in yellow, and the original object will appear in green.

6. Determine the new location for the copy and click the mouse.
7. Click the OK check mark.

2-17 SCALE ENTITIES

The **Scale Entities** tool is used to change the overall size of an entity while maintaining the proportions of the original object. The **Scale Entities** tool includes a **Copy** option. If the **Copy** option is selected, when a scaled drawing is made, the original object will be retained. If the **Copy** option is off (no check mark) when a scaled drawing is made, the original object will be deleted.

Using Scale Entities with Copy On

See Figure 2-26.

1. Access the **Scale Entities** tool.

The **Scale Entities** tool is a flyout from the **Move Entities** tool located on the **Sketch** toolbar.

2. Window the object.
3. Define the scale factor and the number of copies to be made in the appropriate boxes in the **Scale Entities Properties Manager.**
4. Assure that the **Copy** option is on (there is a check mark in the **Copy** box).
5. Click on **Scale about:** in the **Parameters** box.
6. Select a scale point.

In this example the lower left corner of the object was selected.

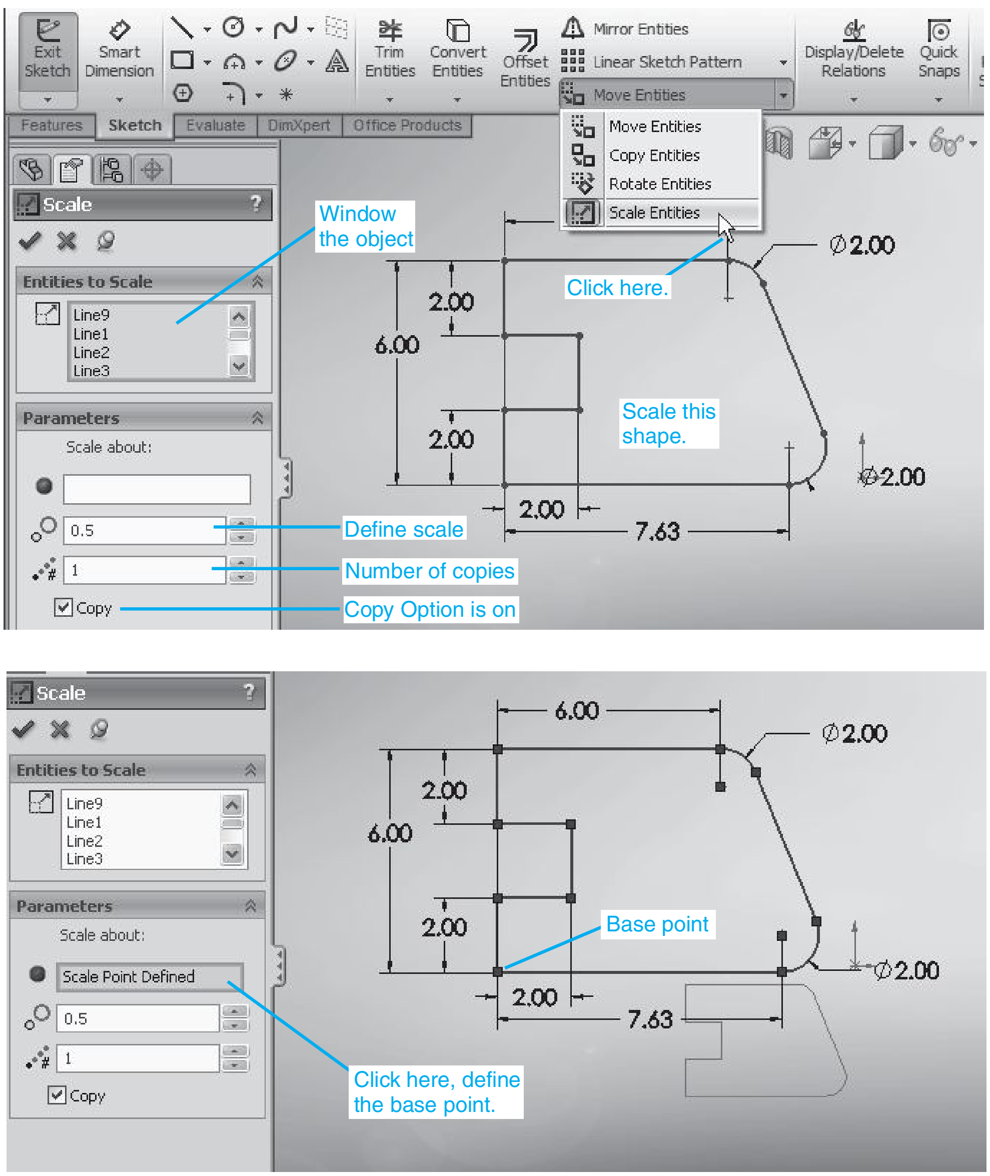


Figure 2-26

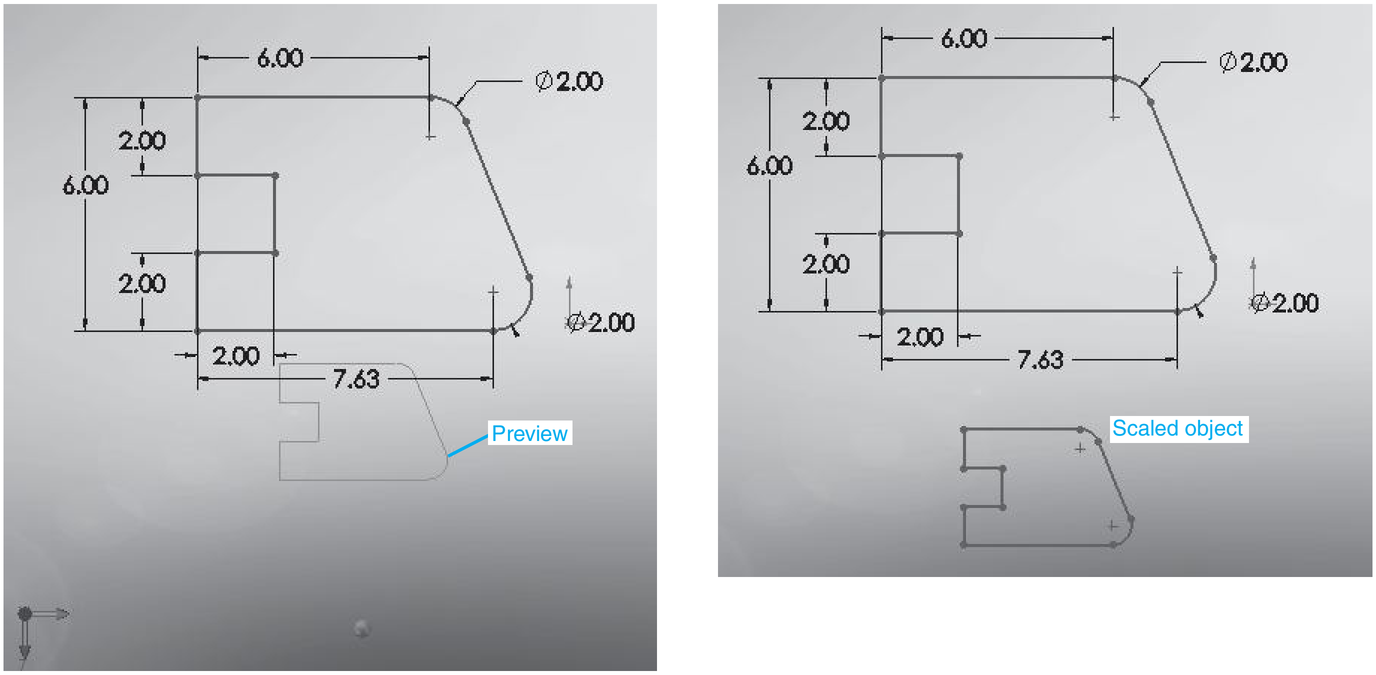


Figure 2-26 ***(continued)***

7. Right-click the mouse.

   The scaled object will appear.

8. Separate the original object and the scaled object.

   In this example the **Move Entities** tool was used to separate the original and the scaled objects.

9. Click the check mark.

2-18 CENTERLINE

Centerlines are used to help define the center of entities. The **Centerline** tool is a flyout from the **Line** tool on the **Sketch** toolbar.

See Figure 2-27.

1. Draw a **2.00 × 4.00** rectangle.

   Draw the rectangle offset from the origin as shown.

2. Draw a centerline diagonally across the rectangle.
3. Click the origin.
4. Hold down the **<Ctrl>** key, click the centerline, and release the **<Ctrl>** key.

   The **Select Entities** dialog box will appear.

5. Click the **Midpoint** option in the Add Relations box.

   The centerline will be centered on the origin.

2-19 LINEAR SKETCH PATTERN

The **Linear Sketch Pattern** tool is used to create patterns of sketched entites in the X and Y directions. Figure 2-28 shows a square. In this section a 3 × 3 linear pattern will be created from the square.

> ***Note:***
> A row is defined as a pattern in the horizontal direction, and a column is a pattern in the vertical direction.

1. Start a new **Part** document, click the **Sketch** group, and select the **Top Plane** option.
2. Draw a **1.00 × 1.00-in.** square as shown.
3. Access the **Linear Sketch Pattern** tool.

   The **Linear Sketch Pattern** tool is located on the **Sketch** toolbar.

4. Define the distance between the squares in the pattern.

The distance between the squares is measured from the lower left corner of the original square to the lower left corner of the next square. In this example a distance of 2.00 in. was specified.

5. Define the number of squares in a row of the pattern.

This example requires three squares in a row of the pattern (X-direction).

The **Linear Sketch Pattern** tool will automatically ask for the definition of the X-axis direction.

6. Define the X-axis by clicking the lower horizontal line of the square.

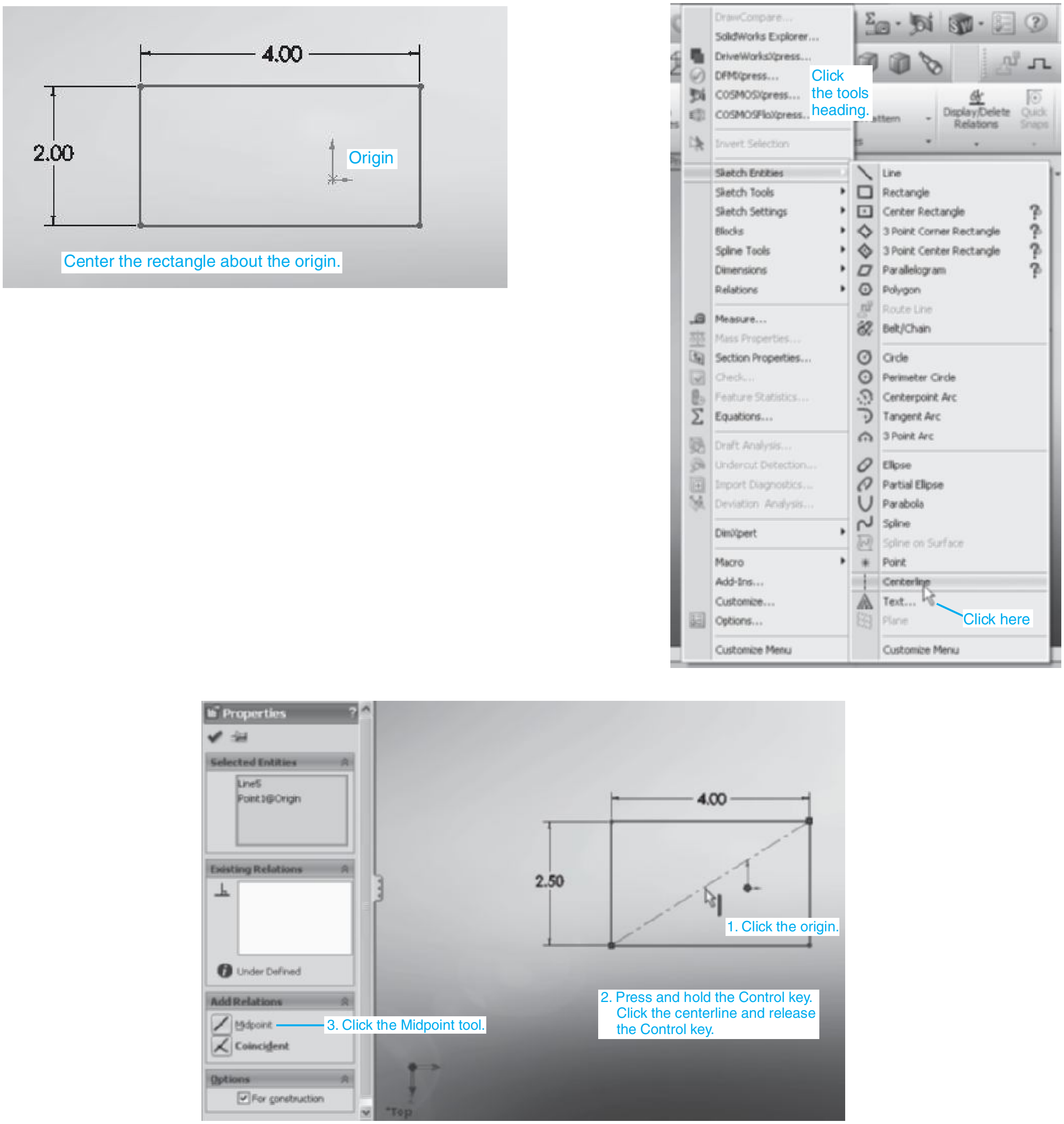


Figure 2-27

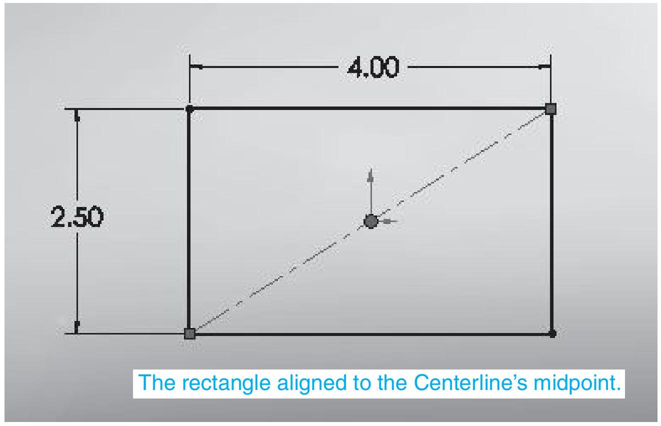


Figure 2-27 *(continued)*

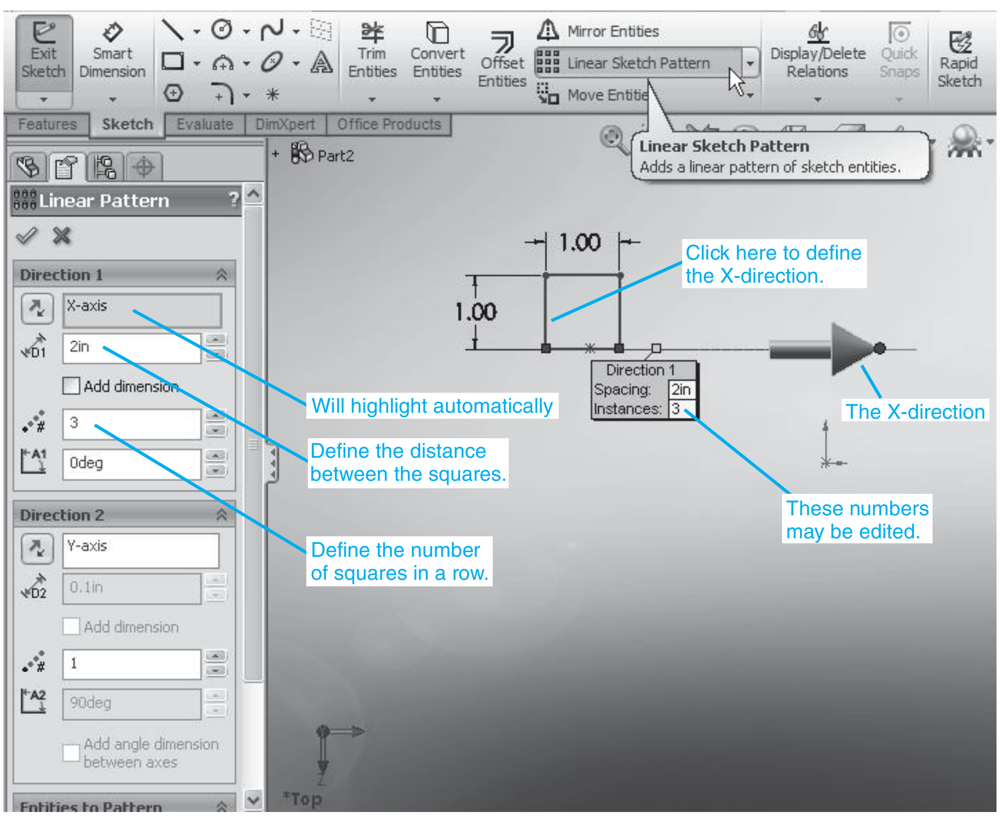
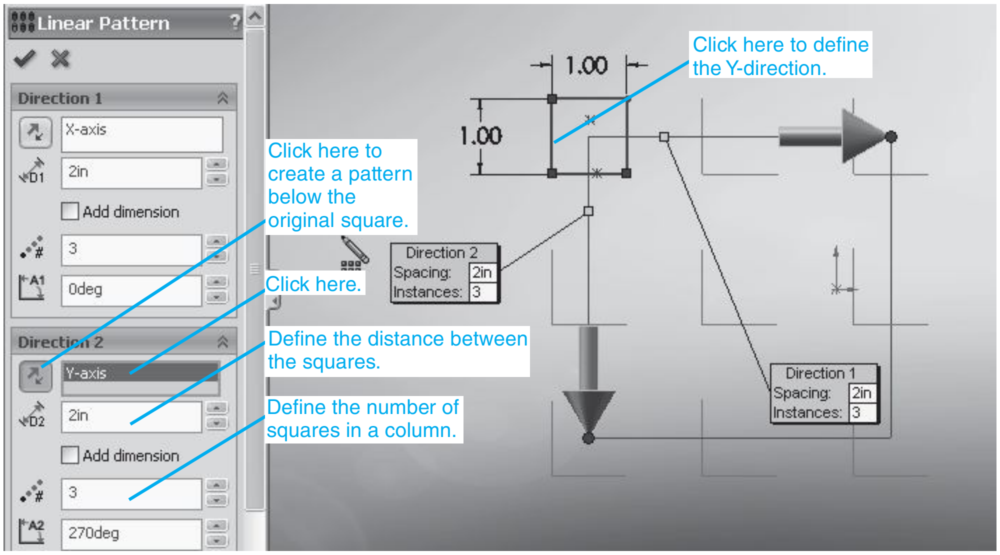
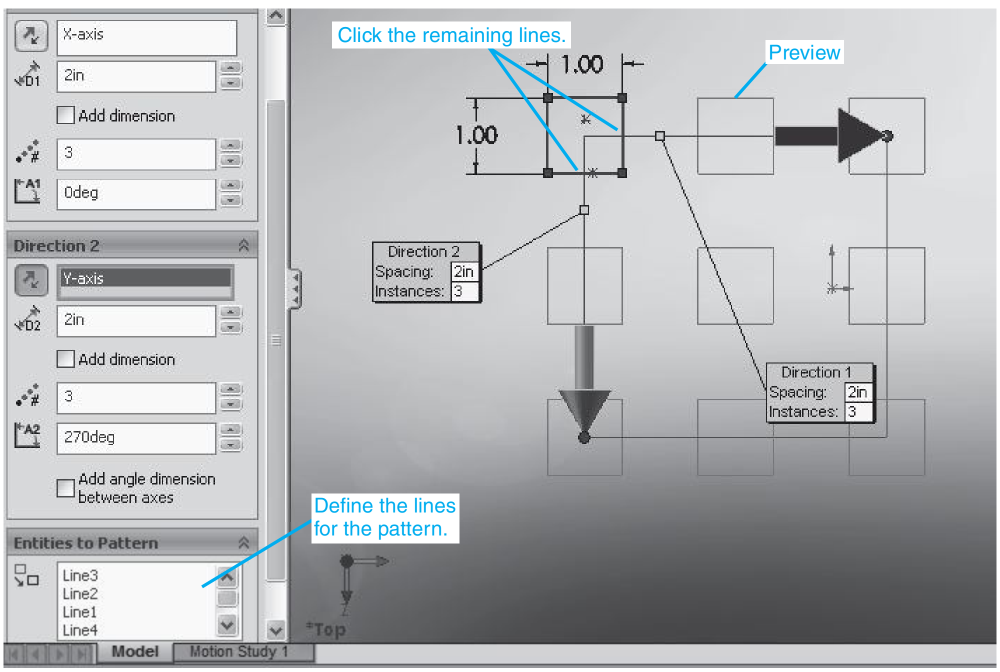


Figure 2-28





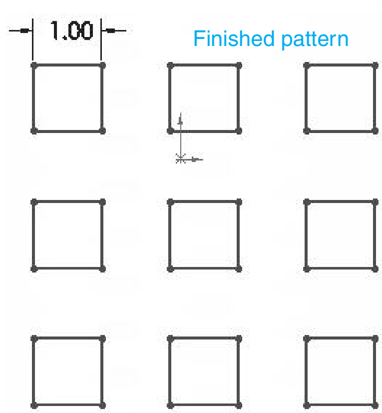


Figure 2-28 *(continued)*

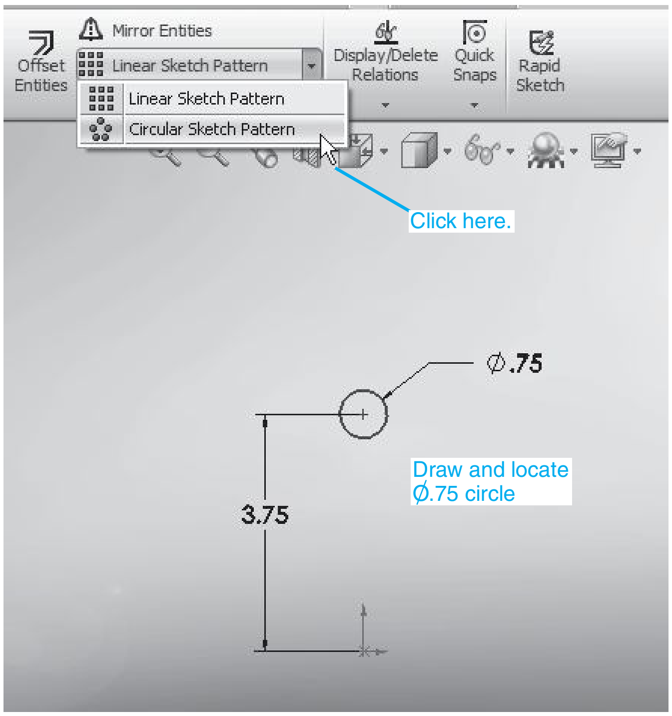


Figure 2-29

A large directional arrow will appear indicating the X-direction.

7. Click the Y-axis box
8. Click the direction arrow to the left of the Y-axis box.

Clicking this arrow box will define the Y-direction as downward toward the bottom of the screen.

9. Define the distance between the boxes and the number of boxes in the column.
10. Click the left vertical line of the square to define the Y-direction.

A larger arrow will appear defining the Y-direction.

11. Click the **Entities to Pattern** box in the **Linear Sketch Pattern Properties Manager** and click the remaining lines of the square.

> ***Note:***
> A preview of the pattern will appear on the screen.

12. Click the OK check mark.

> **TIP**
> If you edit the dimensions of a pattern formed using the **Linear Sketch Pattern** tool, SolidWorks will readjust the size of the sketch entity and pattern entities.

2-20 CIRCULAR SKETCH PATTERN

The **Circular Sketch Pattern** tool is used to create patterns about a center point such as a bolt circle. See Figure 2-29. In this example a Ø0.75-in. circle will be patterned about a center point located 3.75 in. from the center point of the circle.

1. Start a new **Part** document, click the **Sketch** group, and select the **Top Plane** option.
2. Draw a **Ø0.75-in**. circle **3.75 in.** from the screen's origin.

In this example the origin was used for convenience; any starting point can be used.

3. Access the **Circular Sketch Pattern** tool.

The **Circular Sketch Pattern** tool is a flyout from the **Linear Sketch Pattern** tool located on the **Sketch** toolbar.

4. Enter the number of entities in the pattern in the **Circular Sketch Pattern Properties Manager.**
5. Click the circle.

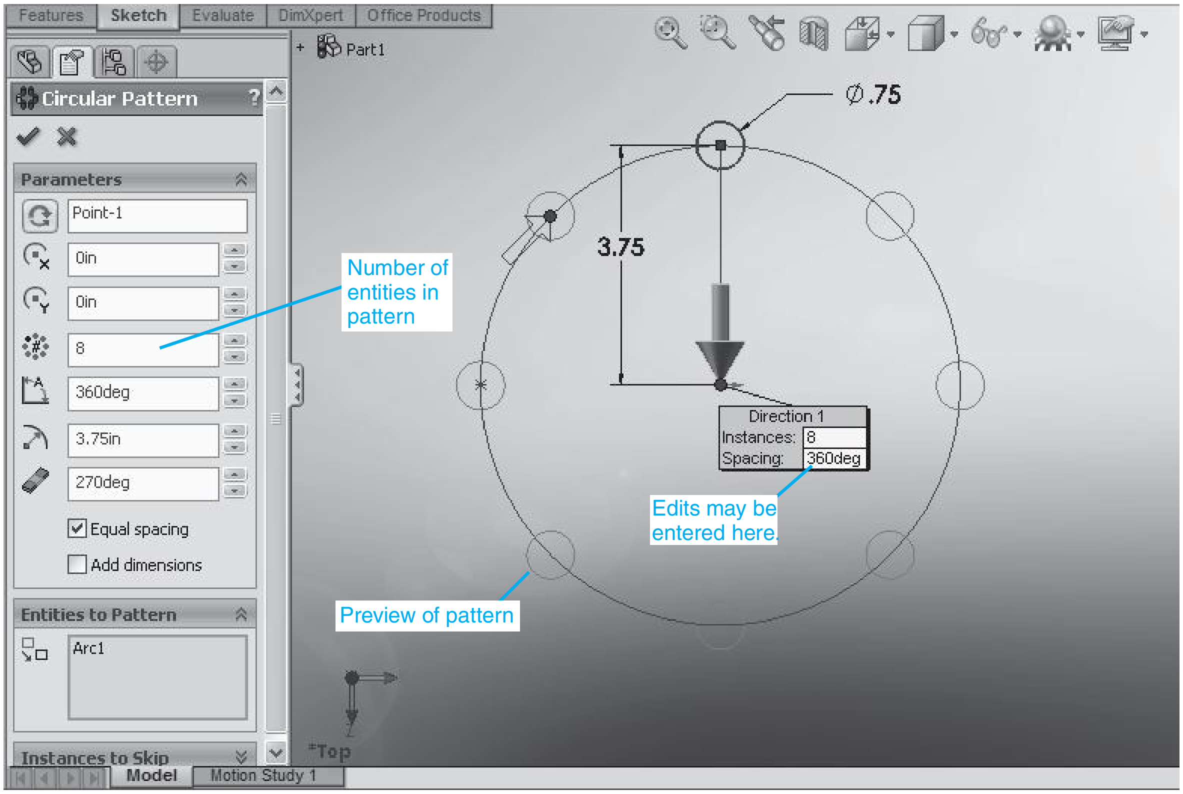
> ***Note:***
> The circle used in this example contains only one entity. More complex sketches would require all lines in the entity to be identified.

6. Click the OK check mark.

2-21 SAMPLE PROBLEM SP2-1

Any shape can be used to create a circular pattern. Figure 2-30 shows a slot shape located within a large circle. A circular pattern can be created using the slot shape.

1. Draw the circle and slot shown in Figure 2-30.
2. Access the **Circular Sketch Pattern** tool.



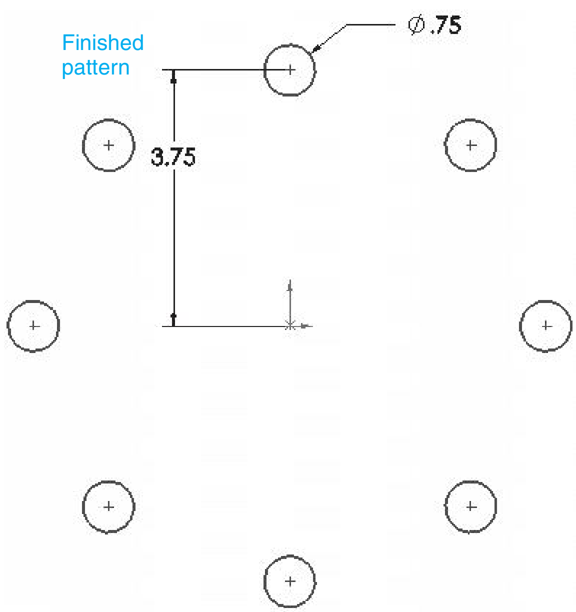
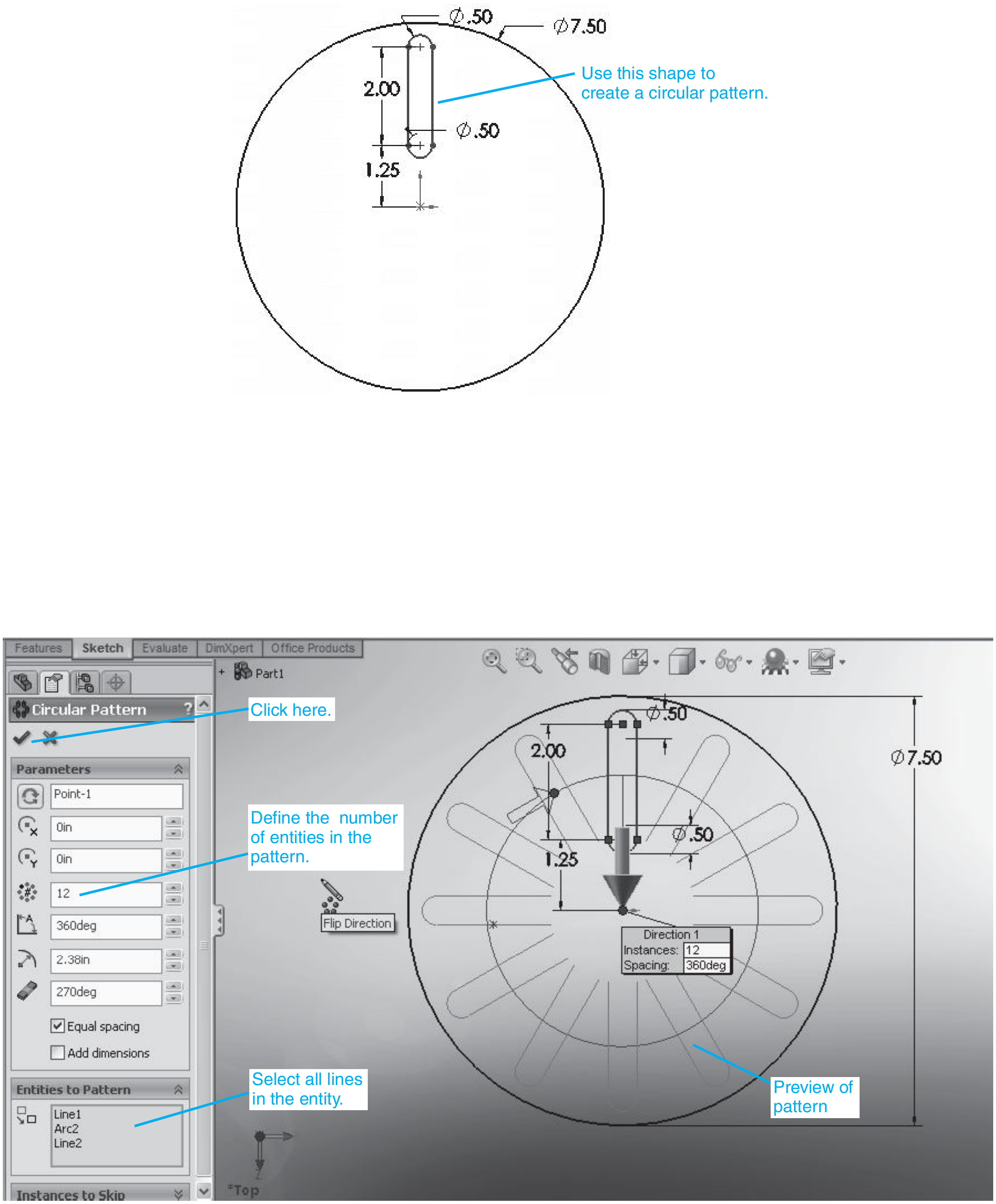
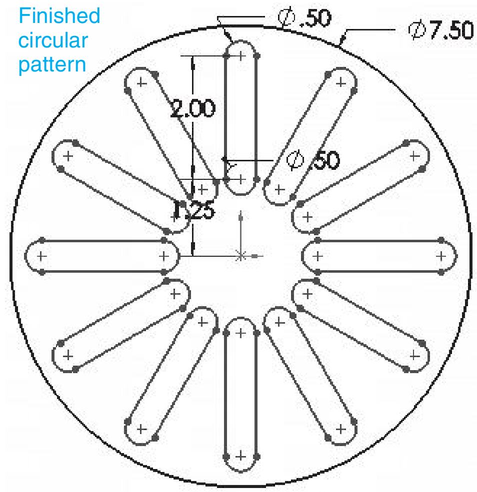


Figure 2-29 *(continued)*



Ø.50
Ø7.50
Use this shape to create a circular pattern.
2.00
Ø.50
1.25
Features
Sketch
Evaluate
DimXpert
Office Products
Part1
Circular Pattern
Click here.
Parameters
Point-1
0in
0in
12
360deg
2.38in
270deg
Equal spacing
Add dimensions
Define the number of entities in the pattern.
Flip Direction
Entities to Pattern
Line1
Arc2
Line2
Select all lines in the entity.
Instances to Skip
*Top
Direction 1
Instances: 12
Spacing: 360deg
Preview of pattern

Figure 2-30



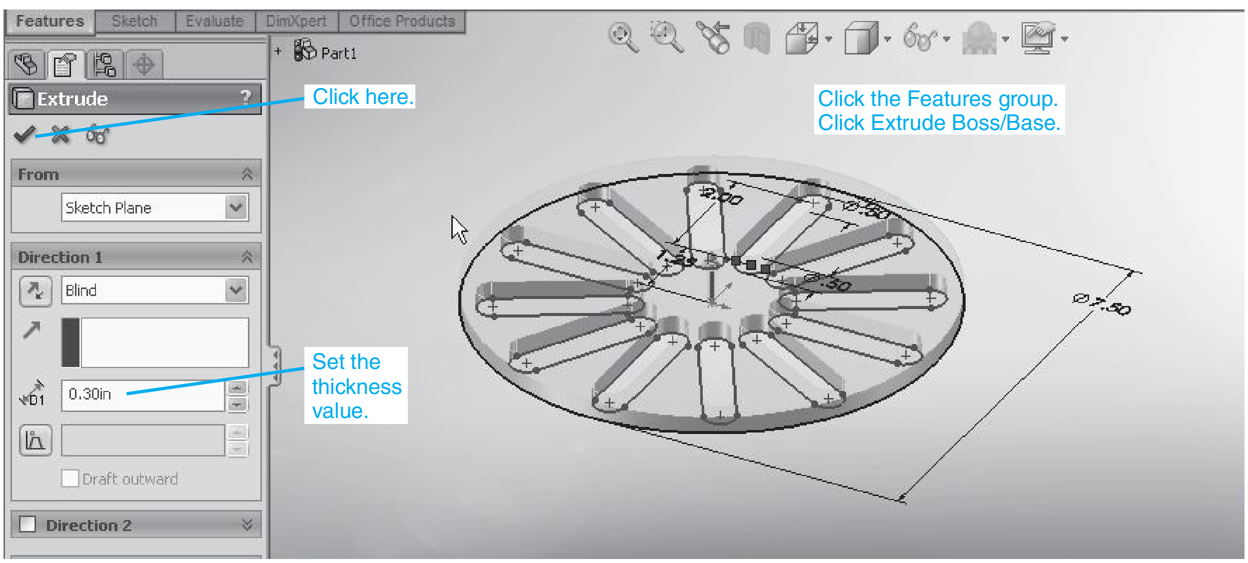


Figure 2-30 *(continued)*

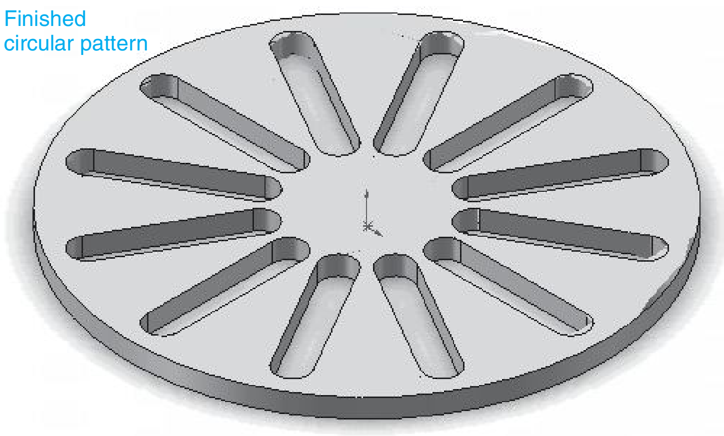


Figure 2-30 ***(continued)***

See Section 2-20.

3. Click the **Entities to Pattern** box and enter all the entities that define the slot.
4. Define the number of entities in the pattern as **12.**
5. Click a point on the drawing screen to create a yellow preview of the pattern.
6. Click the check mark.
7. Click the **Features** group, then click the **Extruded Boss/Base** tool.
8. Set the thickness of the extrusion for **0.30.**
9. Click the OK check mark.

> **TIP**
>
> Do not overdefine the shapes and contours to be included in a circular pattern, as this may make the tool inoperable.

2-22 SAMPLE PROBLEM SP2-2

Figure 2-31 shows a shape that includes fillets. The shape is initially drawn square, that is, with 90° corners, then the fillets are added.

1. Start a new **Part** document and sketch the T-shape.
2. Use the **Smart Dimension** tool and size the part.
3. Click the **Fillet** tool, set the **Fillet Parameters** for **0.25in** and add the external fillets.
4. Sketch, locate, and size the **1.00 × 1.50** rectangle as shown.
5. Use the **Fillet** tool and add **0.25** fillets to the rectangle.
6. Use the **Circle** tool and draw two **Ø0.50** circles **2.00** apart, **1.50** from the left edge as shown.
7. Use the **Line** tool and draw two horizontal lines tangent to the two circles.
8. Use the **Trim** tool and delete the internal portions of the cirlces, creating a slot as shown.
9. Access the **Features** tools and click the **Extruded Boss/Base** tool.
10. Extrude the T-shape to a thickness of **0.375in.**
11. Click the OK check mark.

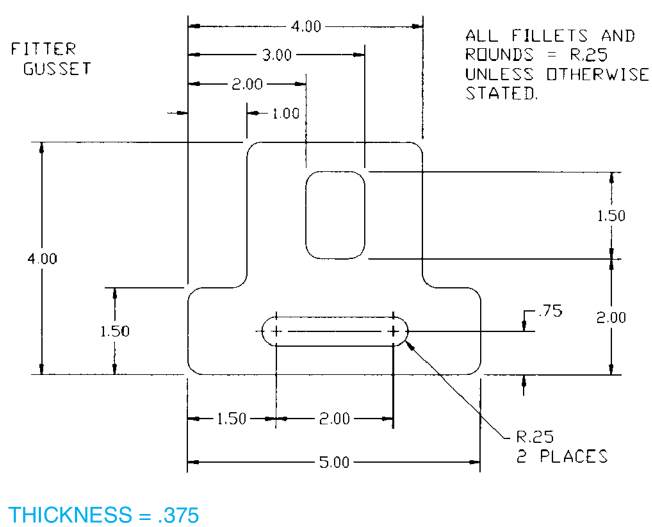
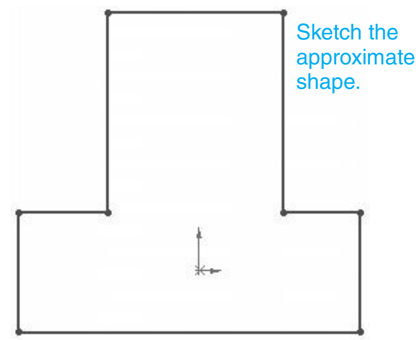
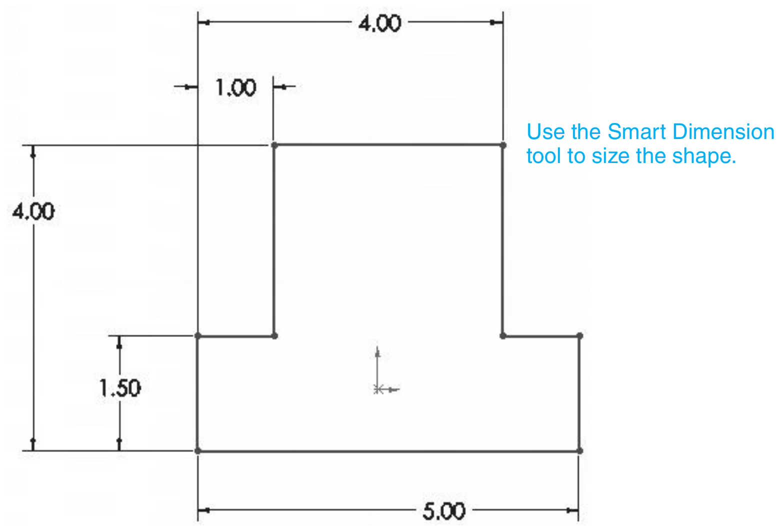


Figure 2-31





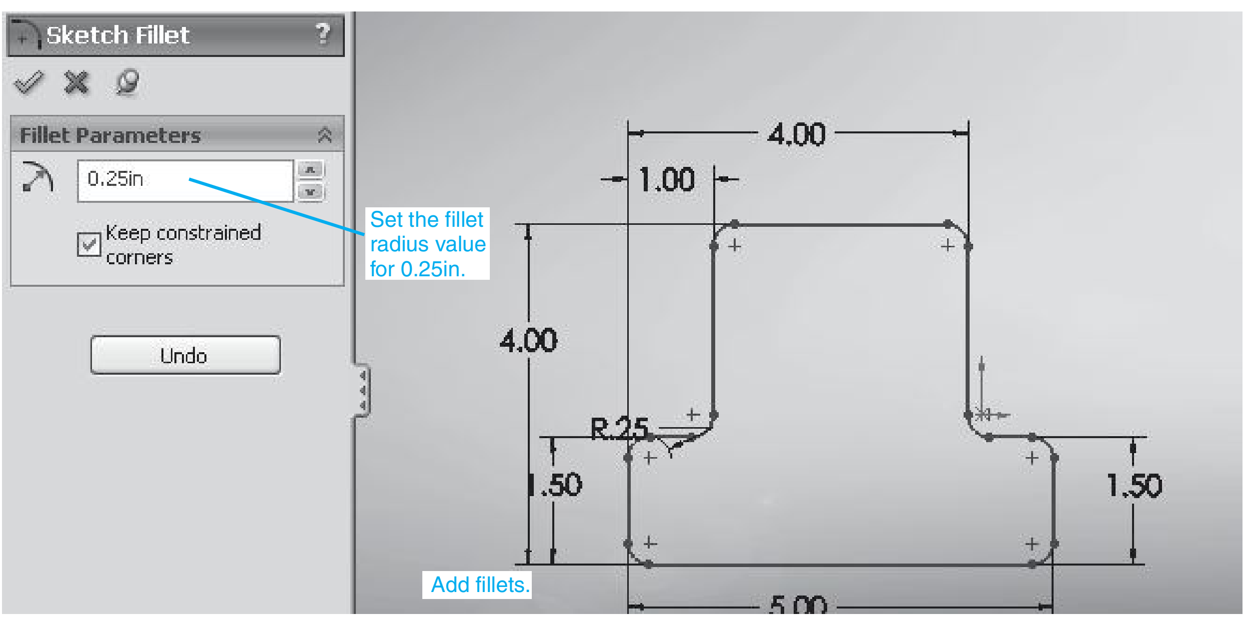


Figure 2-31 ***(continued)***

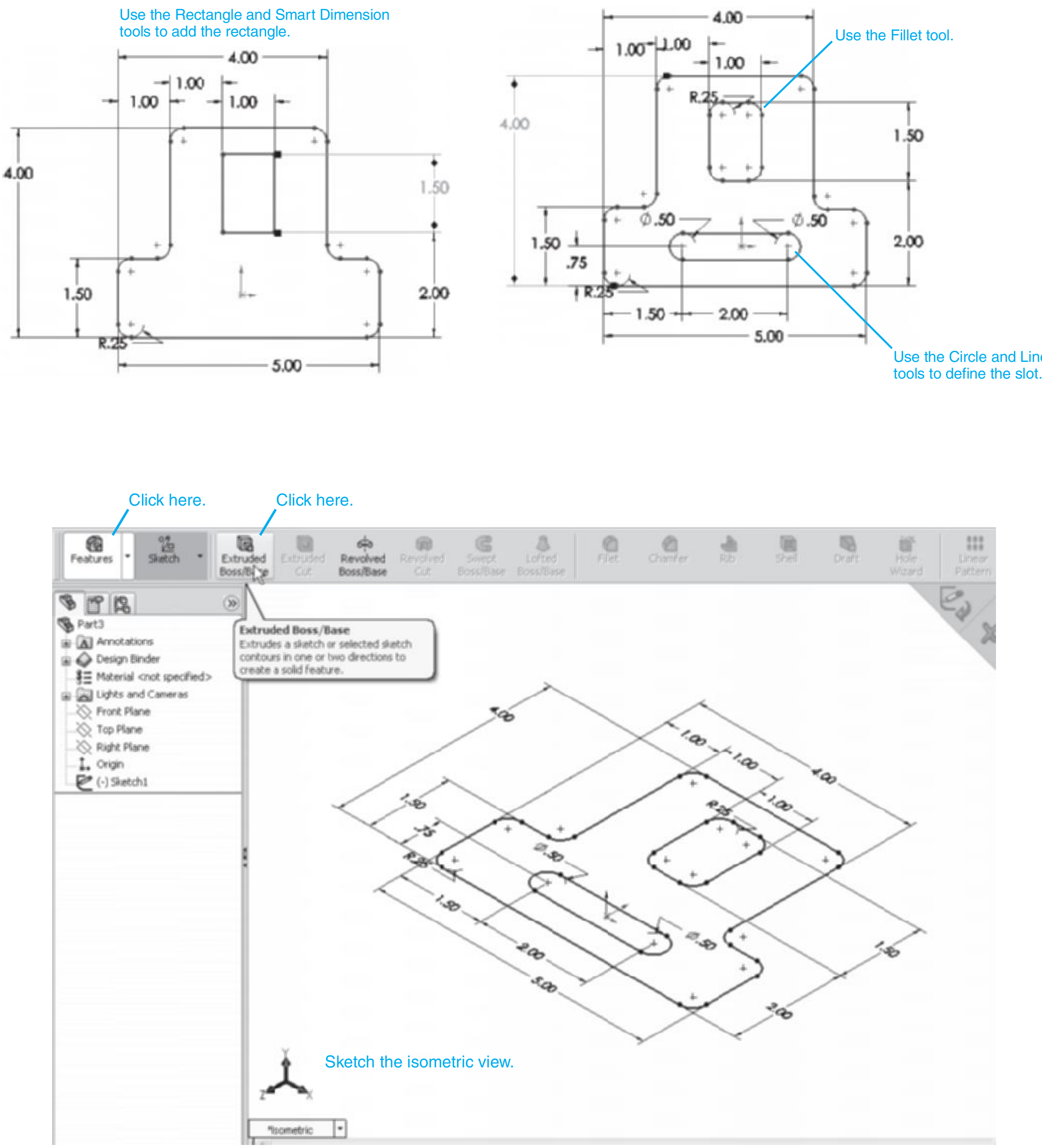


Figure 2-31 *(continued)*

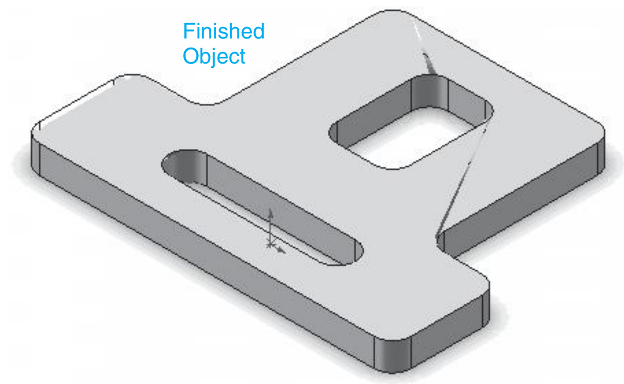


Figure 2-31 ***(continued)***

2-23 TEXT

The **Text** tool is used to add text to a **part** document. See Figure 2-32.

1. Create a new **Part** document.
2. Click the **Tools** heading, click **Sketch Entities,** and click the **Text** tool. See Figure 2-1.
3. Click the **Text** box in the **Text Properties Manager** and type the text required.

The text will appear on the screen.

To Change the Font and Size of Text

1. Make sure that the **Use document font** box in the **Text Properties Manager** is off; that is, there is no check mark in the box.
2. Click the **Font** box.

The **Choose Font** dialog box will appear.

3. Select the desired font and height.

It is best to avoid fonts that are too stylistic. Simple block-type letters are best for engineering drawings.

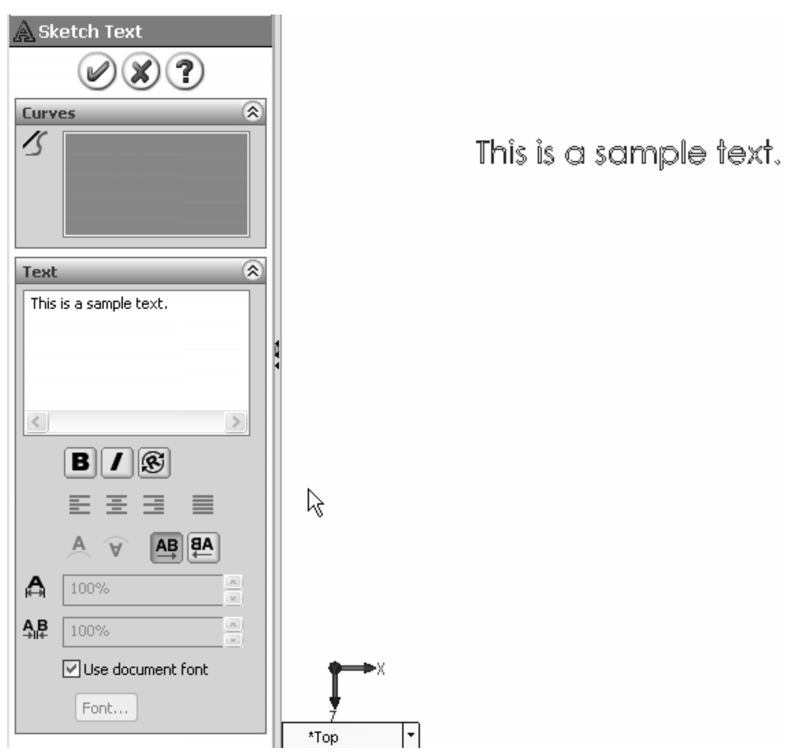


Figure 2-32

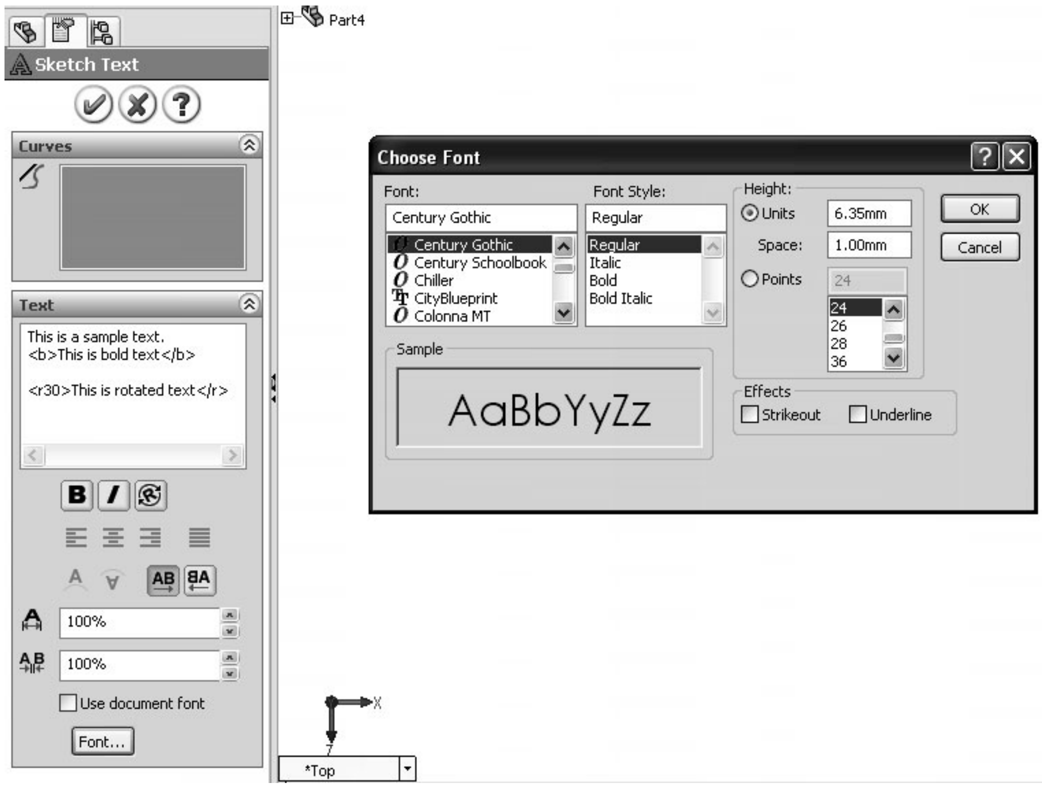


Figure 2-33

> ***Note:***
> The default font for SolidWorks is Century Gothic.

2-24 PROJECTS

Project 2-1:

Redraw the objects in Figures P2-1 through P-24 using the given dimensions. Create solid models of the objects using the specified thicknesses.

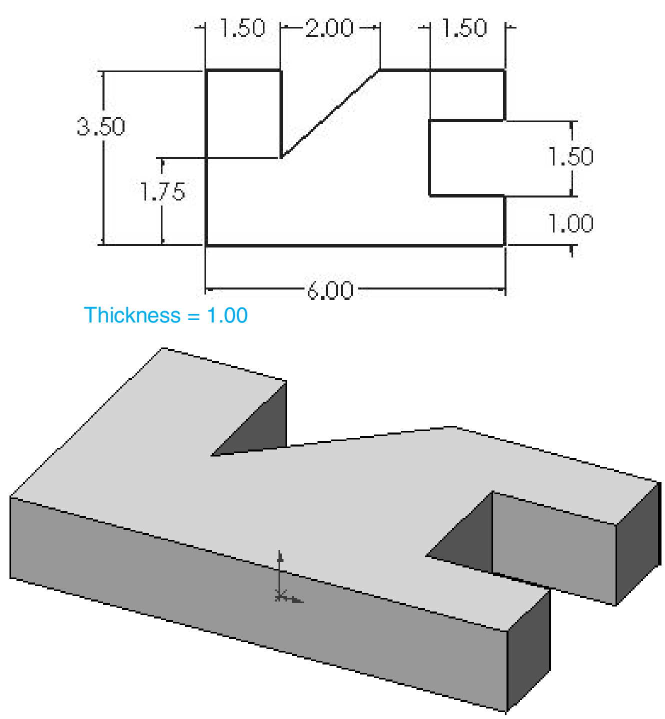


Figure P2-1 INCHES

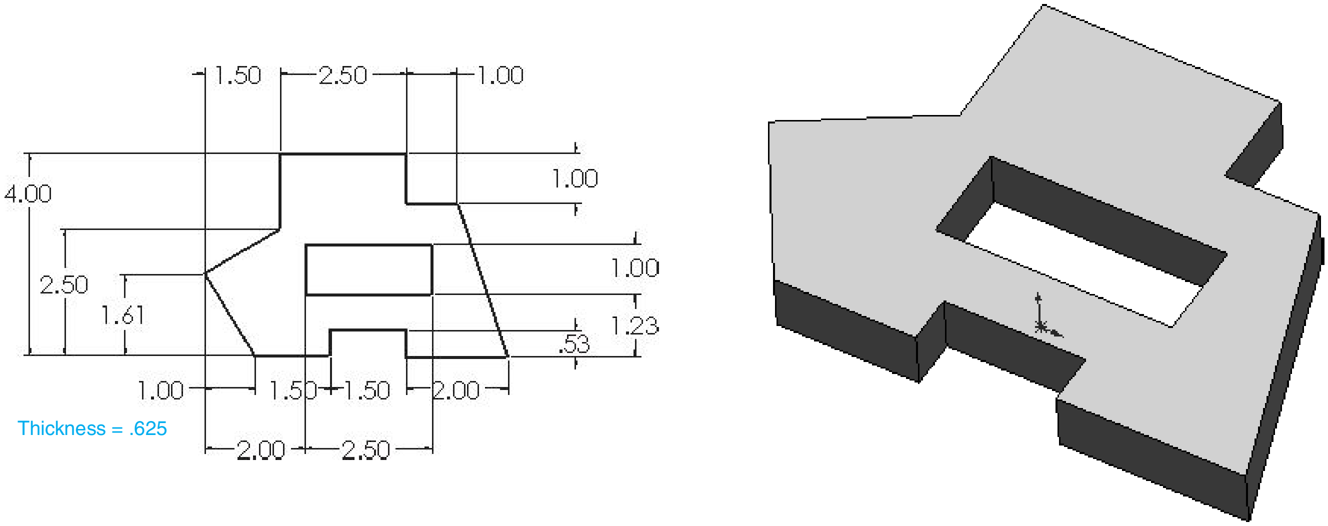


Figure P2-2 INCHES

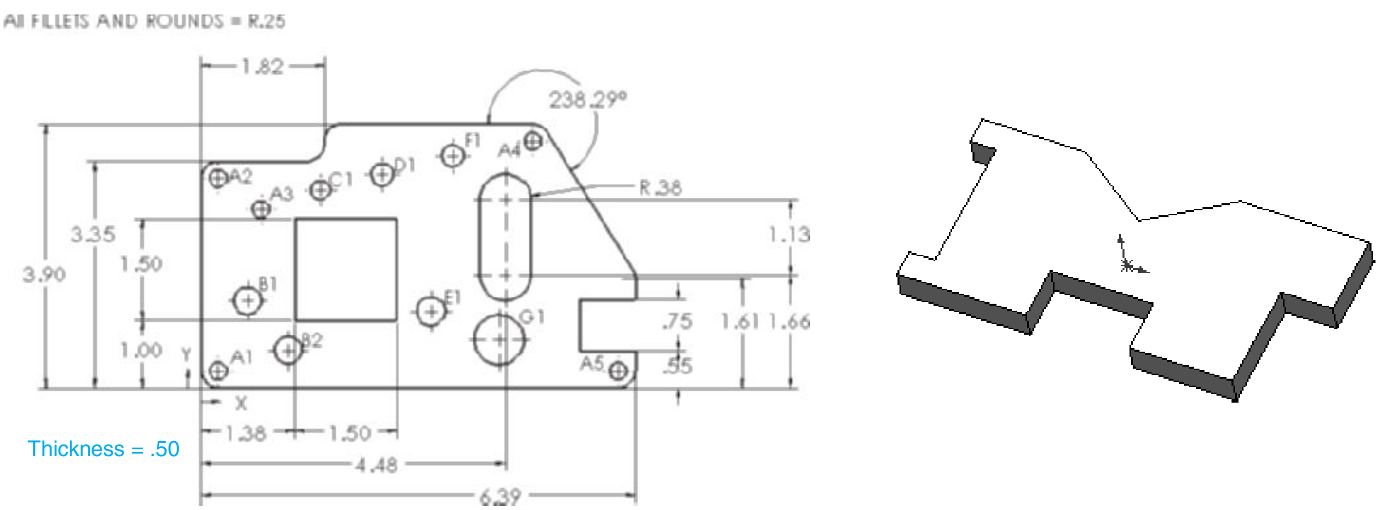


Figure P2-3 MILLIMETERS

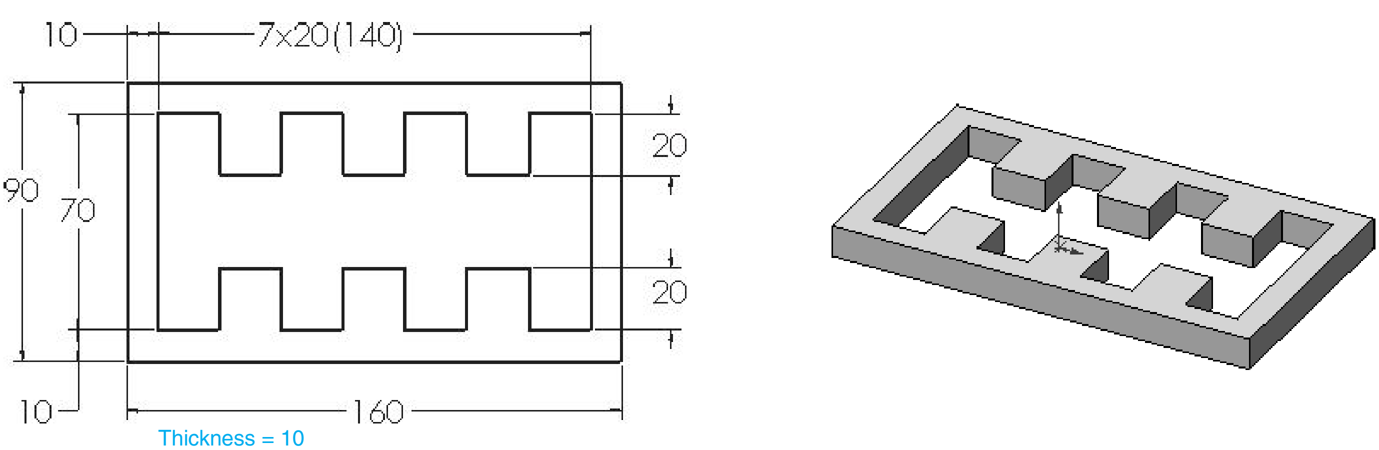


Figure P2-4 MILLIMETERS

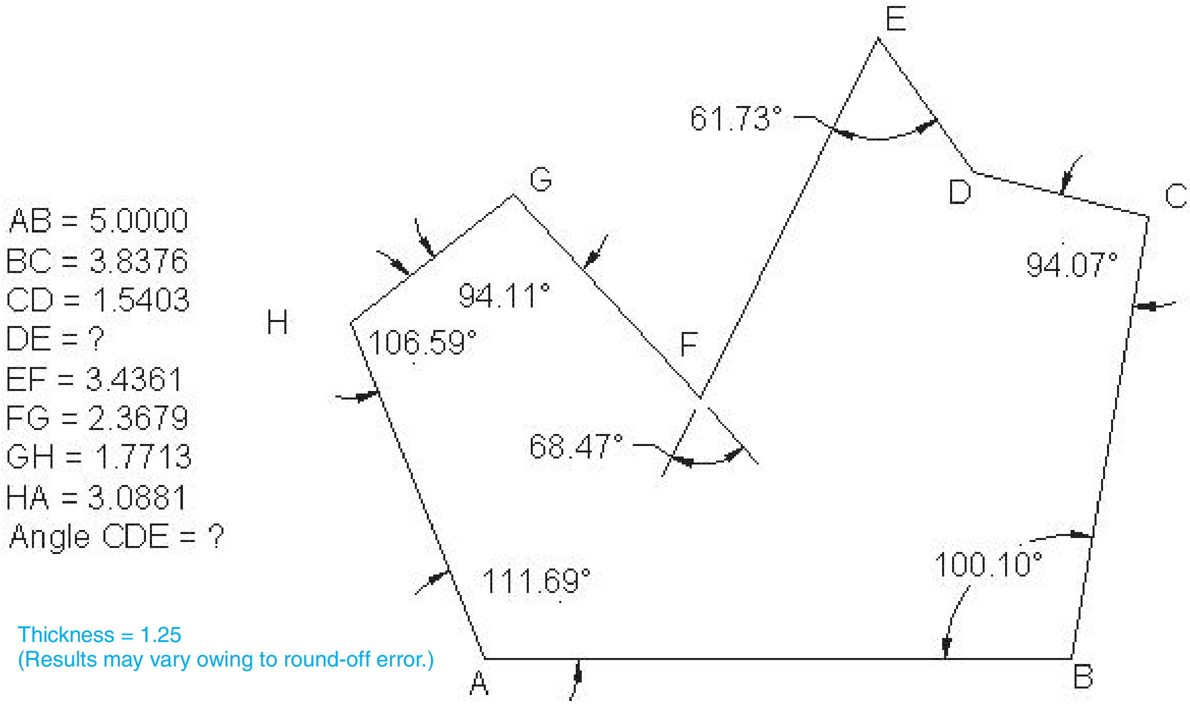


Figure P2-5 INCHES

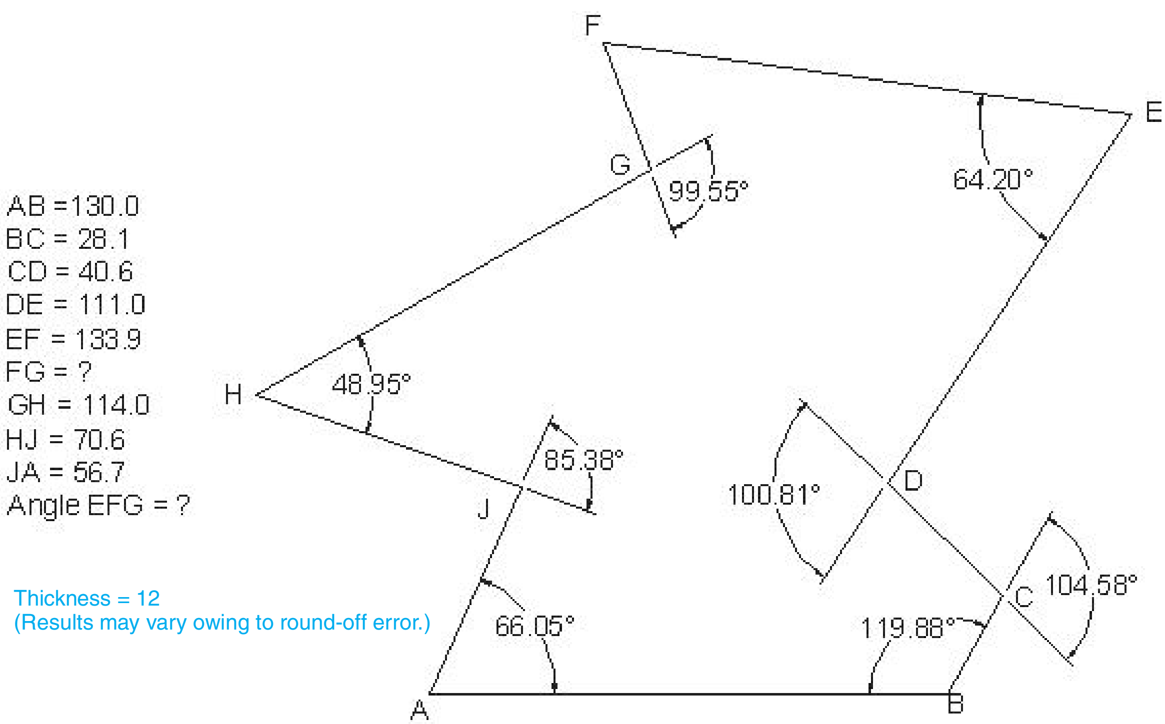


Figure P2-6 MILLIMETERS

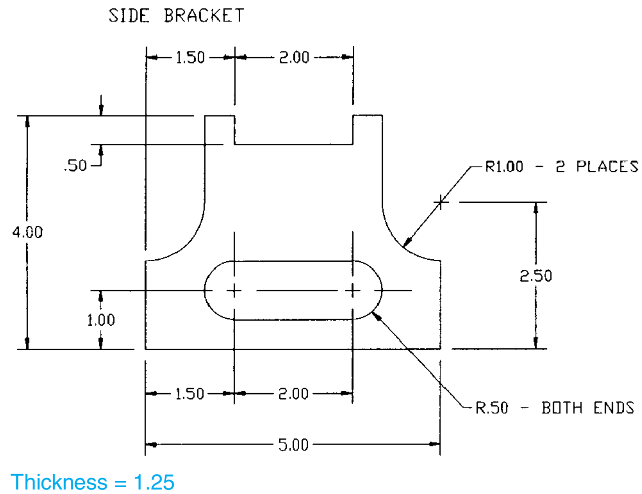


Figure P2-7 INCHES

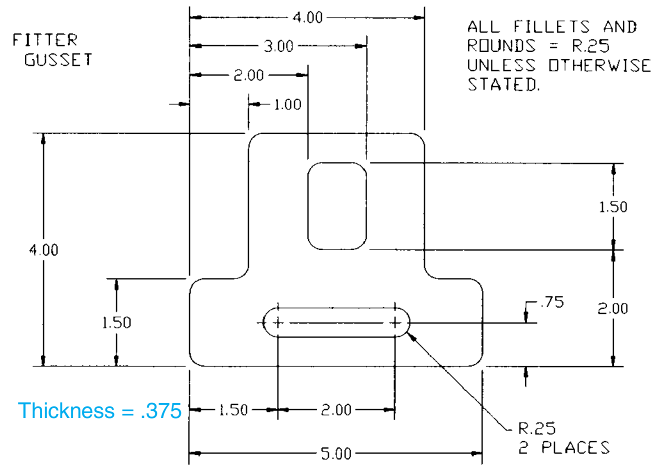


Figure P2-9 INCHES

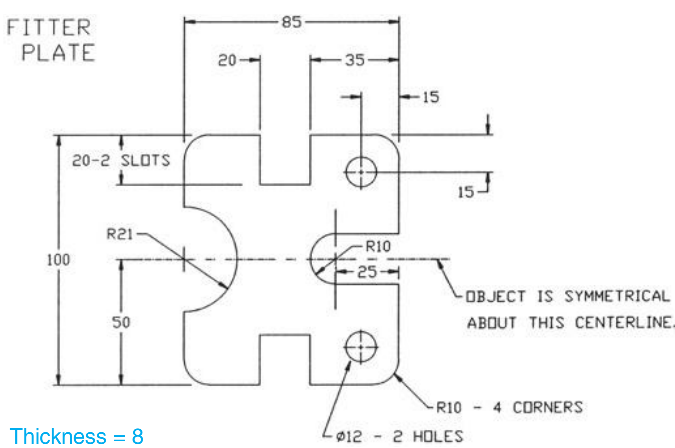


Figure P2-8 MILLIMETERS

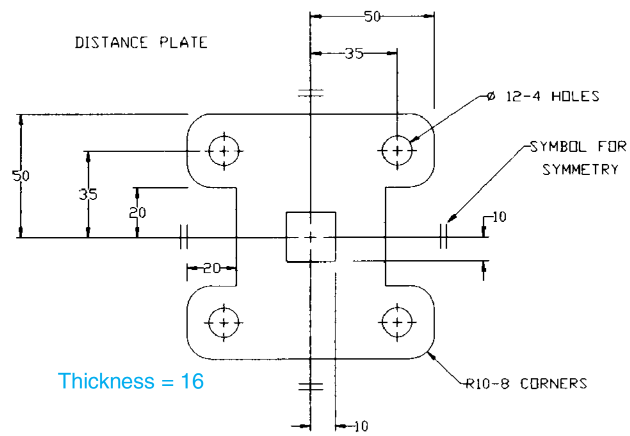


Figure P2-10 MILLIMETERS

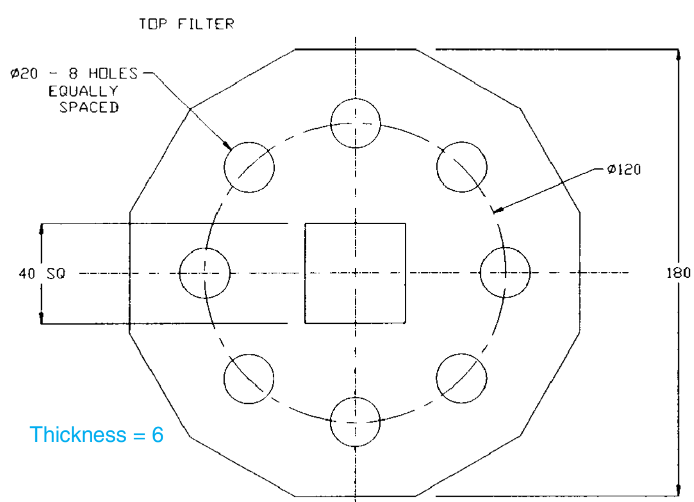


Figure P2-11 MILLIMETERS

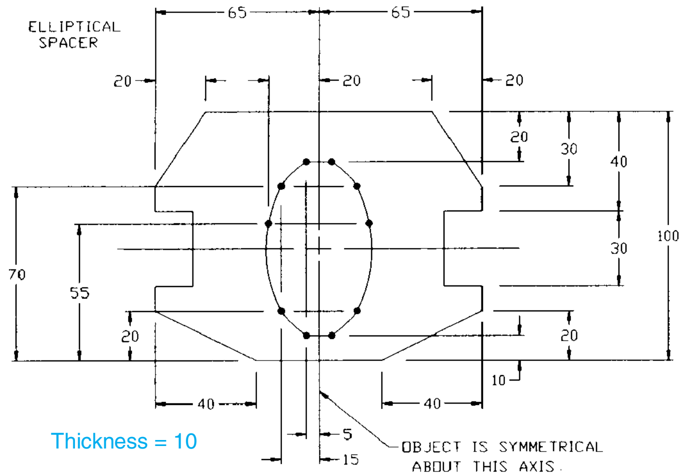


Figure P2-12 MILLIMETERS

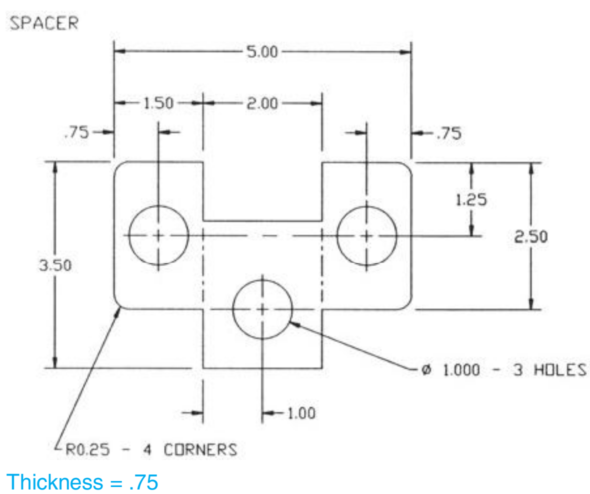


Figure P2-13 INCHES

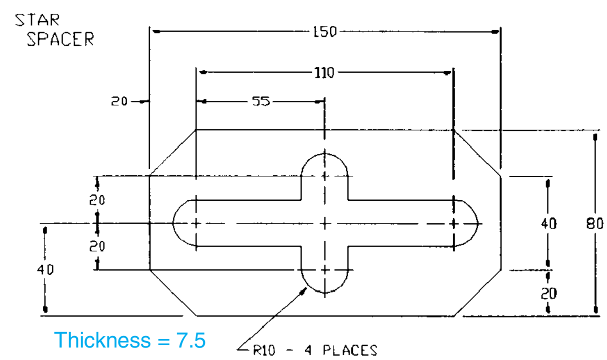
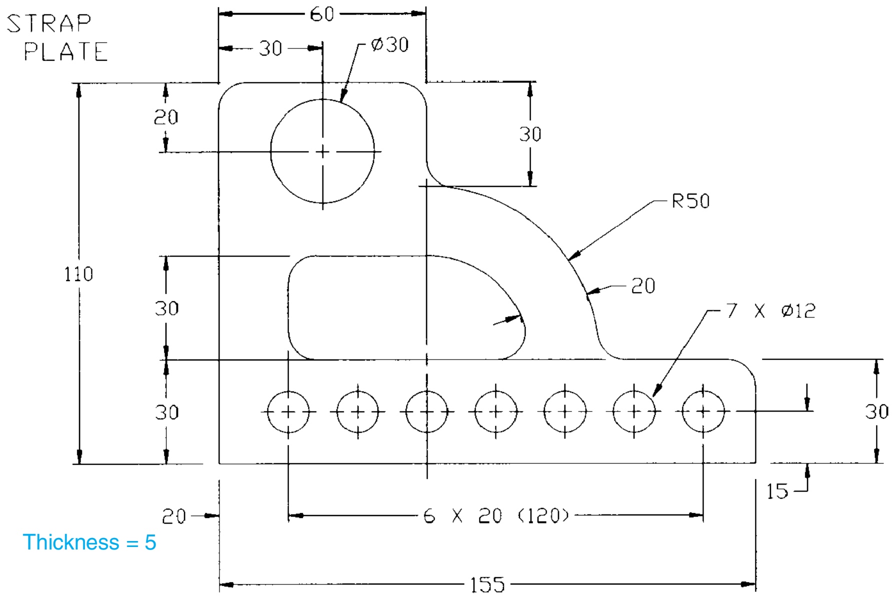
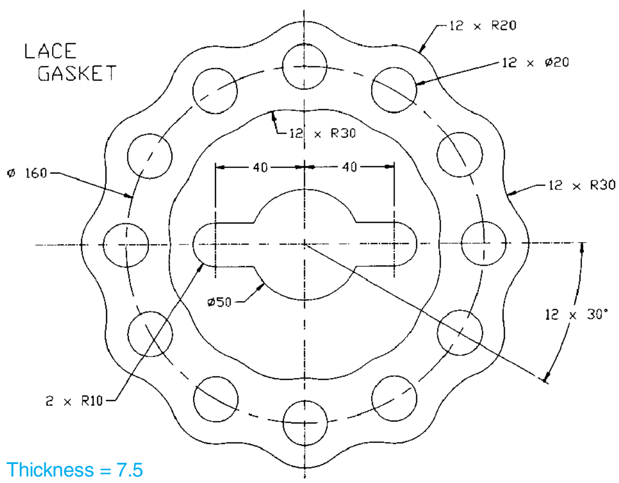


Figure P2-14 MILLIMETERS



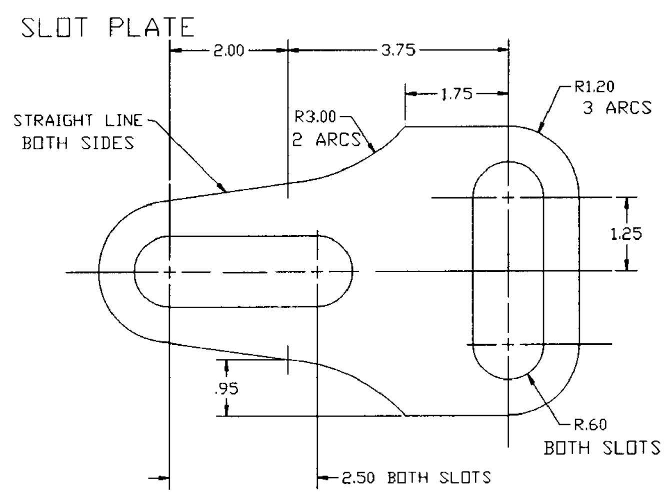
Thickness = 5

Figure P2-15 MILLIMETERS



Thickness = 7.5

Figure P2-16 MILLIMETERS



NOTE: Object is symmetrical about its horizontal centerline

Thickness = .875

Figure P2-17 INCHES

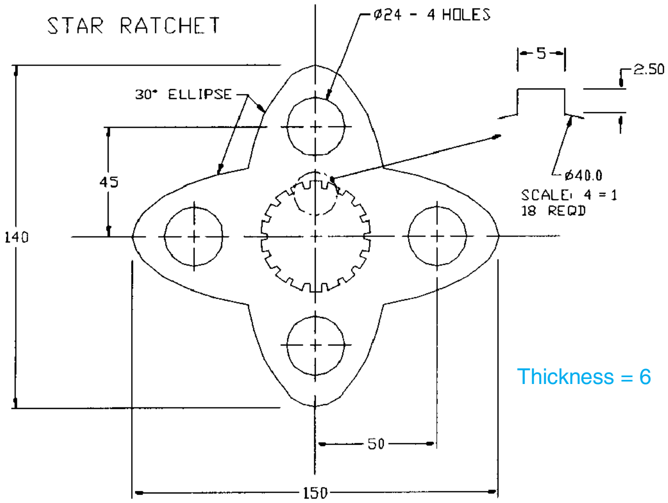


Figure P2-18 MILLIMETERS

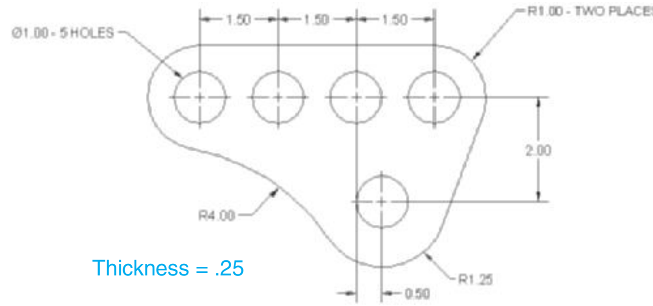


Figure P2-20 INCHES

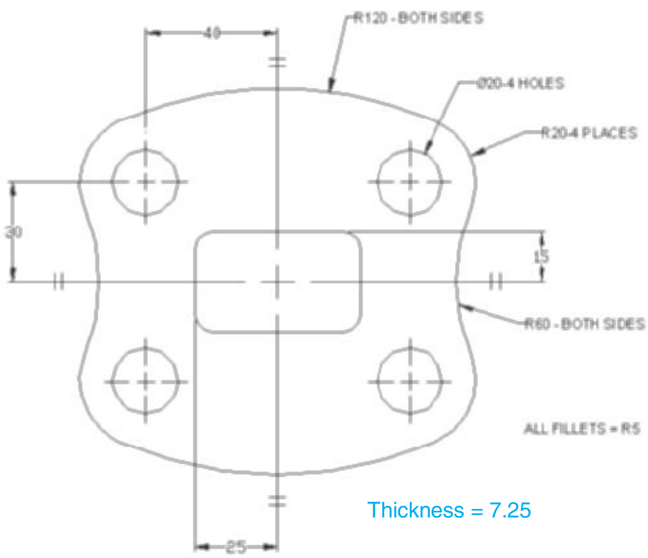


Figure P2-19 MILLIMETERS

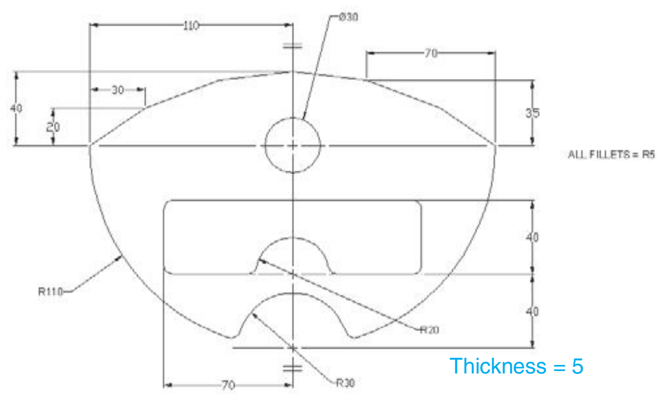


Figure P2-21 MILLIMETERS

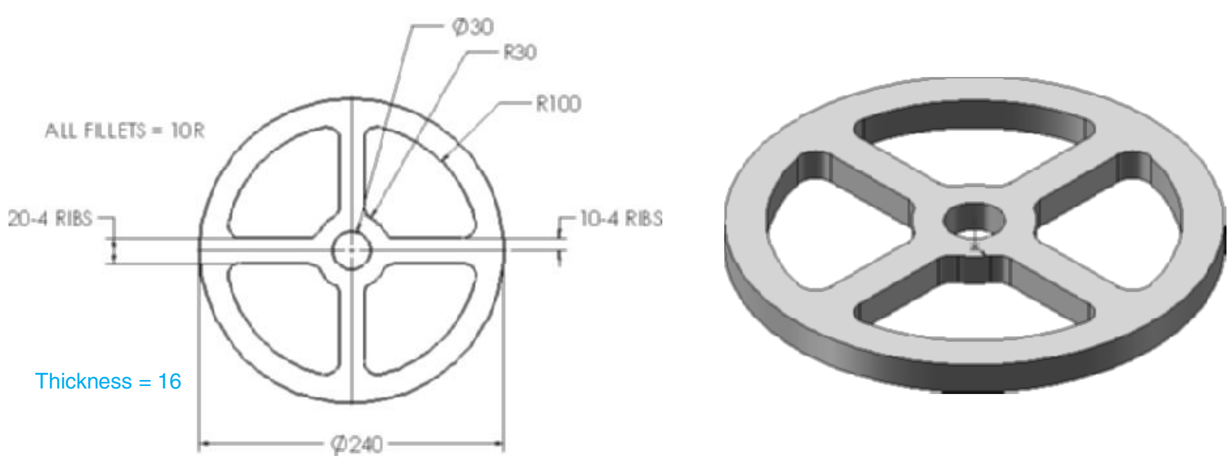
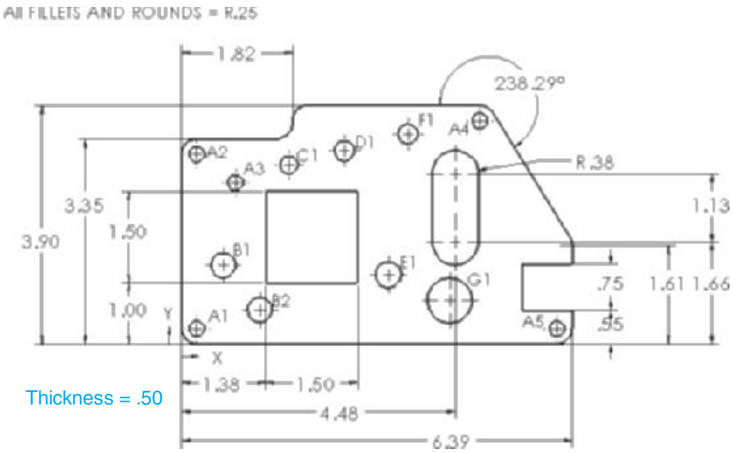


Figure P2-22 MILLIMETERS



TAG	X LOC	Y LOC	SIZE
A1	.25	.25	Ø.25
A2	.25	3.10	Ø.25
A3	.88	2.64	Ø.25
A4	4.88	3.65	Ø.25
A5	6.14	.25	Ø.25
B1	.68	1.29	Ø.40
B2	1.28	.56	Ø.40
C1	1.76	2.92	Ø.29
D1	2.66	3.16	Ø.33
E1	3.38	1.13	Ø.42
F1	3.71	3.43	Ø.32
G1	4.39	.71	Ø.73

Figure P2-23 INCHES

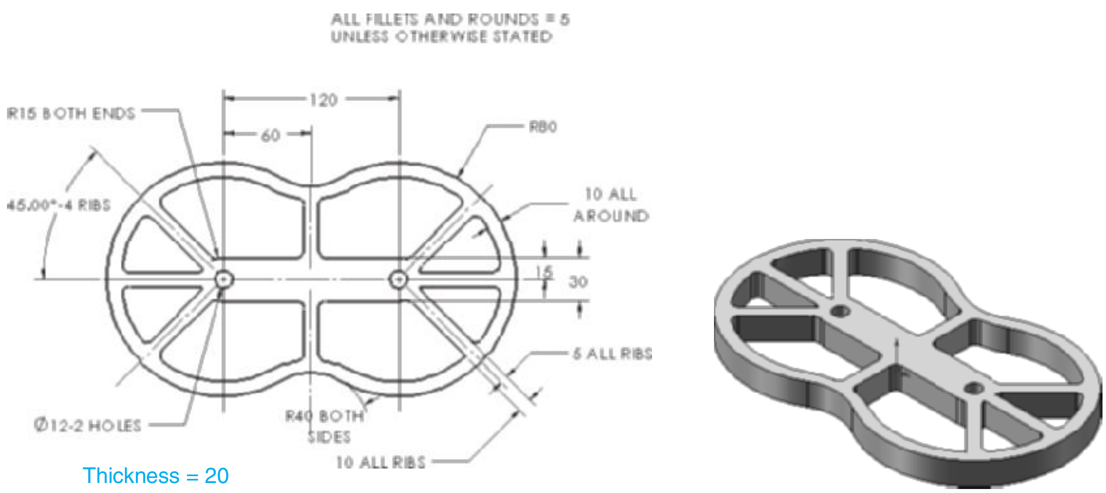


Figure P2-24 MILLIMETERS

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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.