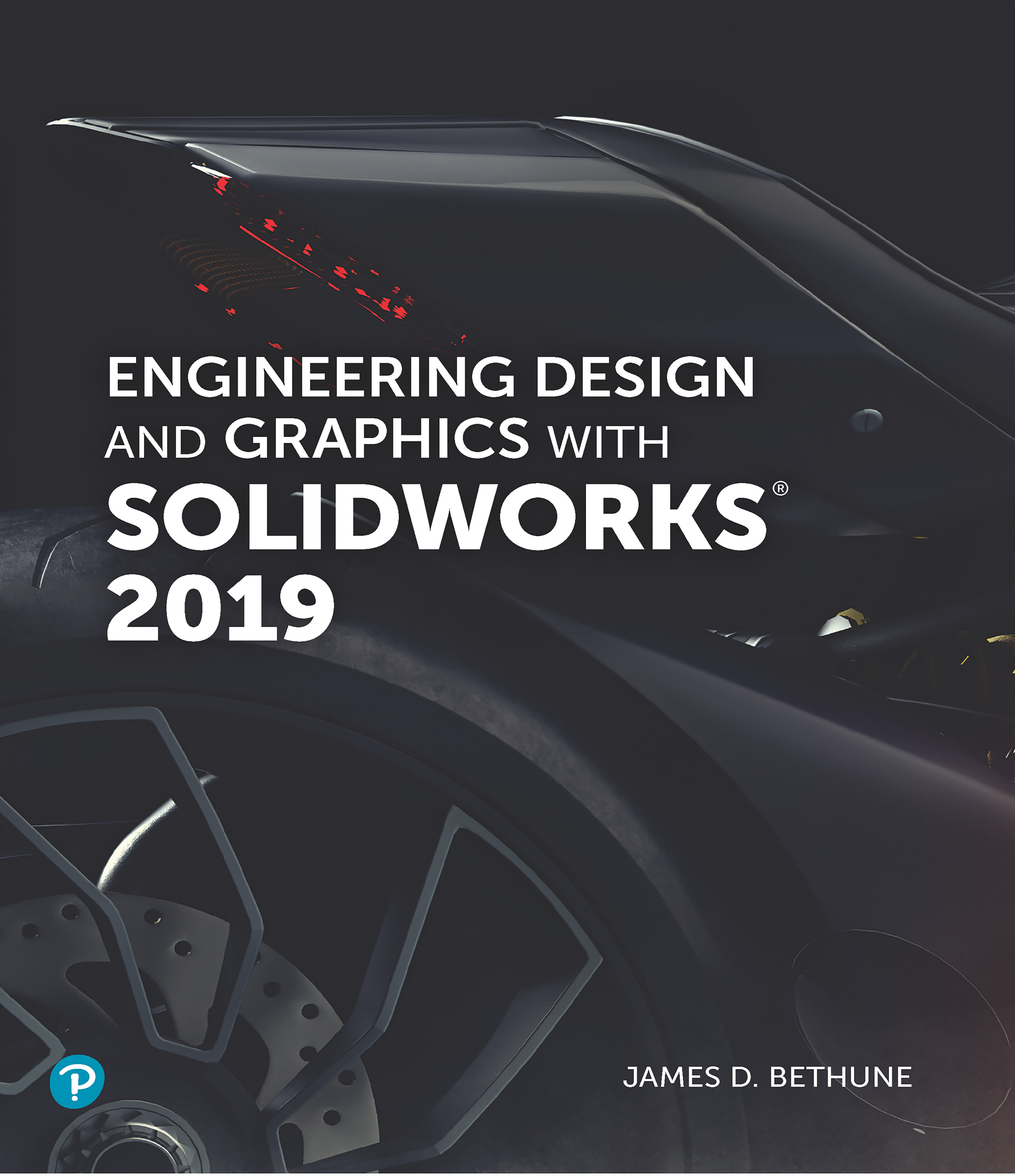

# ENGINEERING DESIGN AND GRAPHICS WITH SOLIDWORKS® 2019

JAMES D. BETHUNE

# Engineering Design and Graphics with SolidWorks® 2019

James D. Bethune

Pearson

**Engineering Design and Graphics with SolidWorks® 2019**



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**Composition:** codeMantra
**Proofreader:** Abigail Manheim

Library of Congress Control Number: 2019931697

ISBN 10: 0-13-540175-5
ISBN 13: 978-0-13-540175-0

1 19

# Preface

This book shows and explains how to use SolidWorks® 2019 to create engineering drawings and designs. Emphasis is placed on creating engineering drawings including dimensions and tolerances and using standard parts and tools. Each chapter contains step-by-step sample problems that show how to apply the concepts presented in the chapter.

The book contains hundreds of projects of various degrees of difficulty specifically designed to reinforce the chapter's content. The idea is that students learn best by doing. In response to reviewers' requests, some more difficult projects have been included.

**Chapter 1** and **2** show how to set up a part document and how to use the SolidWorks **Sketch** tools. **Sketch** tools are used to create 2D part documents that can then be extruded into 3D solid models. The chapters contain an explanation of how SolidWorks' colors are used and of how shapes can be fully defined. The usage of mouse gestures, S key, and origins is also included. The two chapters include 43 projects using both inches and millimeters for students to use for practice in applying the various **Sketch** tools.

**Chapter 3** shows how to use the **Features** tools. **Features** tools are used to create and modify 3D solid models. In addition, reference planes are covered, and examples of how to edit existing models are given.

**Chapter 4** explains how to create and interpret orthographic views. Views are created using third-angle projection in compliance with ANSI standards and conventions. The differences between first-angle and third-angle projections are demonstrated. Five exercise problems are included to help students learn to work with the two different standards. Also included are section views, auxiliary views, and broken views. Several of the projects require that a 3D solid model be drawn from a given set of orthographic views to help students develop visualization skills.

**Chapter 5** explains how to create assembly drawings using the **Assembly** tools (**Mate**, exploded **View**) and how to document assemblies using the **Drawing Documents** tools. Topics include assembled 3D solid models, exploded isometric drawings, and bills of materials (BOMs). Assembly numbers and part numbers are discussed. Both the **Animate Collapse/Explode** and **Motion Study** tools are demonstrated. In addition, the title, release, and revision blocks are discussed. An explanation of how to use **Interference Detection** is given.

**Chapter 6** shows how to create and design with threads and fasteners. Both ANSI inch and ANSI metric threads are covered. The **Design Library** is presented, and examples are used to show how to select and size screws and other fasteners for assembled parts.

**Chapter 7** covers dimensioning and is in compliance with ANSI standards and conventions. There are extensive visual examples of dimensioned shapes and features that serve as references for various dimensioning applications.

**Chapter 8** covers tolerances. Both linear and geometric tolerances are included. This is often a difficult area to understand, so there are many examples of how to apply and how to interpret the various types of tolerances. Standard tolerances as presented in the title block are demonstrated. Many of the figures have been updated.

**Chapter 9** explains bearings and fit tolerances. The **Design Library** is used to create bearing drawings, and examples show how to select the correct interference tolerance between bearings and housing, and clearance tolerances between bearings and shafts.

**Chapter 10** presents gears. Gear terminology, gear formulas, gear ratios, and gear creation using the SolidWorks **Toolbox** are covered. The chapter relies heavily on the **Design Library**. Keys, keyways, and set screws are discussed. Both English and metric units are covered. There is an extensive sample problem that shows how to draw a support plate for mating gears and how to create an assembly drawing for gear trains. The projects at the end of the chapter include two large gear assembly exercises.

**Chapter 11** will help students prepare for the CSWA certification exam. There are many sample questions and examples. Students should time how long it takes them to do each problem. This will help them get used to working under time pressure.

The **Appendix** includes fit tables for use with projects in the text. Clearance, locational, and interference fits are included for both inch and millimeter values.


## Acknowledgments

I would like to acknowledge the reviewers of this text: Peggy Condon-Vance, Penn State Berks; Lisa Richter, Macomb Community College; Julie Korfhage, Clackamas Community College; Max P. Gassman, Iowa State University; Paul E. Lienard, Northeastern University; and Hossein Hemati, Mira Costa College.

Thanks to editor Chhavi Vig. Thanks to my family—David, Maria, Randy, Sandra, Hannah, Will, Madison, Jack, Luke, Sam, and Ben.

A special thanks to Cheryl.

*James D. Bethune*

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# 1 chapterone
# Getting Started

## CHAPTER 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 angular and circular shapes
- Learn how to draw holes
- Learn how to use **Sketch** tools
- Learn how to change units of a part

### 1-1 Introduction

SolidWorks is a *parametric modeler*. A solid modeler uses dimensions, parameters, and relationships to define and drive 3D shapes. Solid modelers make it easy to edit and modify parts as they are constructed. This capability is ideal for creating new designs.

Parametric modelers use dimensions to drive the shapes. For example, to create a line of a defined length, a line is first sketched, and then the length dimension is added. The line will assume the length of the dimension. If the dimension is changed, the length of the line will change to match the new dimension.

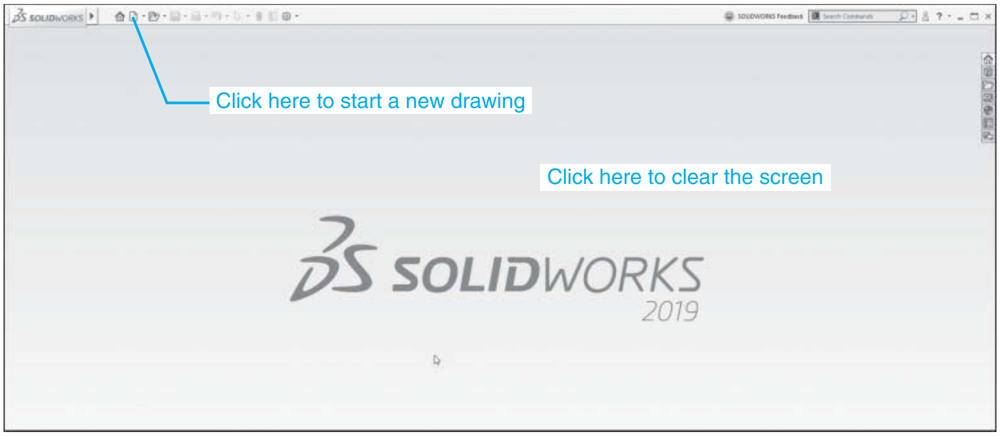
When using *non-parametric modelers*, a line is drawn and a dimension added. The dimension will define the length of the existing line but not drive it. If the length of the line is changed, the dimensions will not change. A new dimension is required to define the length of the line.

This chapter will show you how to start a **New** drawing and introduce the **Line**, **Circle**, and **Edit** tools. The **Smart Dimension** tool will be used to define and edit lines and circles. Line colors and relationships will also be introduced.

## 1-2 Starting a New Drawing

**Figure 1-1**

Figure 1-1 shows the initial SolidWorks screen.




### To Start a New Drawing


**1** Click the **New** tool icon at the top of the drawing screen.

A new drawing screen will appear. See Figure 1-2. The **New SolidWorks Document** dialog box will appear. SolidWorks can be used to create three types of documents: **Part**, **Assembly**, and **Drawing**.

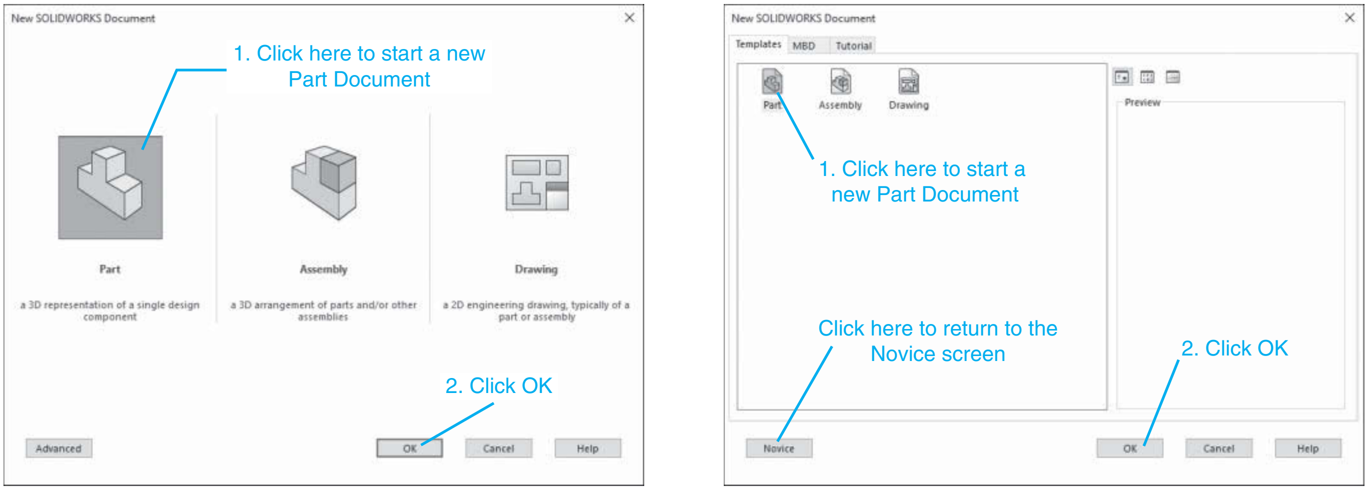



**Figure 1-2**

There are two versions of the **New SolidWorks Document** dialog box: Novice and Advanced. The Advanced version includes Tutorials. Either version can be used to access the **Part Document** area.

**Part** drawings are 3D solid models of individual parts.

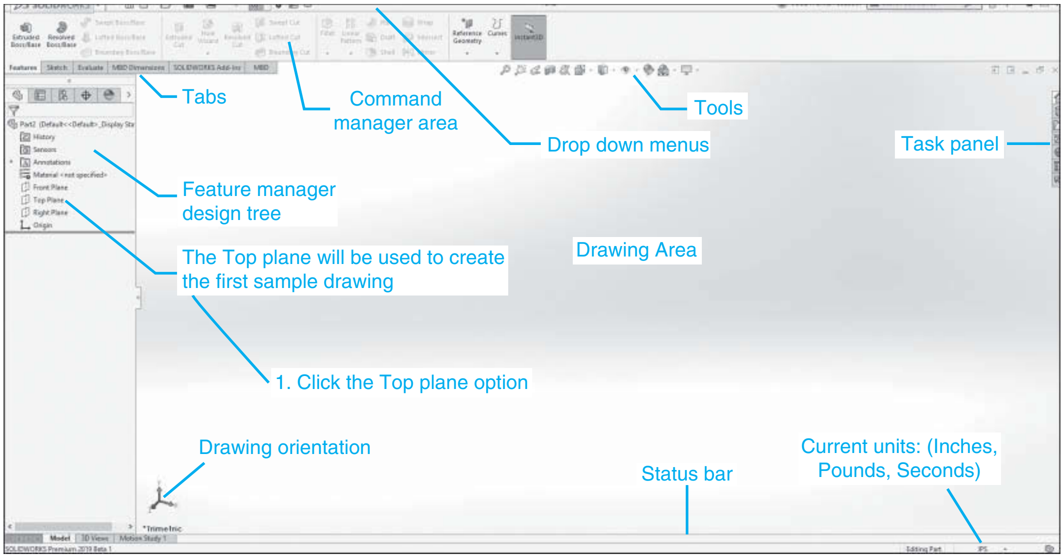
**Assembly** drawings are used to create drawings of assemblies that contain several Part drawings.

**Drawing** drawings are used to create orthographic views of the Part and Assembly drawings. Dimensions and tolerances can be applied to **Drawing** drawings.

2 Click the **Part** tool and then click the **OK** box.

The **Part** drawing screen will appear. See Figure 1-3. Note the different areas of the screen. The **Features** tab is currently activated, so the **Features** tools are displayed. Each tool icon on the **Features** toolbar is accompanied by its name. These names can be removed and the toolbar condensed to expand the size of the drawing screen. For clarity these named tools will be included in the first few chapters of the book so you gain enough knowledge of the tools to work without their names.

Figure 1-3




## To Select a Drawing Plane

SolidWorks uses one of three basic planes to define a drawing: **Front**, **Top**, and **Right**. These planes correspond to the planes used to define orthographic views that will be explained in Chapter 4. The **Top** plane will be used to demonstrate the first few tools.

3 Define the plane on which the part will be created.

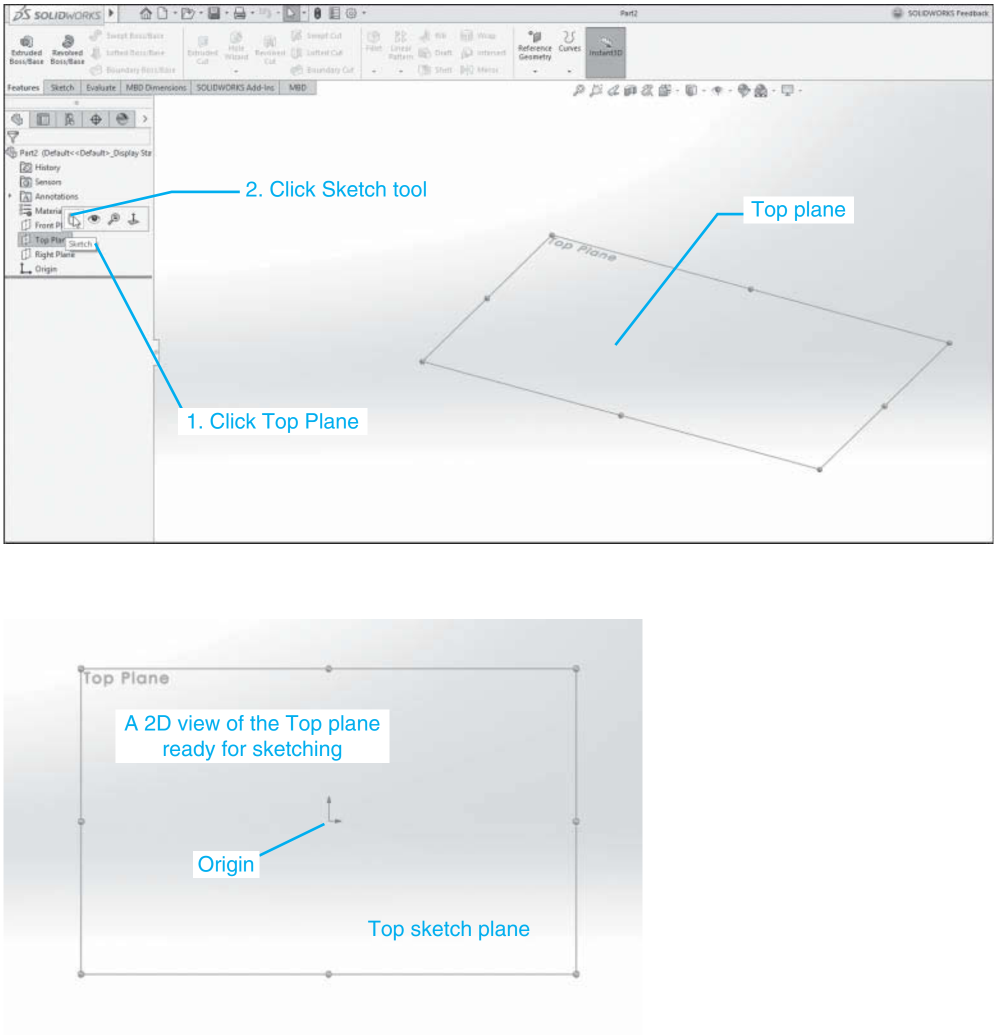
4 Click the **Top plane** option in the **Feature manager** box on the left side of the drawing screen.

See Figure 1-4. An outline of the **Top** plane will appear using the **Trimetric** orientation, that is, a type of 3D orientation.

5 Click the **Sketch** tool as shown in Figure 1-4.

The **Top** plane's orientation will change to a 2D view. The **Top** plane appears as a rectangle because the view is taken at 90° to the plane. This means that all 2D shapes drawn on the plane will appear as true shapes.

**Figure 1-4**

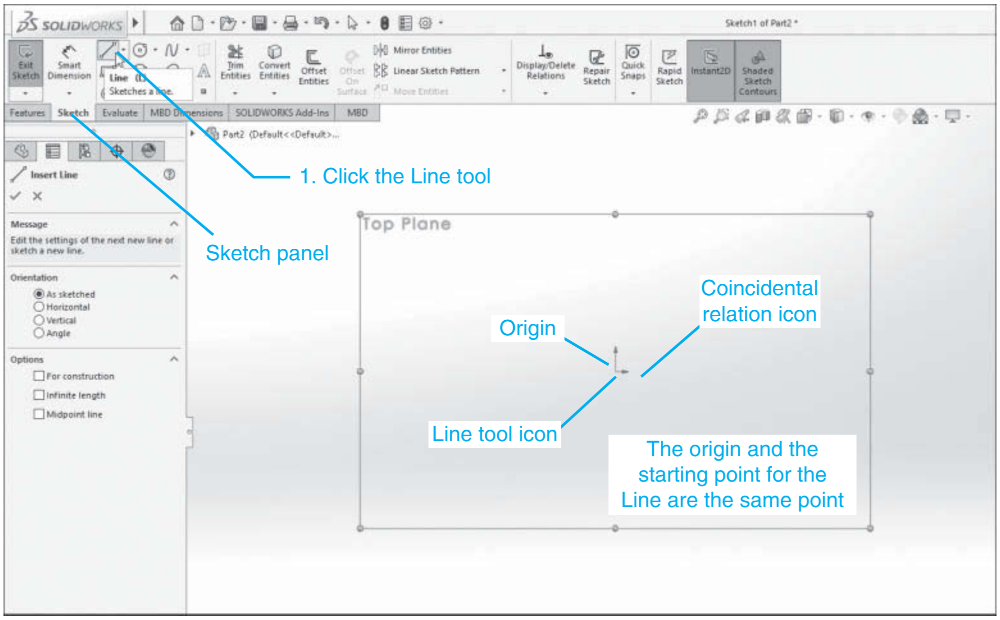



**6** Click the **Line** tool.

With the **Line** tool activated, locate the cursor on the origin. The origin is indicated by the two red arrows spaced 90° apart. See Figure 1-5.

Two icons will appear on the screen: the **Line** tool icon indicating that the **Line** tool is active, and the **Coincident relationship** icon indicating that the origin and the starting point for the line are on the same point.

Figure 1-5






**7** Move the cursor away from the origin horizontally to the right.

As you move the cursor away from the origin a distance, an angle value will appear. See Figure 1-6. The distance is as measured from the origin or starting point for the line and the angle is based on the SolidWorks definition of 0° as a horizontal line to the left of the starting point. We are drawing to the right, so the angular value is 180°.

Two other icons will also appear: the **Line** tool icon and the horizontal relationship icon.

**8** Click the mouse to define the endpoint of the line.

**9** Move the cursor vertically downwards. Do not click the mouse.

A new line will be drawn using the endpoint of the horizontal line as the starting point for the vertical line. Distance and angle values will appear based on the new starting point, and the **Line** and vertical relationship icons will appear.

**10** Press the Escape **<Esc>** key or right-click the mouse and click the **Select** option.

**11** Click the **Smart Dimension** tool, click the line, and move the cursor away from the line.

A dimension will appear.

**12** Click the mouse to define the location of the dimension.

The **Modify** dialog box will appear.

**13** Enter a distance value for the line and click the green **OK** check mark.

**14** Click anywhere on the drawing screen to complete the line drawing.

The dimension can be moved by locating the cursor on the dimension, pressing and holding the mouse button, and dragging the cursor.

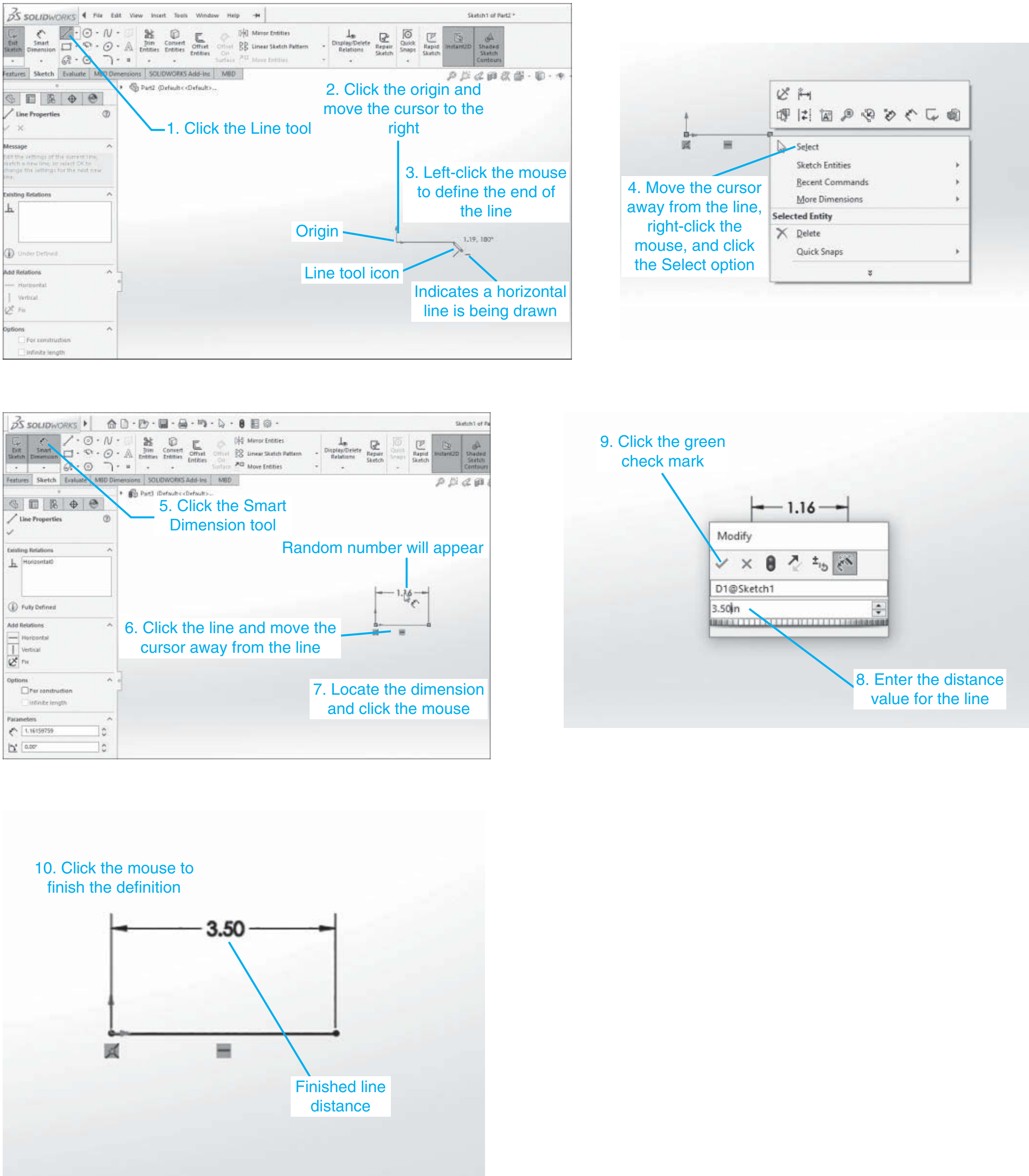



**Figure 1-6**

**15** Click the **File** tab located at the top of the screen.

See Figures 1-7 and 1-8.

**16** Click the **Don't Save** option.

The screen will return to the original SolidWorks screen.

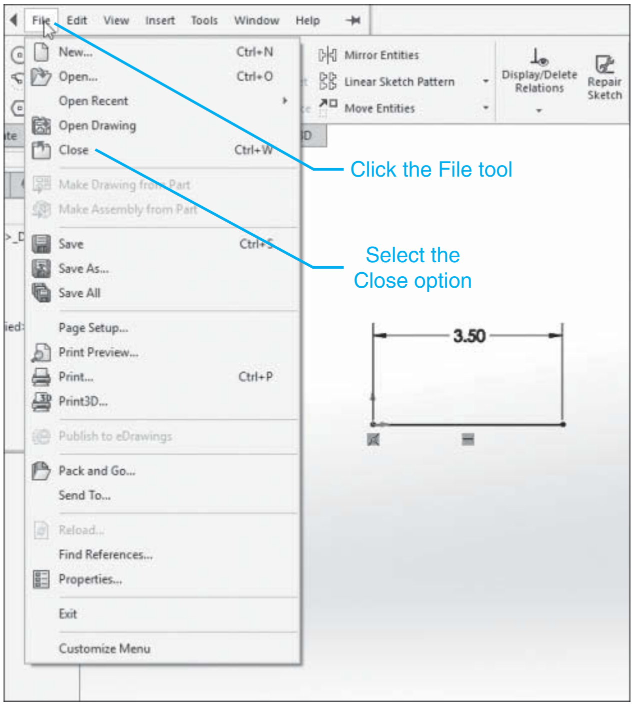



Figure 1-7

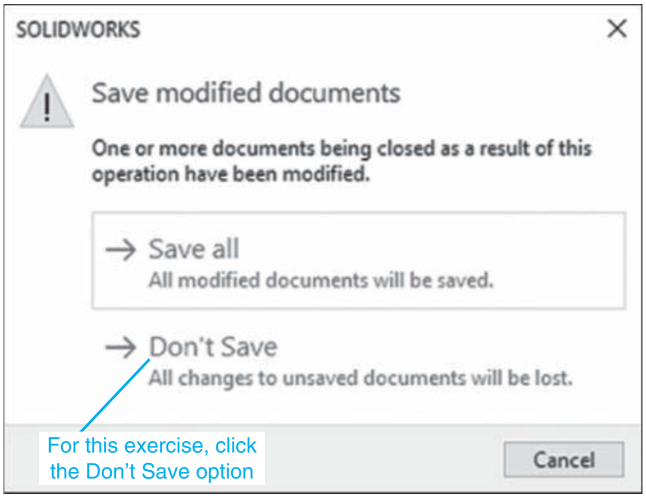



Figure 1-8

## 1-3 SolidWorks Colors

As you work with SolidWorks you will notice that the lines change colors. These color changes let you know the status of the sketch being drawn. There are four basic colors.

BLACK = Fully Defined

BLUE = Under Defined

RED = Over Defined

YELLOW = Redundant

## 1-4 Creating a Fully Defined Circle

In this section we will sketch a circle to help understand the difference between a fully defined and an under defined **Part**.

Start a **New Part** drawing and click the **Top plane** tool as defined in Figure 1-4.

1 Click the **Sketch** tab. (It may already be activated.)

2 Click the **Circle** tool.

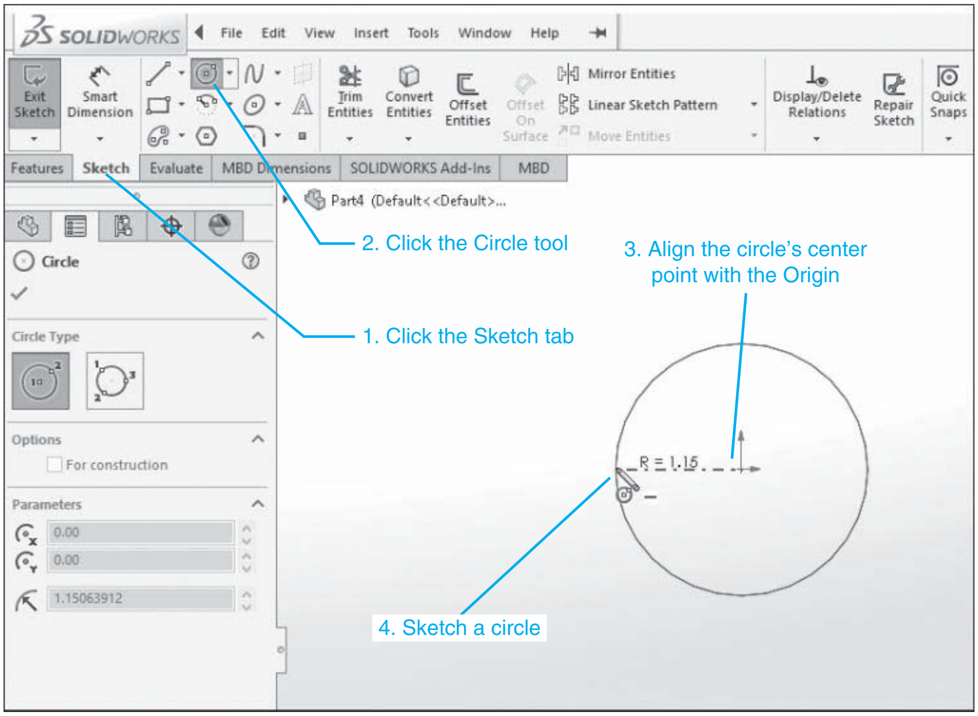
3 Locate the cursor on the origin, click the mouse, and drag the cursor away from the origin center point.

Note that the Coincident relationship symbol appears next to the origin, indicating that the center point of the circle is located on the origin.

4 Click the mouse to define a sketch radius for the circle.

This is a temporary radius, that is, a sketched radius, and is not the final radius. The circle will be blue, indicating that it is not fully defined. See Figure 1-9.

**Figure 1-9**



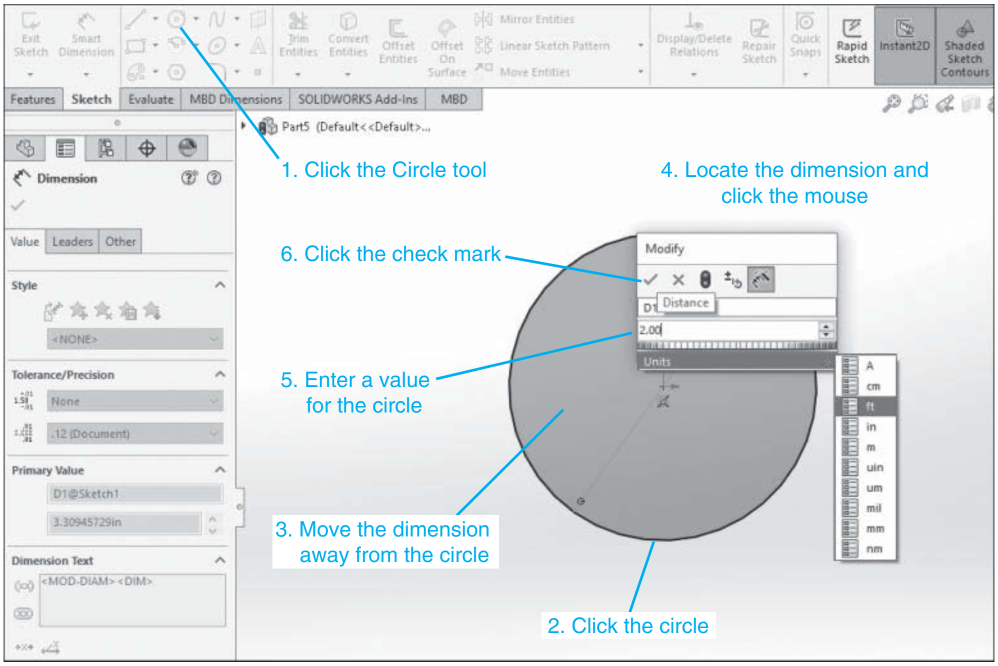

**5** Click the **Smart Dimension** tool on the **Sketch** panel.

**6** Click the circle and move the cursor away from the circle.

A dimension will appear. See Figure 1-10.

**7** Select a location for the dimension and click the mouse.

**Figure 1-10**



The circle will initially be blue, not fully defined, until the mouse is clicked, locating the circle's dimension. When the mouse is clicked, the circle will turn black; it is now fully defined. We know the circle's diameter and location.

When the mouse is clicked, the **Modify** dialog box will appear. The sketched diameter value will be listed in the box. This sketched diameter value is now the circle's diameter until we enter a new value.

8 Enter a diameter value for the circle.

In this example a value of 2.00 was entered.

9 Click the green **OK** check mark in the **Modify** box to enter the diameter value.

10 Click the green **OK** check mark in the **Manager** area to finish defining the circle.

## To Change an Existing Dimension

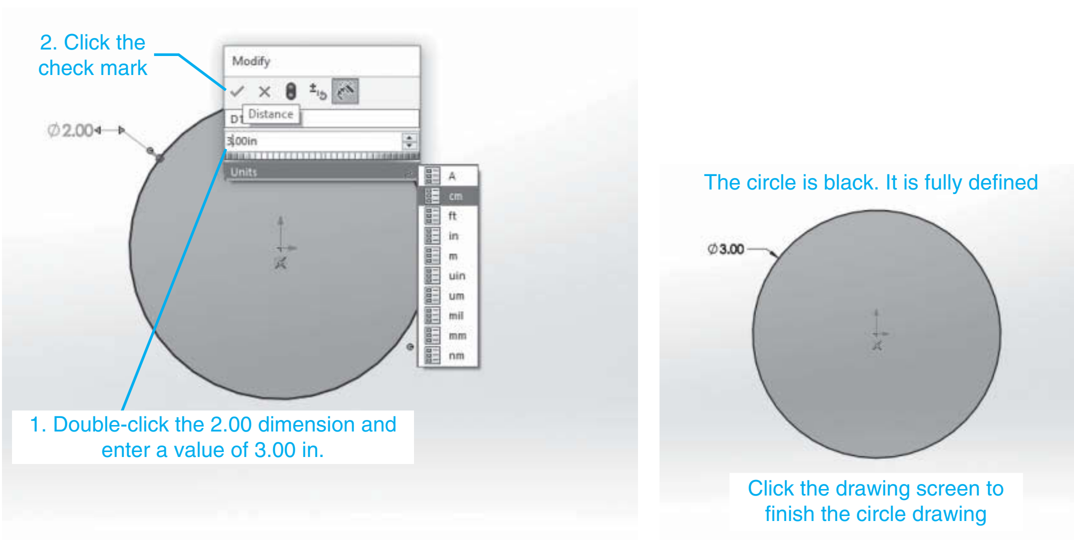
1 Double-click the **2.00** dimension.

The **Modify** dialog box will reappear.

2 Enter a new value.

In this example a value of **3.00** was entered. See Figure 1-11. The circle's diameter will change to 3.00 and the circle's color will remain black. The circle still is fully defined.

**Figure 1-11**



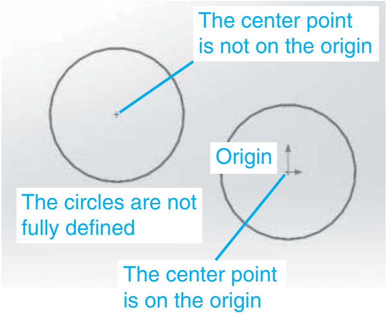

Note that the words **Fully Defined** appear at the bottom of the screen.

The circle is fully defined because both its diameter and location are known. The location was fully defined when we located the circle's center point on the origin. Every circle needs a locational value and a diameter value to be fully defined. The locational value may be linear, an X and Y component value, or polar, an angular and radius value.

## Fully Defined Entities

To help understand when an entity is fully defined, sketch two circles, one with its center point on the origin and the other with its center point not on the origin. See Figure 1-12. Both circles are under defined because the diameter values have not been defined. Both circles are sketched circles.

Figure 1-12




Use the **Smart Dimension** tool and define both their diameters as **Ø2.00**. The circle located on the origin will be black. It is fully defined. Both its diameter and location are known. The circle with its center point not located on the origin will remain blue. It is not fully defined. Its location is unknown. See Figure 1-13.

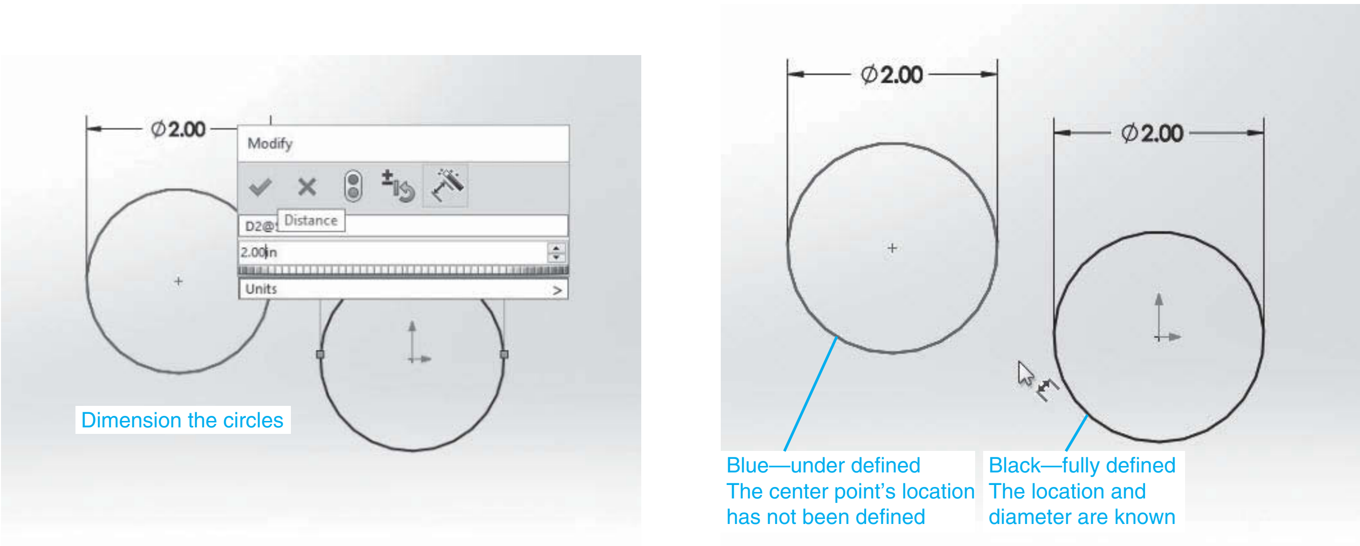



Figure 1-13

Figure 1-14 shows the two Ø2.00 circles again. This time, dimensions have been added to the circle not located on the origin. The dimensions define the circle's center point relative to the origin. It is now fully defined. Its color will change to black.

> **NOTE**
>
> Always include the origin as part of a 2D sketch.

Figure 1-14

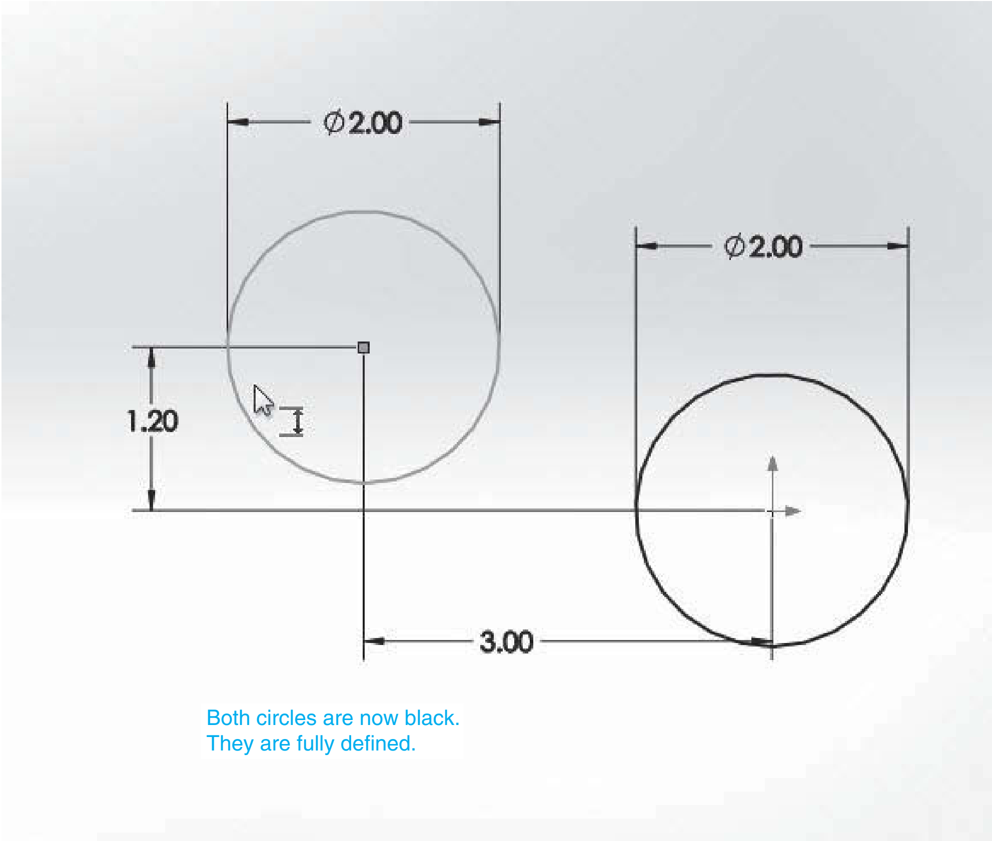



Figure 1-15 shows the two Ø2.00 circles with an extra dimension. The 1.20 vertical dimension is not needed to define the location of the hole not centered on the origin. A 1.20 vertical dimension already exists. The 1.20 dimension is redundant, so the drawing lines change to yellow.

Figure 1-15 also shows the **Make Dimension Driven?** dialog box. A driving dimension drives the shape and/or location of the object. If the driving dimension is changed, the shape or location will change. Driven dimensions are reference dimensions. They are sometimes added to a drawing for

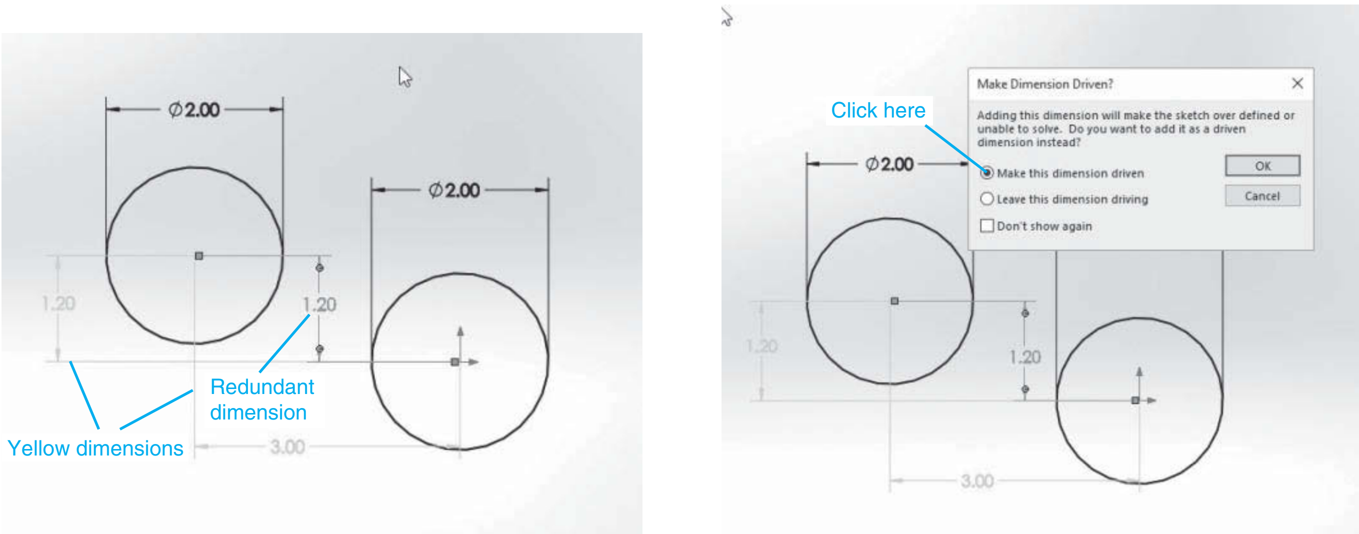



Figure 1-15

clarity. For example, a reference dimension could be used to show the overall value of a string of smaller dimensions. See Chapter 7, in this example it would be better to delete the extra 1.20 dimension. If you save it on the drawing, click the **Make this dimension driven** option and click **OK**. It will appear as a gray color. See Figure 1-16.

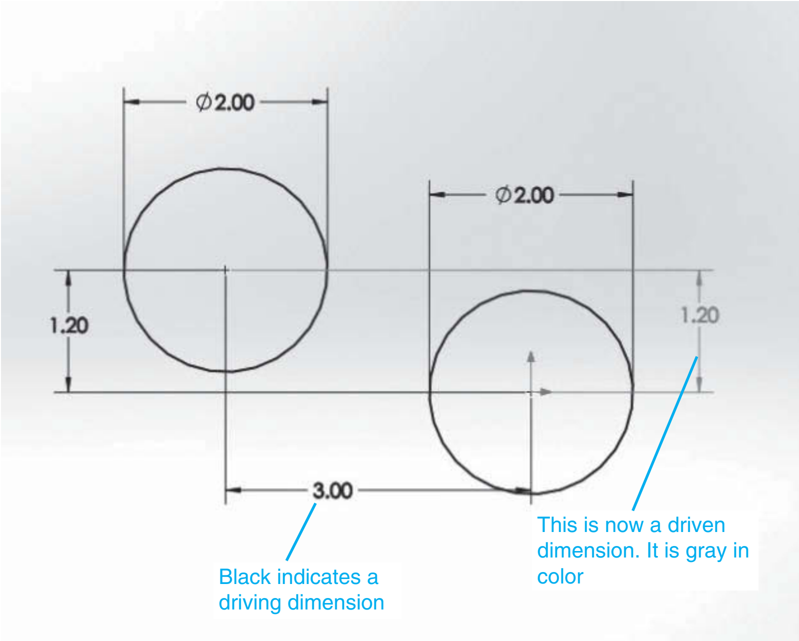



Figure 1-16

## 1-5 Units

This book will present examples and exercise problems using English units (inches) and Metric units (millimeters). Figure 1-17 shows the dimensioned circles created in the previous section. Note the letters **IPS** to the right of the **Fully Defined** callout. IPS stands for inch, pound, and second, the current units.

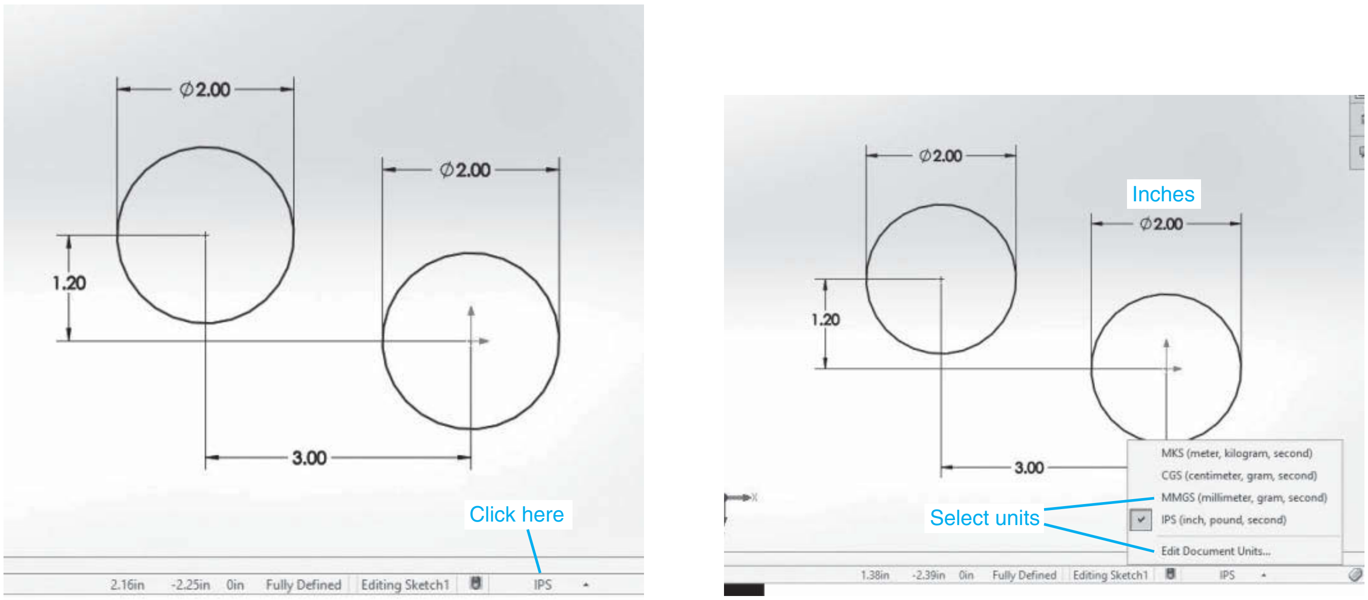
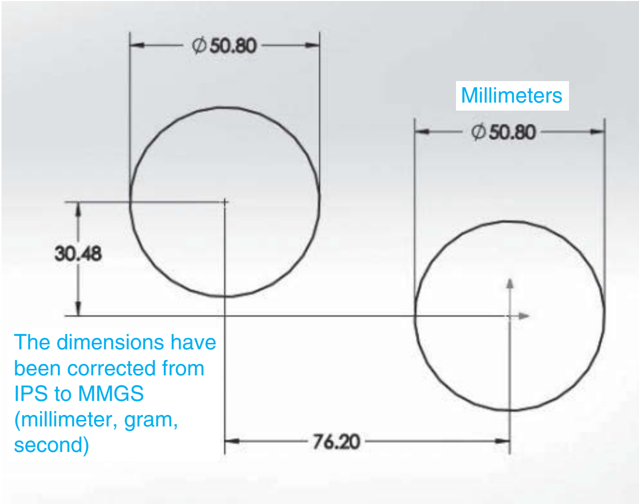



Figure 1-17

**Figure 1-17**
(*Continued*)






## To Change Units

1. Click the **IPS** callout at the bottom of the screen.
2. Select the desired units.

In this example millimeters (**MMGS**) was selected. MMGS stands for millimeter, gram, and second. The letters **MMGS** appear at the bottom of the screen, indicating the drawing units are now millimeters.

3. Click the **Undo** tool. The new dimensions appear.

> **NOTE**
> The converted millimeter dimensions are not whole numbers as were the inch units. It is better to do a drawing in either inches or millimeters from the beginning and not to convert units as a drawing is created. This helps prevent round-off errors.

# 1-6 Rectangle
## To Sketch a Rectangle

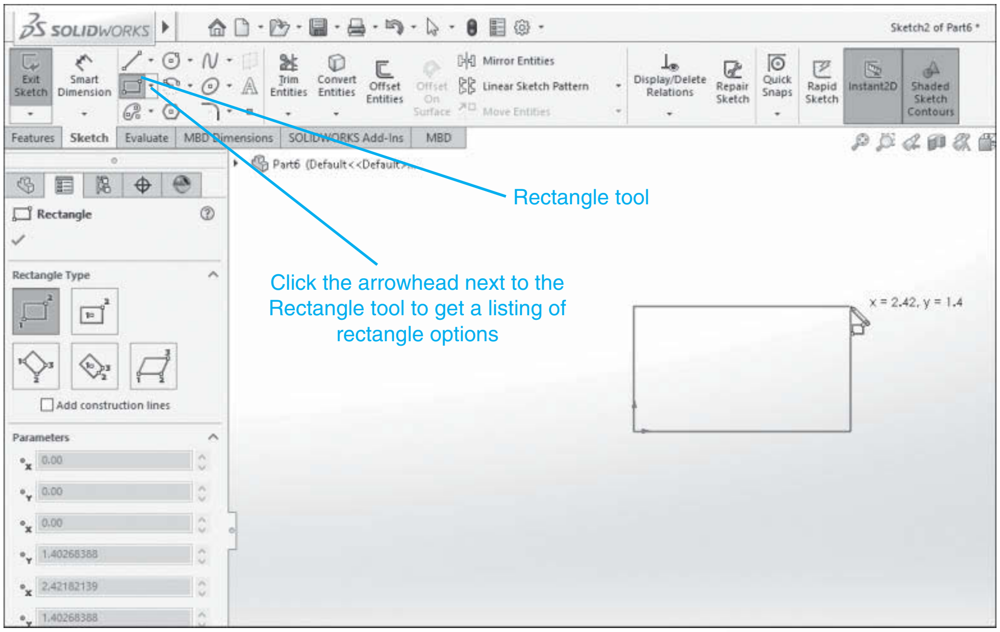
See Figure 1-18. The example was created on the **Top Plane** using the **Rectangle** tool. The units are inches.

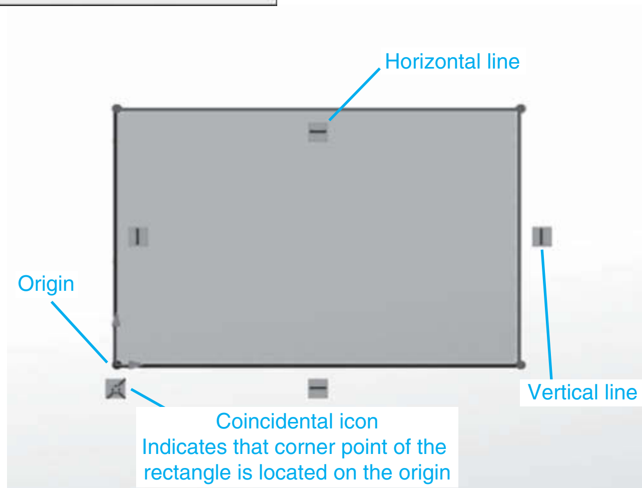
1. Start a **New Part** drawing, click the **Top Plane** option, and click the **Sketch** tool.

See Figure 1-18. The outline rectangle for the **Top Plane** will rotate to the *Normal* orientation, that is, you are looking at the plane from a 90° orientation. This means that any shape drawn on the plane will be a true shaped line. This concept will be covered in Chapter 4 on orthographic views.

2. Click the **Corner Rectangle** tool.

Five options for drawing a rectangle are listed. The different options are helpful when creating designs. It is recommended you take a few minutes and try each option. Only **Corner Rectangle** will be used in this chapter.


Rectangle tool
Click the arrowhead next to the Rectangle tool to get a listing of rectangle options


Horizontal line
Origin
Vertical line
Coincidental icon
Indicates that corner point of the rectangle is located on the origin

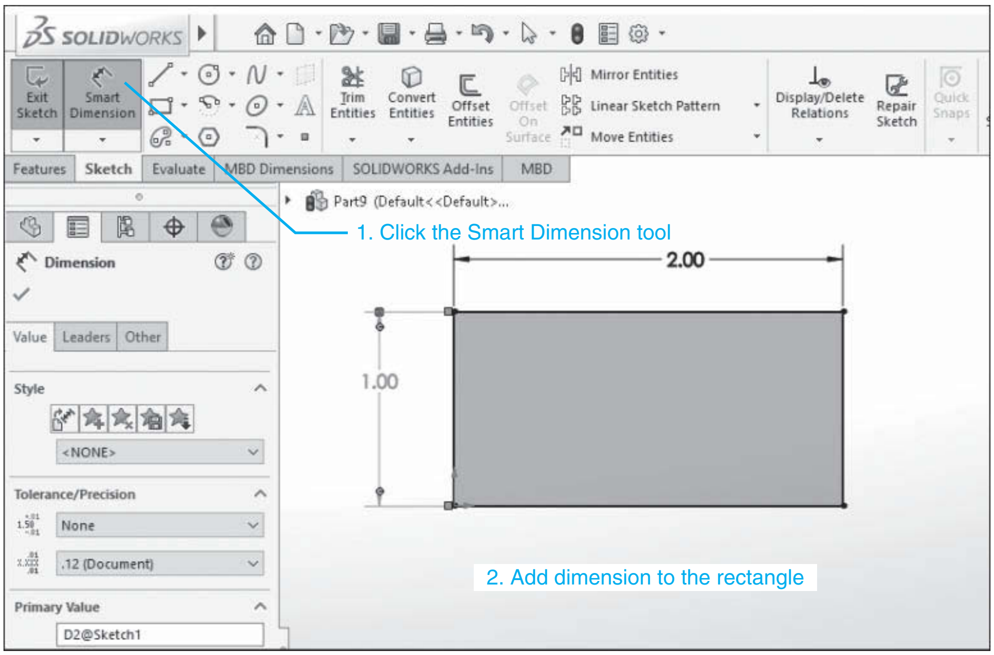

1. Click the Smart Dimension tool
2.00
1.00
2. Add dimension to the rectangle

**Figure 1-18**

3 Click the origin and move the cursor up and to the right.

4 Click the mouse to define the line's endpoint.

5 Press the **<Esc>** key or right-click the mouse and click the **Select** option.

Note that two relationships are defined: coincident and horizontal. The starting point was located on the origin, so they are coincidental and the rectangle is drawn. Note also that the rectangle is not fully defined because its size has not been defined.

Releasing the mouse button will define the length of the sketched line, but you are still in the **Sketch** mode. If you click the mouse again, a new rectangle will begin.

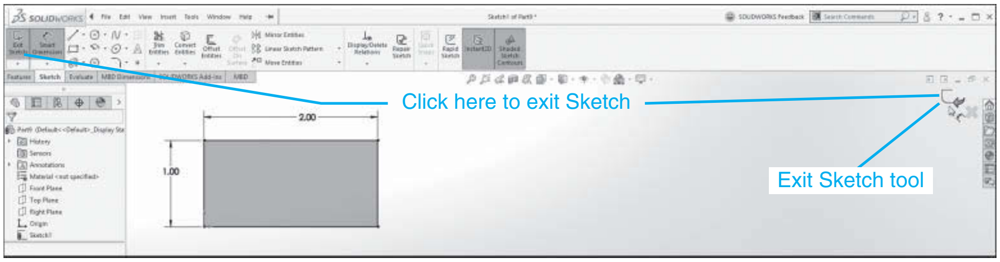
6 Use the **Smart Dimension** tool to define the size of the rectangle.

## To Exit the Sketch Mode

1 Click the **Exit Sketch** icon on the **Sketch** panel or click the **Exit Sketch** icon that appears in the upper right corner of the drawing screen.

See Figure 1-19.

Figure 1-19



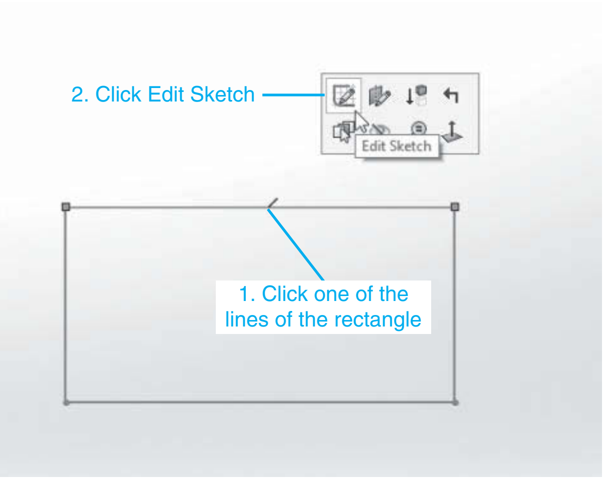

## To Reenter the Sketch Mode

Once you have created a sketch and left the **Sketch** mode, you can return to work on the sketch by using the **Edit Sketch** mode. See Figure 1-20.

1 Click an entity in the existing sketch.

2 Click the **Edit Sketch** tool.

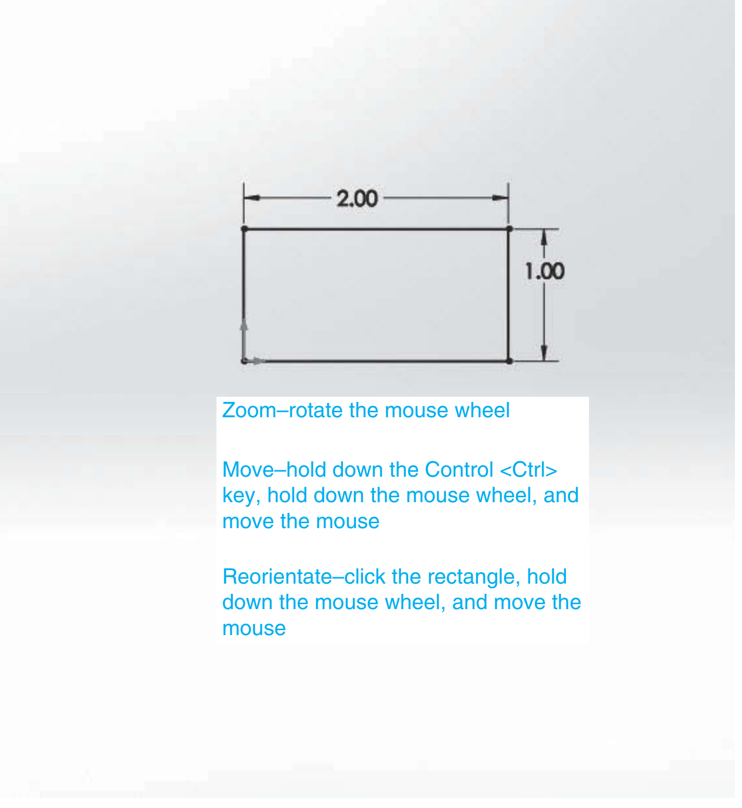
Figure 1-20




## 1-7 Moving Around the Drawing Screen

SolidWorks includes several methods that allow you to move entities about the screen. Entities can be moved, zoomed, or reorientated. Figure 1-21 shows the line created in the previous section.

Figure 1-21




### To Zoom the Line

**1** Rotate the mouse wheel.

The line will increase and decrease in length.

### To Move the Line

**1** Hold down the Control **<Ctrl>** key; press and hold down the mouse wheel.

**2** Move the mouse around.

The line will follow the mouse movement.

### To Reorientate the Line

1. Click the line.
2. Hold down the mouse wheel and move the mouse.

The mouse's orientation will follow the mouse movement.

## 1-8 Orientation

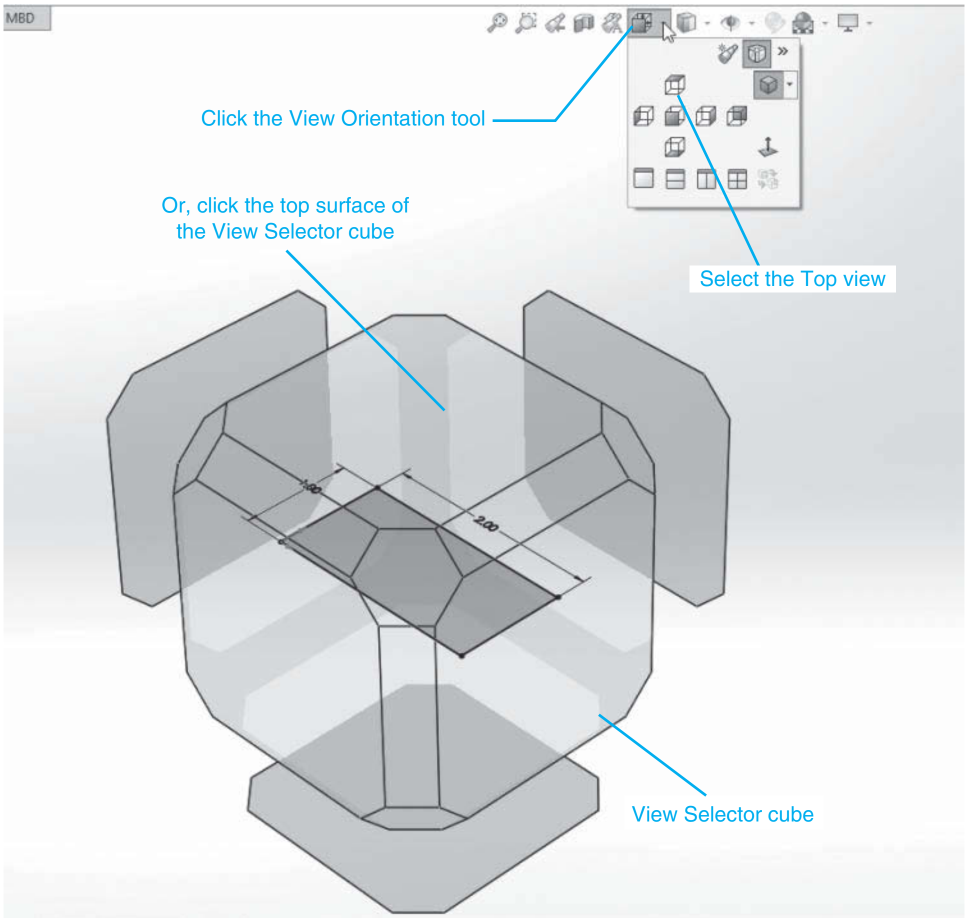
The rectangle in the previous sections was created in the **Top view** orientation. As you work on a sketch, the orientation may change. There are three ways you can use to return the sketch to its original orientation.

### To Return to the Top View Orientation – View Selector

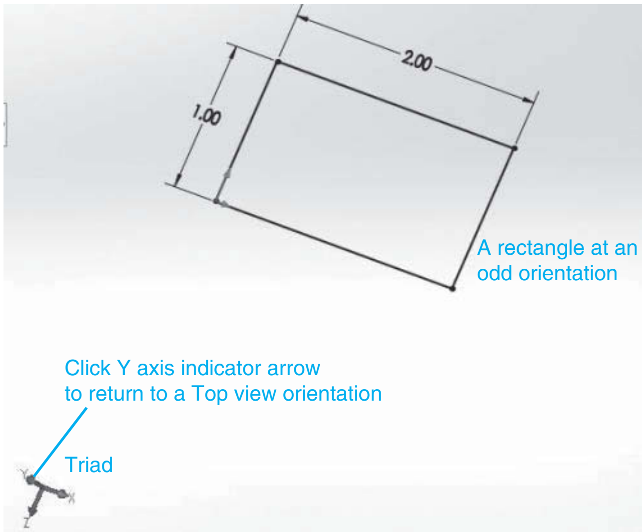
1. Click the **View Orientation** tool at the top of the drawing screen.

The **View Selector** cube will appear. See Figure 1-22. If the cube does not appear, click the **View Selector** icon on the **View Orientation** tool panel.

Figure 1-22




Chapter 1



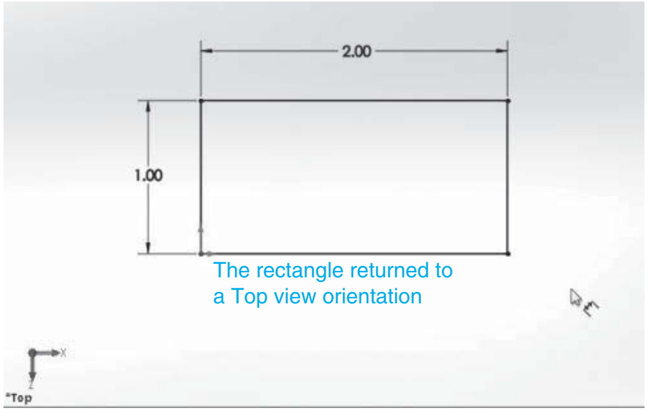




**Figure 1-22**
(*Continued*)

2 Click the top surface of the **View Selector** cube.

The sketch will return to the **Top view** orientation.

## To Return to the Top View Orientation – Top View

See Figure 1-22.

1 Click the **View Orientation** tool at the top of the drawing screen.

2 Click the **Top view** tool.

## To Return to the Top View Orientation – Orientation Triad

The **Orientation Triad** is located in the lower left corner of the drawing screen. See Figure 1-22.

SolidWorks defines the **Top Plane** as the XZ plane. The Y axis is 90° to the XZ plane, so a view taken along the Y axis will generate a top view of the plane.

1 Move the cursor onto the **Orientation Triad**.

2 Click the Y axis indicator arrow.

The triad will reorientate to the **Top view** orientation.

# 1-9 Sample Problem SP1-1

Figure 1-23 shows a 2D shape sketched using the **Line** tool. The dimensions are in millimeters. This section will explain how to draw the shape.

1 Start a **New Part** document, select the **Front Plane**, and create a **Sketch** plane.

See Figure 1-24.

2 Define the dimensional units as millimeters, **MMGS**.

See Figure 1-25.

3 Click the **Line** tool.

4 Select the origin as the starting point for the first line.

**Figure 1-23**

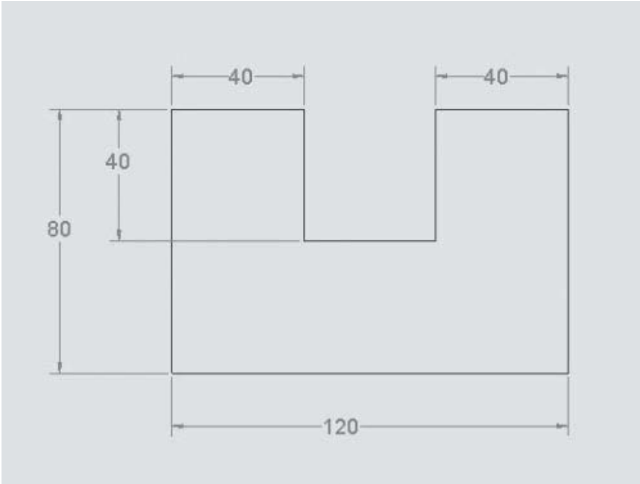



**Figure 1-24**

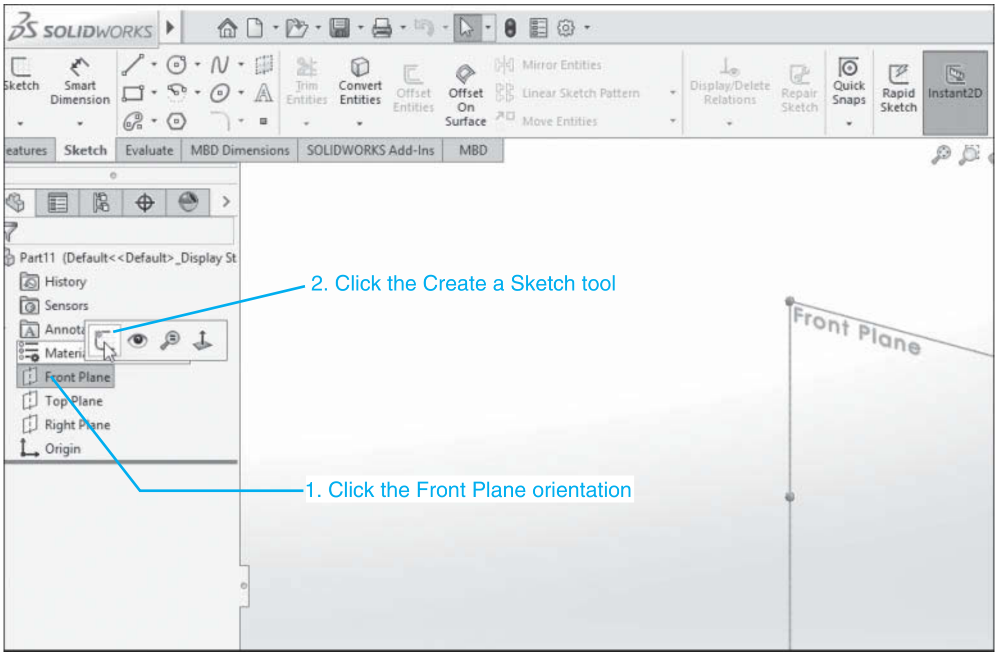
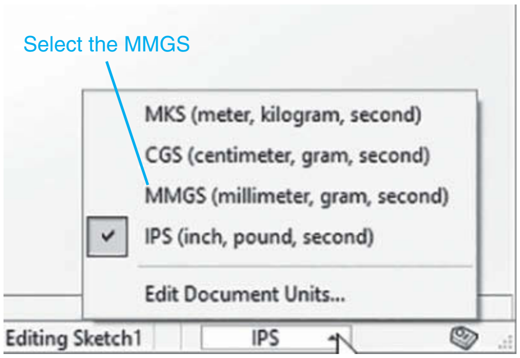



**Figure 1-25**

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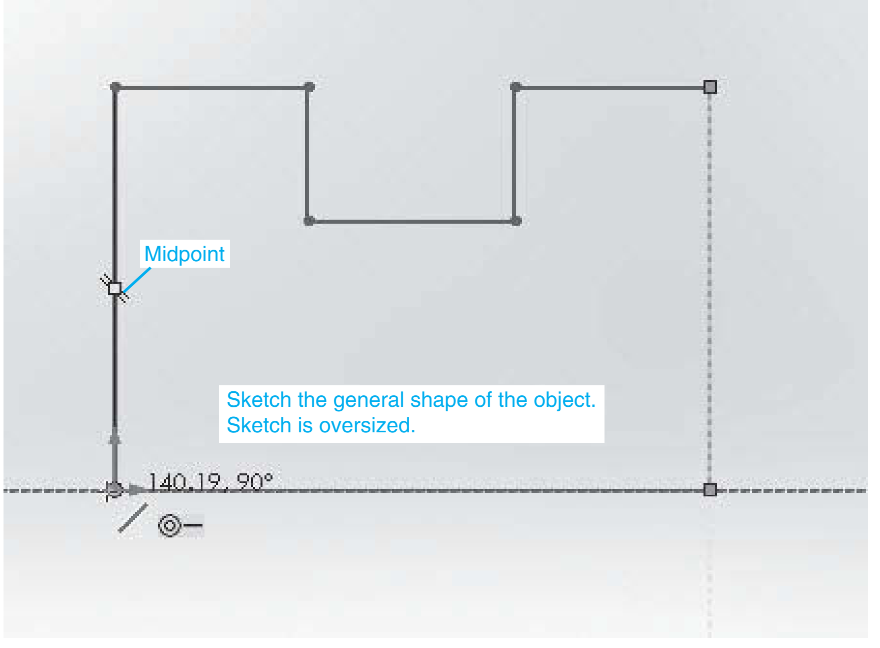
**NOTE**

The line command will generate a series of chain lines, where the endpoint of a sketched line becomes the starting point for the next line, until the line's endpoint is defined by pressing the <Esc> key or right-clicking the mouse and clicking the **Select** option.

See Figure 1-26.

**Figure 1-26**




**5** Sketch the general shape as shown.

**HINT**

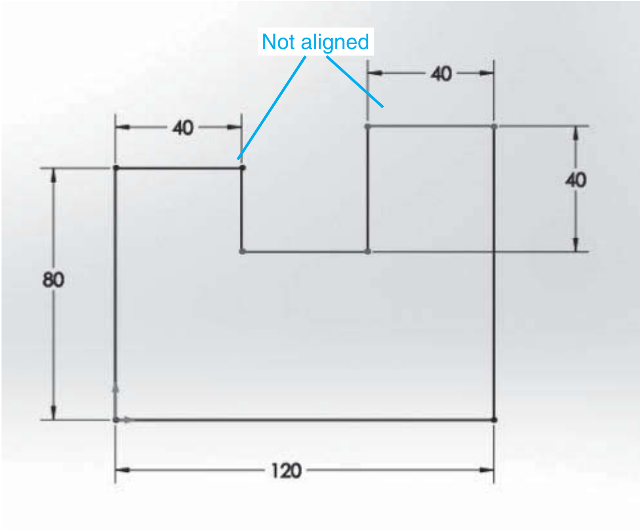
Make each line slightly larger than the stated dimension. Exact values are not required. Use the real-time length values to estimate the length of the longer lines.

Note the double circle relation icon that appears when the end of the last horizontal line drawn is located on the starting point of the first line. This is the **Concentric** relation icon. The Concentric icon indicates that the two points occupy the same location. The midpoint of the right-side vertical line is also defined.

**6** Click the **Smart Dimension** tool and dimension the shape as shown by clicking each line and entering the given dimensional value. See Figure 1-27.

SolidWorks is sensitive to how the dimensions are entered. See Figure 1-28. Note that when the vertical 40 dimension was added to the right side of the shape the adjacent horizontal 40 line moved upwards. This means that the two horizontal 40 lines are no longer aligned. The right 40 line must be fixed in place so that it remains aligned with the other horizontal 40 line when the vertical 40 dimension is added. The vertical 40 dimension will then move the bottom of the slot downwards.

Figure 1-27



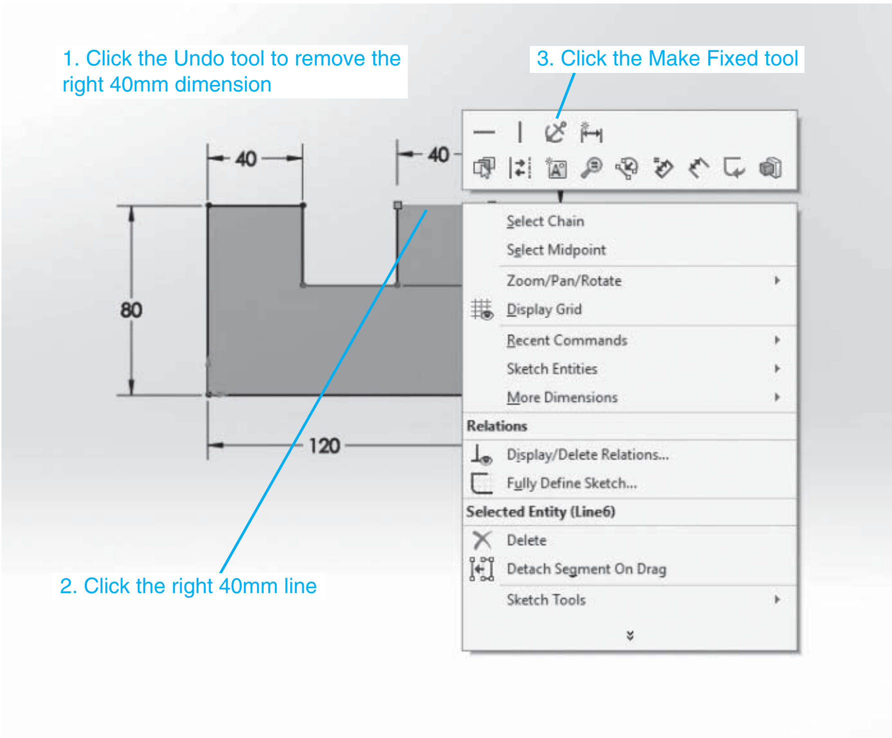

## To Fix a Line in Place

1. Use the **Undo** tool to remove the vertical 40 dimension.
2. Click the right horizontal 40 line.
3. Click the **Make Fixed** tool.

The **Make Fixed** tool's icon is an anchor. When the **Make Fixed** tool is activated, an anchor icon will appear below the line.

4. Use the **Smart Dimension** tool and add a vertical 40 dimension as shown.

Figure 1-28



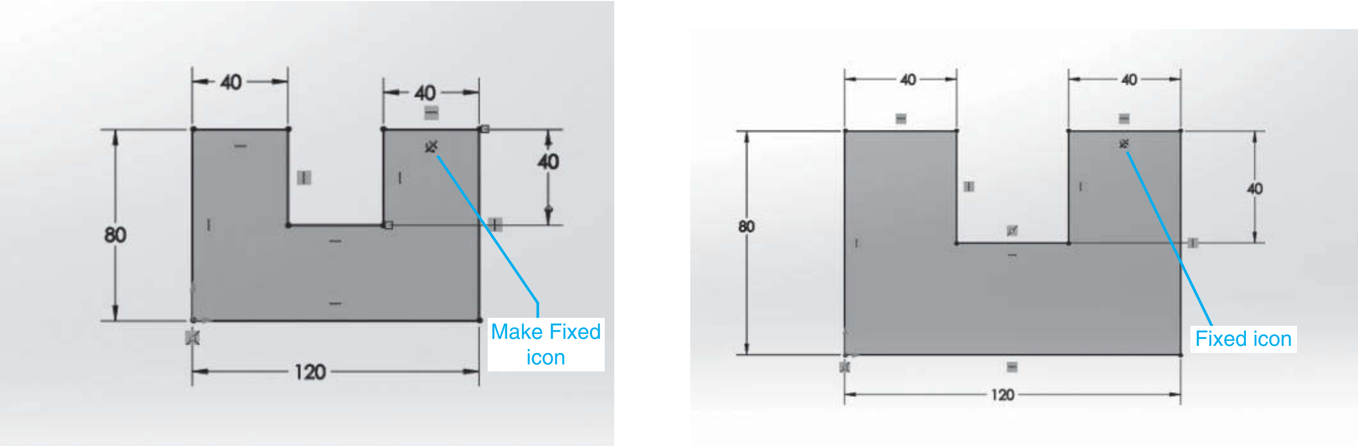



**Figure 1-28**
(*Continued*)

The horizontal line at the bottom of the slot will move, accepting the 40 dimensional changes. The two horizontal lines remain aligned.

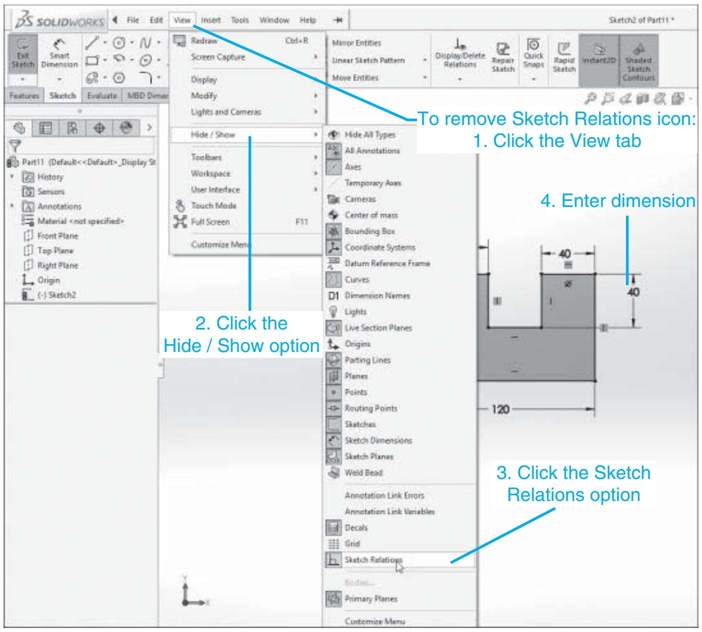
## Sketch Relations

Figure 1-29 shows a view of the object with and without **Sketch Relations**.

To remove the **Sketch Relations** icon:

1. Click the **View** tab at the top of the screen.
2. Click the **Hide/ Show** option.
3. Click the **Sketch Relations** option.

**Figure 1-29**




**NOTE**
2D shapes should always be fully defined before creating 3D models.

## 1-10 Creating 3D Models

The fully defined shape shown in Figure 1-29 can now be used to create a 3D model.

### To Create a 3D Model

1. Click the **Features** tab.
2. Click the **Extrude Boss/Base** tool.

See Figure 1-30. The shape will change orientation to the **Trimetric** format. The sketch was created on the **Front Plane**.

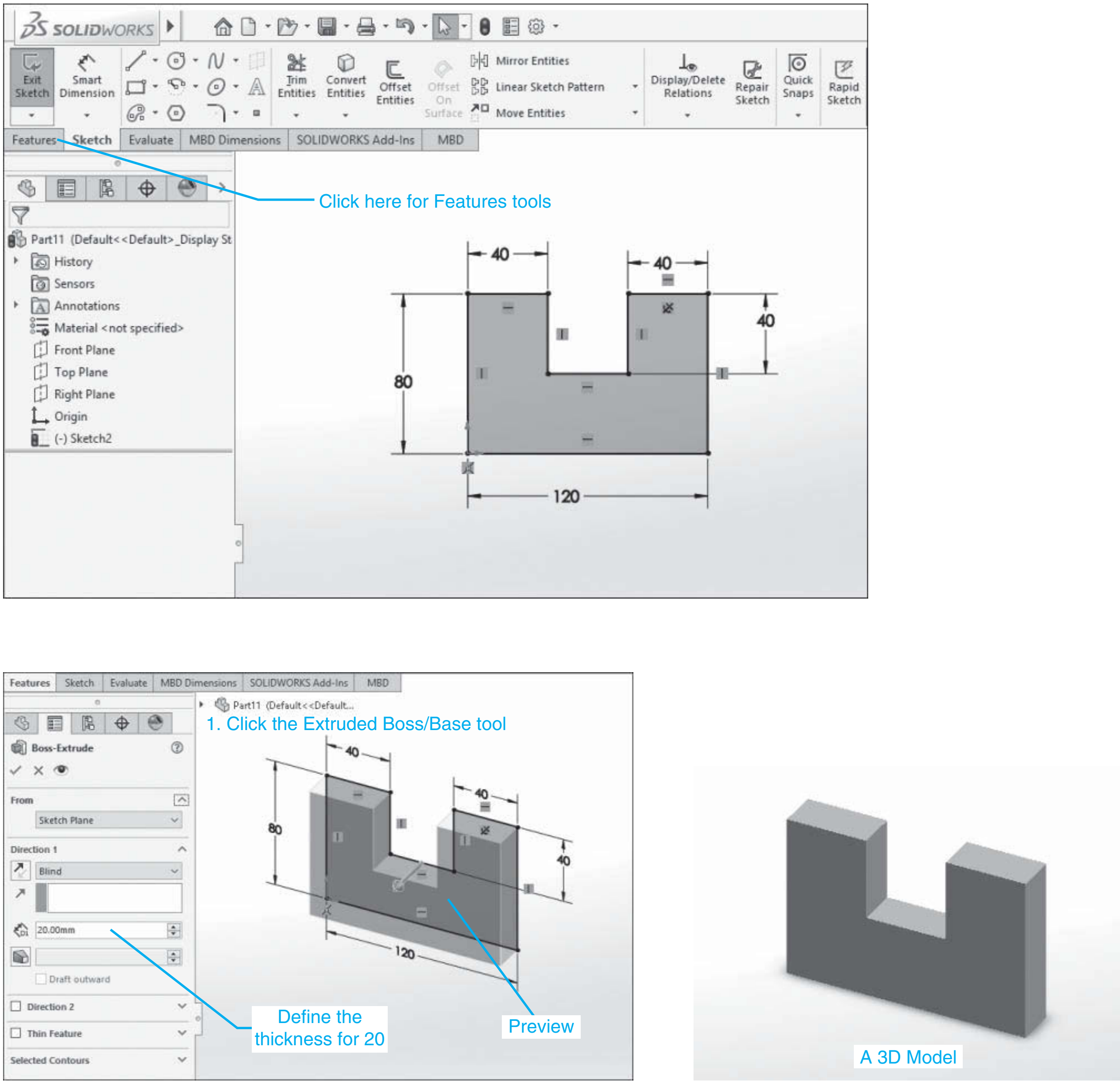



Figure 1-30

**3** Define the depth as **20 mm**.

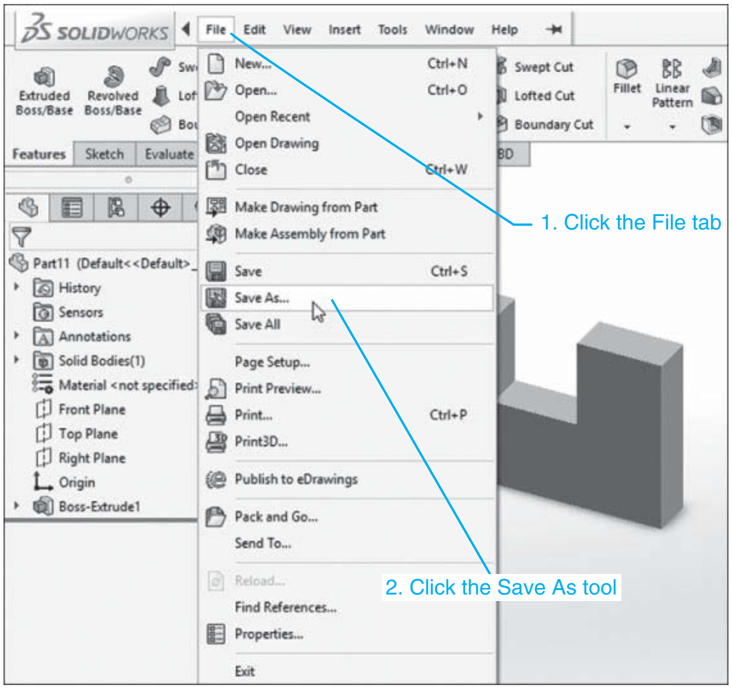
**4** Click the green **OK** check mark.

**5** Click the drawing screen.

## 1-11 Saving a Document

See Figure 1-31.

**Figure 1-31**




## To Save a Document

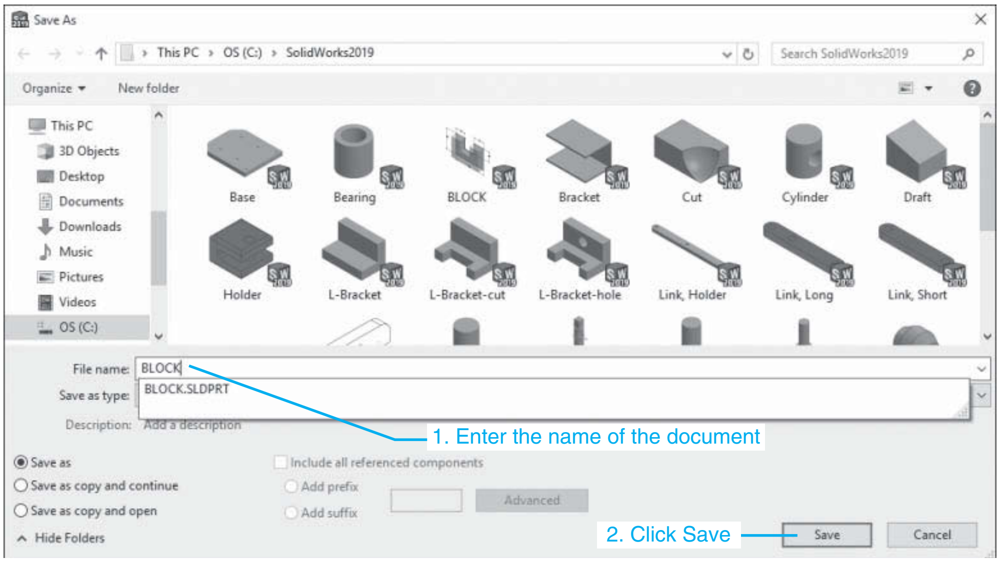
**1** Click the **File** tab at the top of the drawing screen.

A drop-down menu will appear.

**2** Click the **Save As** tool.

The **Save As** dialog box will appear. See Figure 1-32.

**Figure 1-32**



**3** Enter the **File name**.

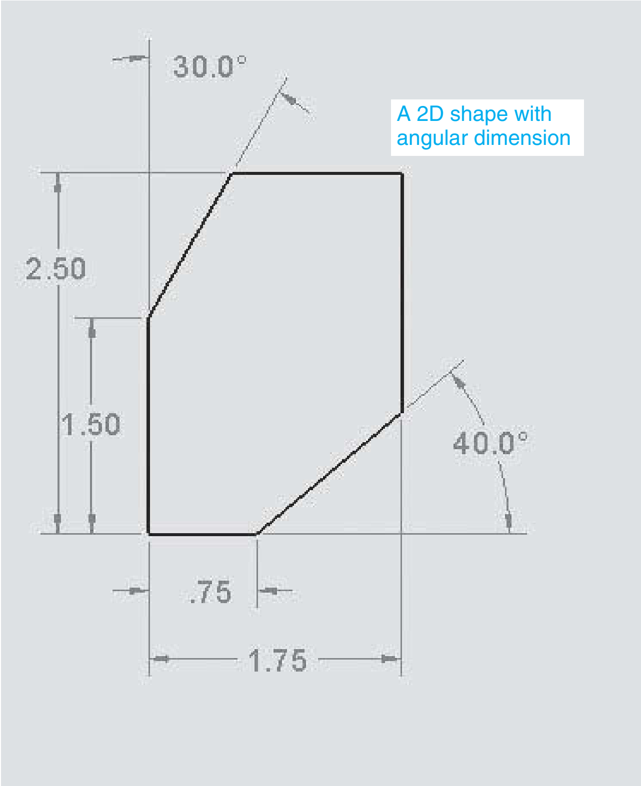
In this example the name **BLOCK** was used.

**4** Click the **Save** box.

## 1-12 Lines and Angles – Sample Problem SP1-2

Figure 1-33 shows a 2D shape that includes two angles. The dimensions are in inches. This section will show how to create the shape.

Figure 1-33




**1** Click the **Sketch** tab, the **Front Plane**, and the **Sketch** tool.

See Figure 1-34.

**2** Use the **Line** tool and sketch the approximate shape.

Start the first line of the shape on the origin. Sketch the shape slightly larger than the final shape.

**3** Add dimensions to the shape.

**4** Click the left vertical line and the left angled line and move the cursor away from the shape to create an angular dimension.

**5** Select a location for the dimension and click the mouse.

**6** Enter the angle value.

In this example the value is **30°**.

**7** Complete the remaining dimensions.

**8** Ensure that the shape is fully defined.

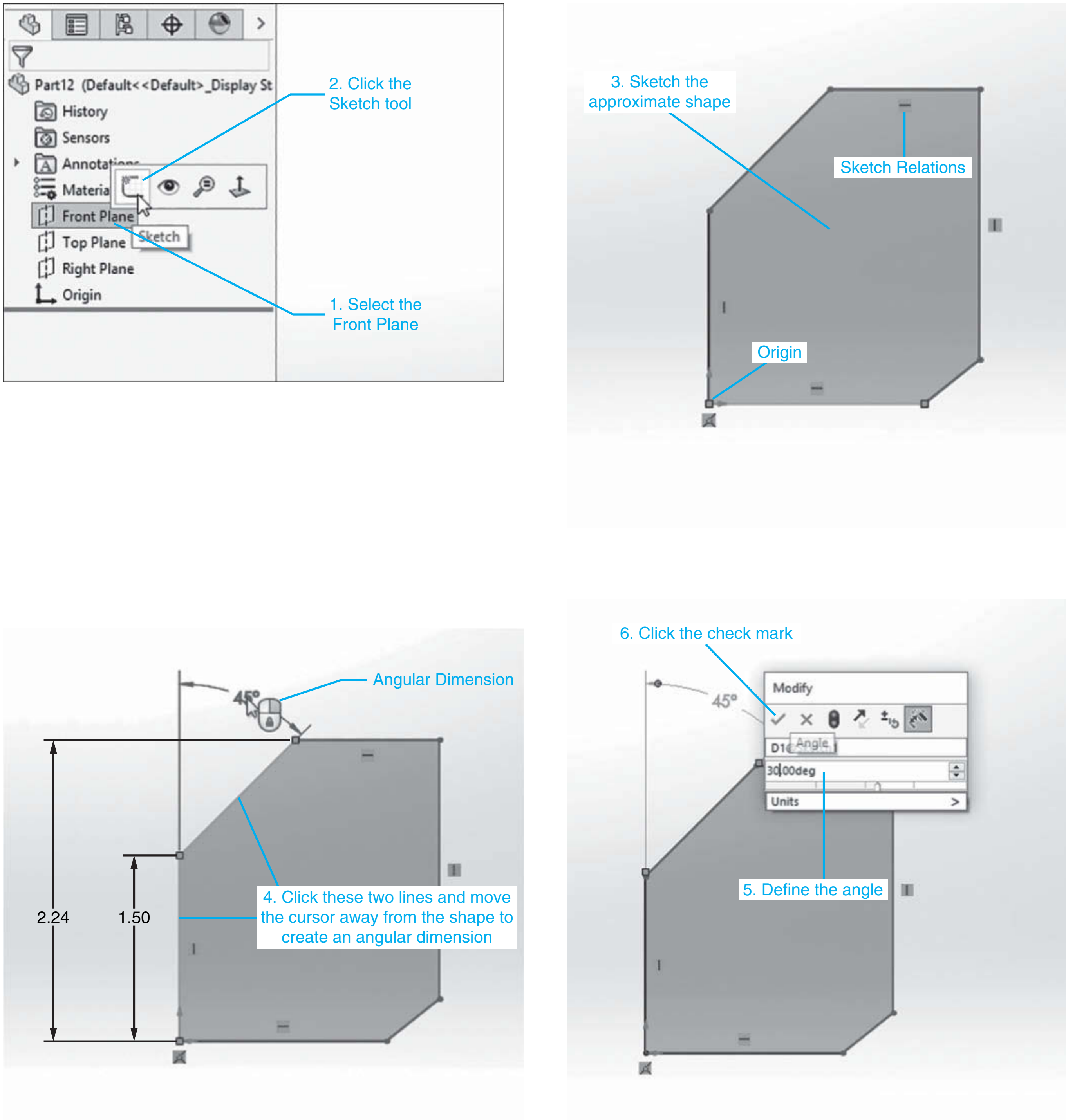

Part12 (Default<<Default>_Display St
History
Sensors
Materia
Front Plane
Top Plane
Right Plane
Origin
Sketch
2. Click the Sketch tool
1. Select the Front Plane
3. Sketch the approximate shape
Sketch Relations
Origin
Angular Dimension
45°
2.24
1.50
4. Click these two lines and move the cursor away from the shape to create an angular dimension
6. Click the check mark
45°
Modify
Angle
30.00deg
Units
5. Define the angle

**Figure 1-34**

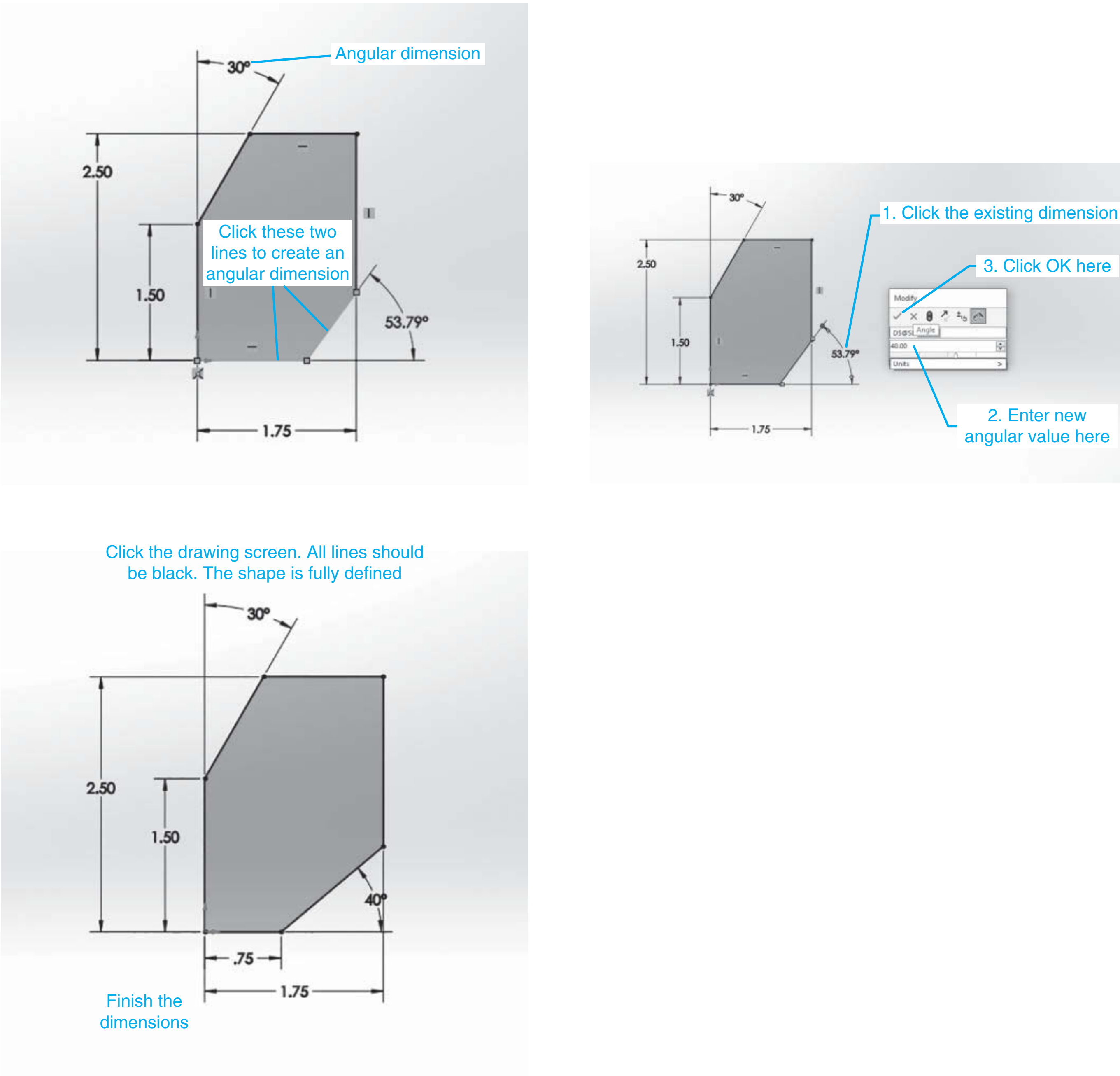



**Figure 1-34**
(*Continued*)

**9** Click the **Features** tab, the **Extrude Boss/Base** tool, and define the depth.

In this example, a depth of **0.50** was entered. See Figure 1-35.

**10** Click the green **OK** check mark and then click the drawing screen.

All the lines in the shape should be black indicating the shape is fully defined. See Figure 1-36.

Figure 1-35

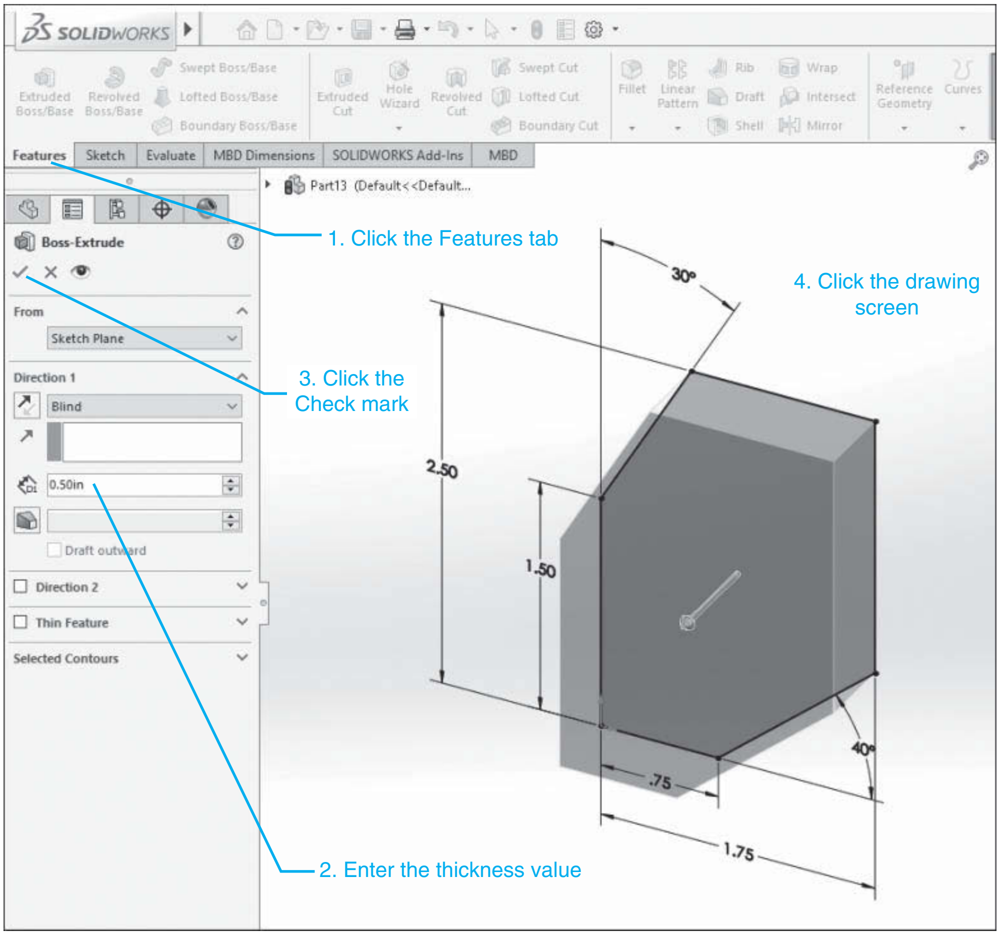
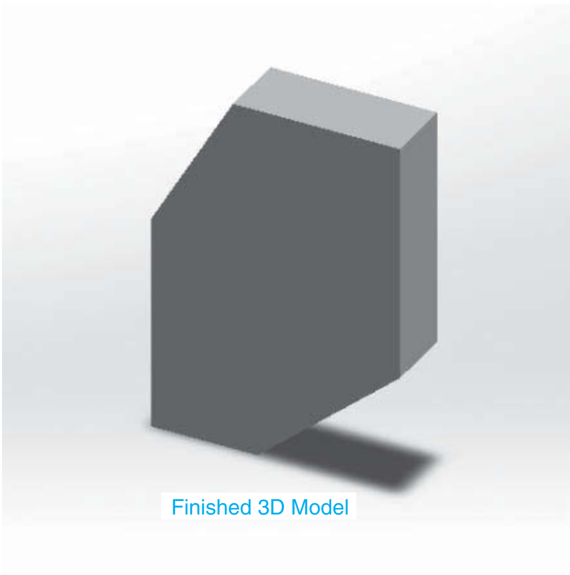



Figure 1-36



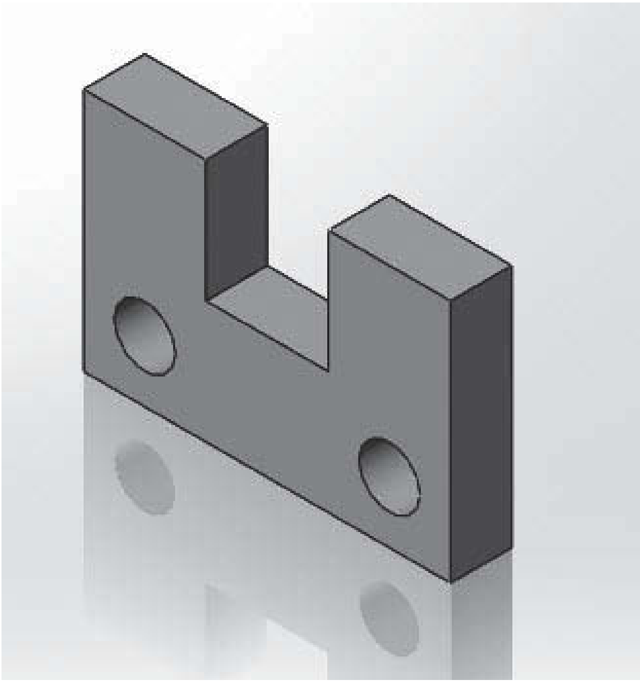
## 1-13 Holes

There are several different ways to create holes using SolidWorks. Most holes are created using the **Hole Wizard** tool. Hole Wizard is explained in Chapter 3. For purposes of this introductory chapter, holes will be created using the **Circle** and **Extrude Cut** tools. A circle will be created and then cut through the 3D shape. All holes will be simple through holes; that is, they will go completely through the shape.

### To Create a Hole

Figure 1-37 shows the 3D shape created in Sample Problem SP1-1. Two Ø20.0 holes have been added.

Figure 1-37



1. Click the **File** tool heading at the top of the screen and click the **Open** option, or click the **Open** tool.
2. Locate and click the **BLOCK** file created and saved in the last section.

   See Figure 1-38. In this example the file was located on the C: drive under the file heading **SolidWorks 2019**.
3. Click the **BLOCK** file, and click **Open**.

   The BLOCK will appear on the screen. See Figure 1-39.
4. Click the **View Orientation** tool and select the **Normal To** option.

   This will create a view from an orientation point 90° to the surface. This is called a *normal* view. See Figure 1-40.

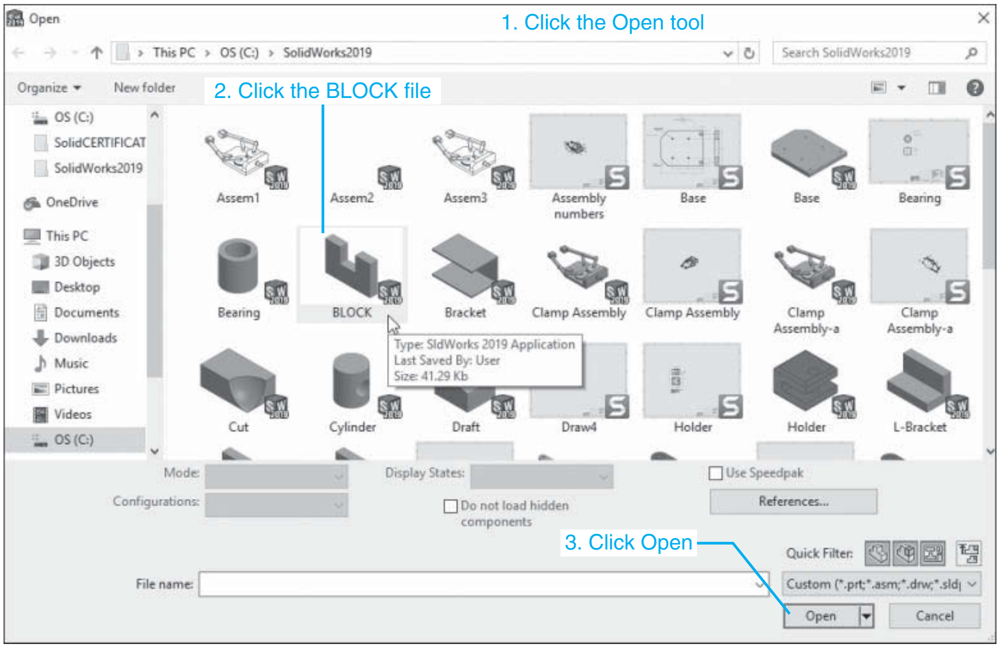
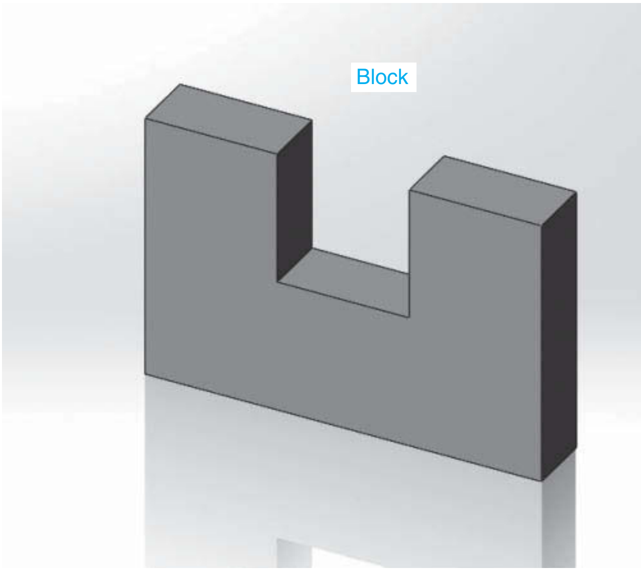



**Figure 1-38**

**Figure 1-39**




For this exercise we will work in a three-dimensional isometric plane. See Figure 1-41.

**5** Again click the **View Orientation** icon, but this time select the small hexagonal surface to create an **Isometric** orientation.

Figure 1-40

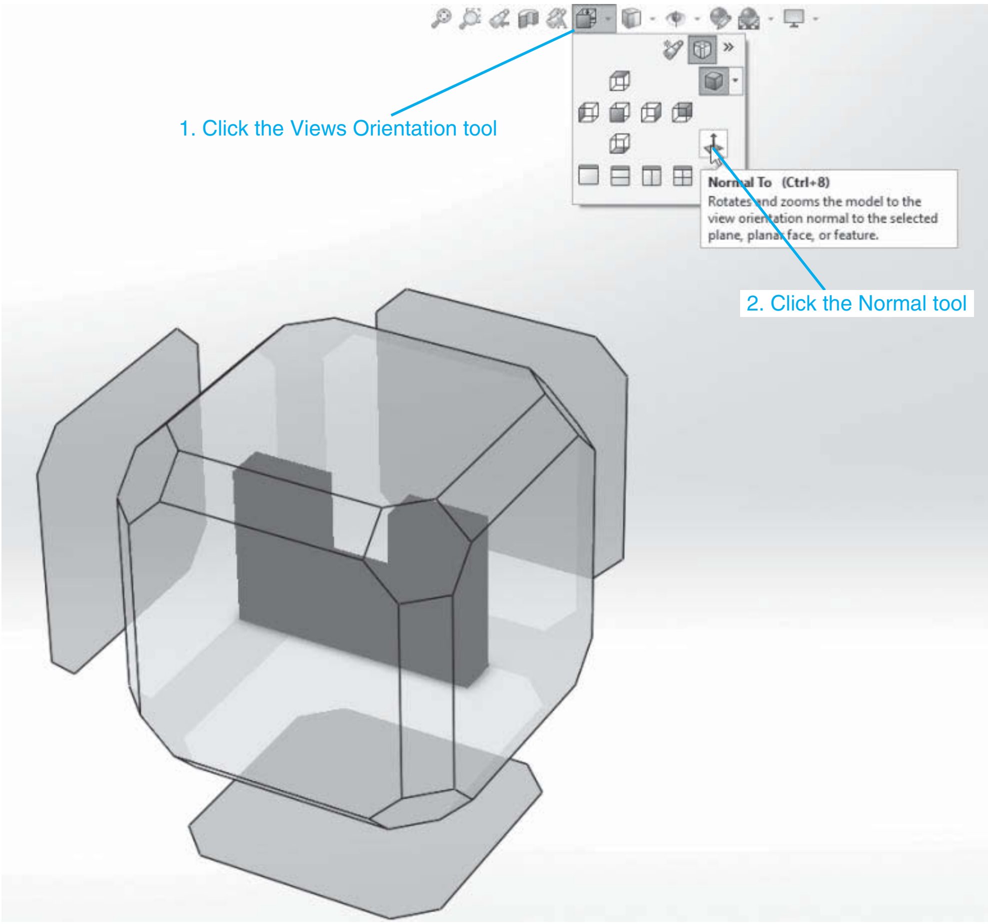



Figure 1-41

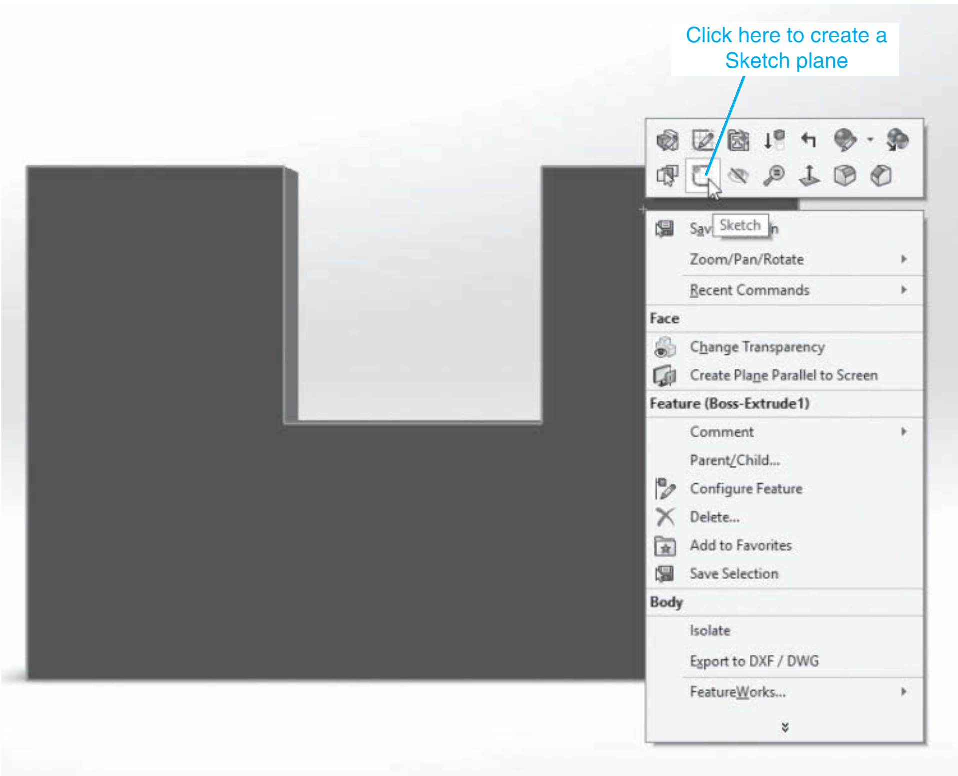
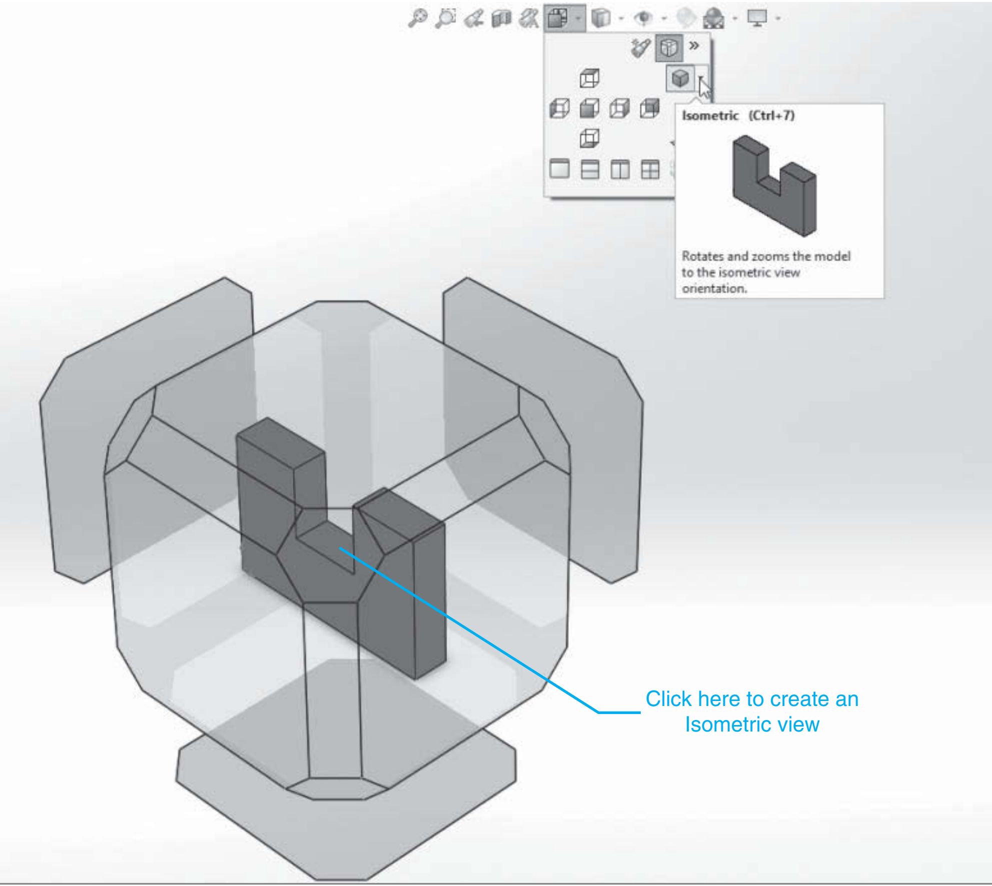
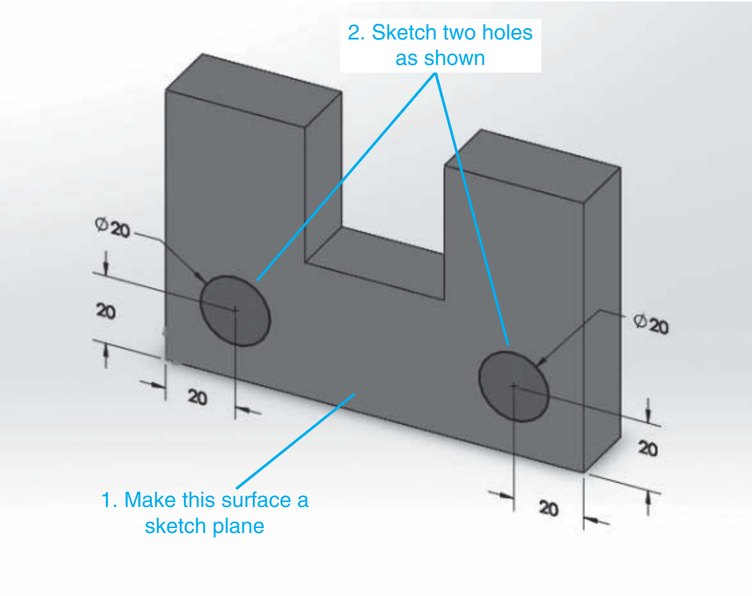


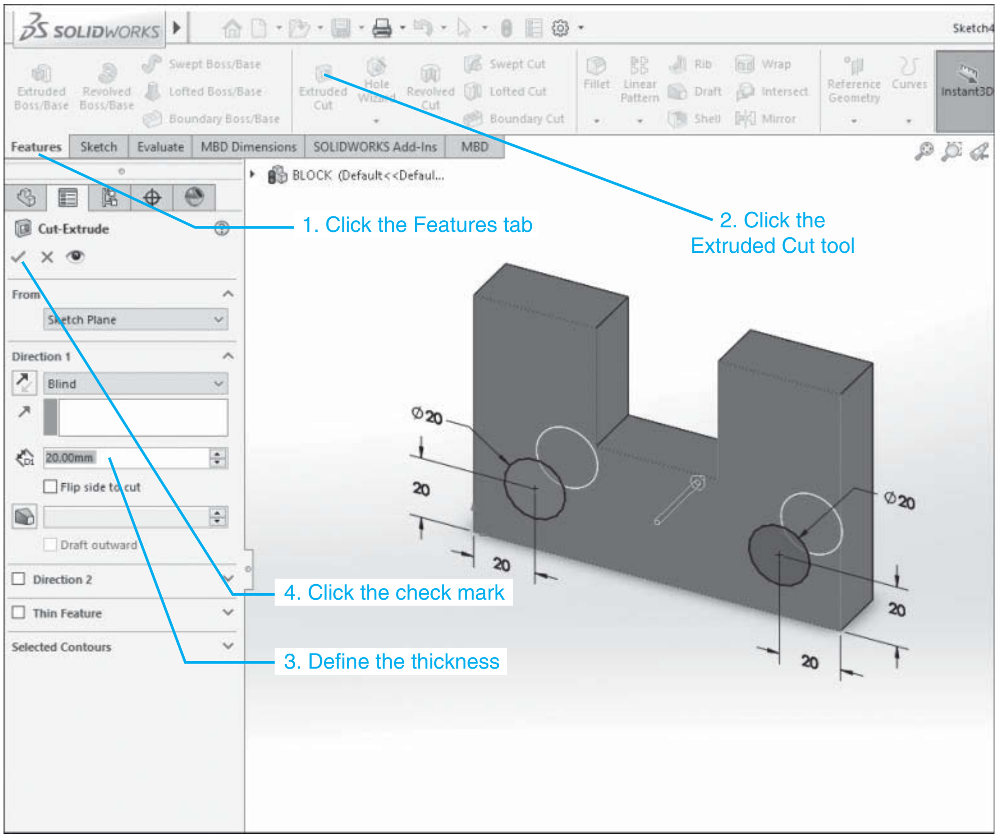
**Figure 1-41**
(*Continued*)

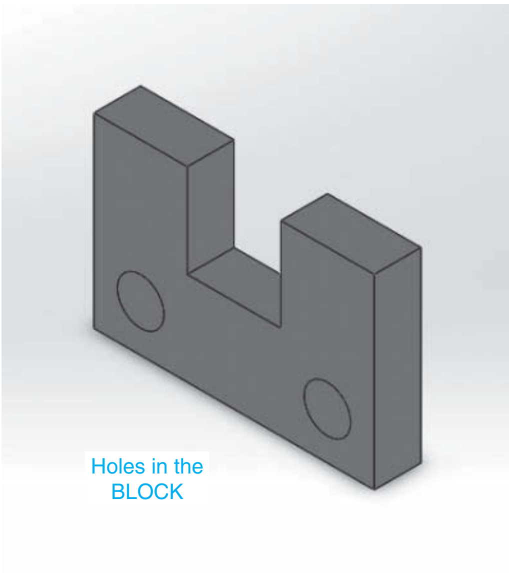



6. Create a **Sketch plane** on the front surface of the BLOCK, then use the **Circle** tool and add two circles using the given dimensions.
7. Click the **Features** tab, and the **Extruded Cut** tool.
8. Set the cut depth for **20**.

**Figure 1-41**
(*Continued*)




The **Extruded Cut** tool should automatically select the two circles. If it does not, click the circles. A preview should appear.

**9** Click the green **OK** check mark.

**10** Click the drawing screen.

The holes should appear in the shape.

# Chapter Projects

## Project 1-1:

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

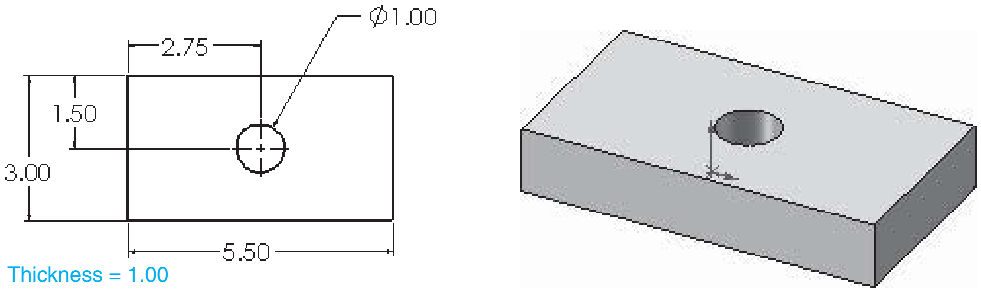



**Figure P1-1**
INCHES

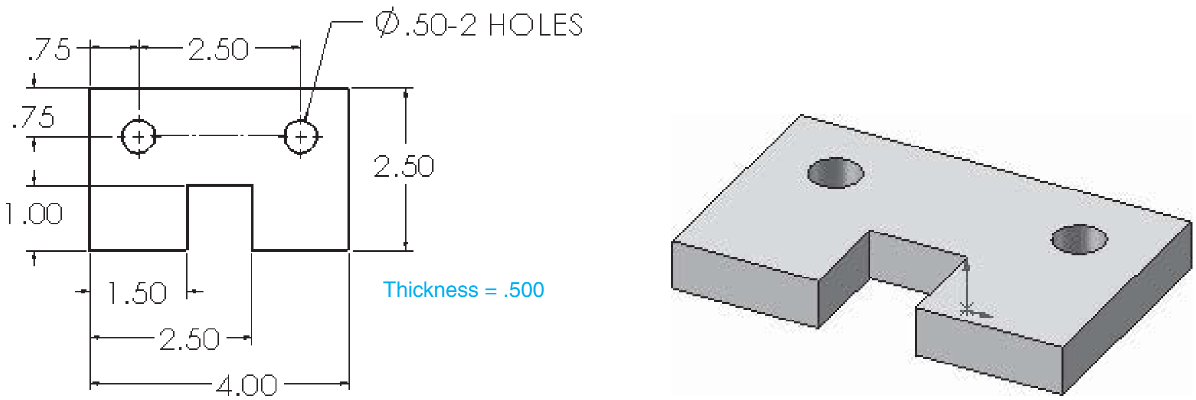



**Figure P1-2**
INCHES

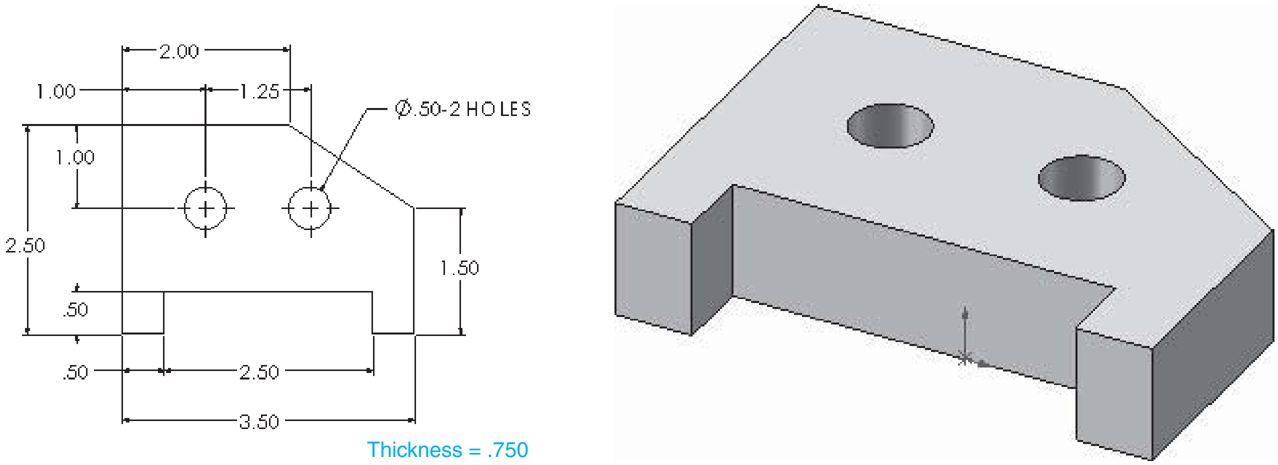



**Figure P1-3**
INCHES

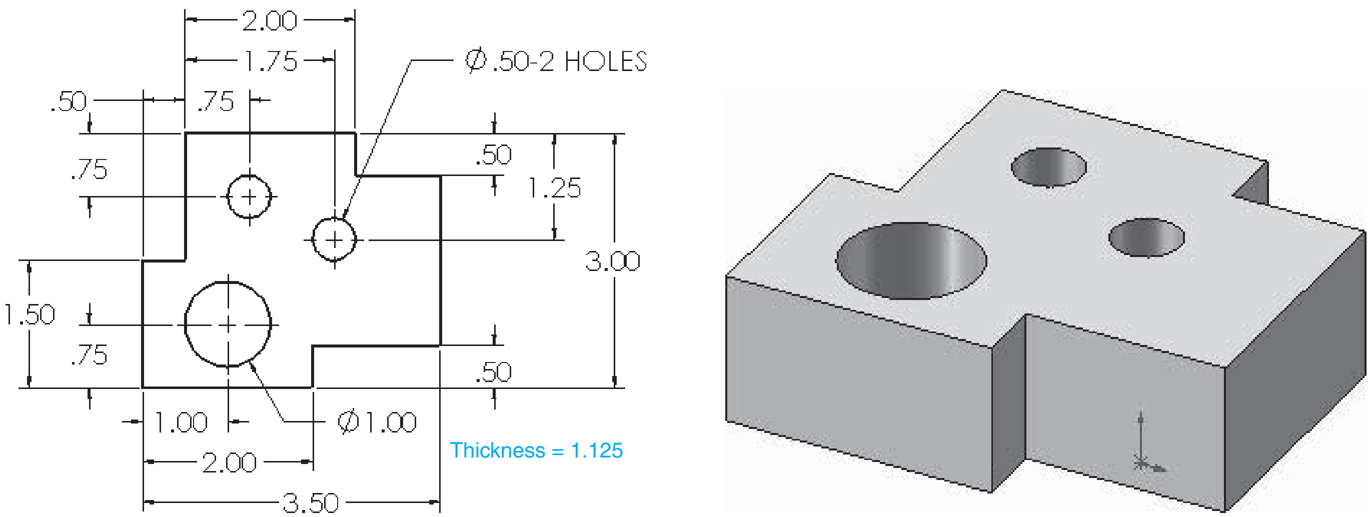



**Figure P1-4**
INCHES

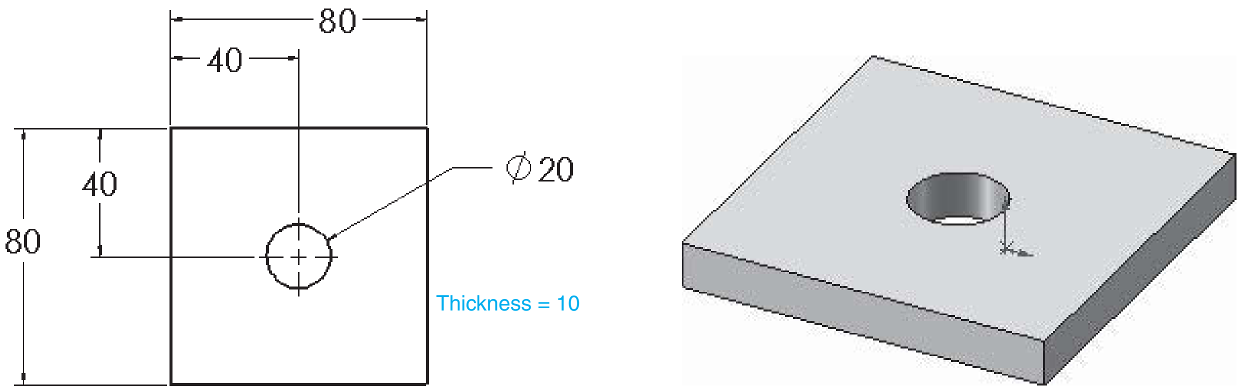



**Figure P1-5**
MILLIMETERS

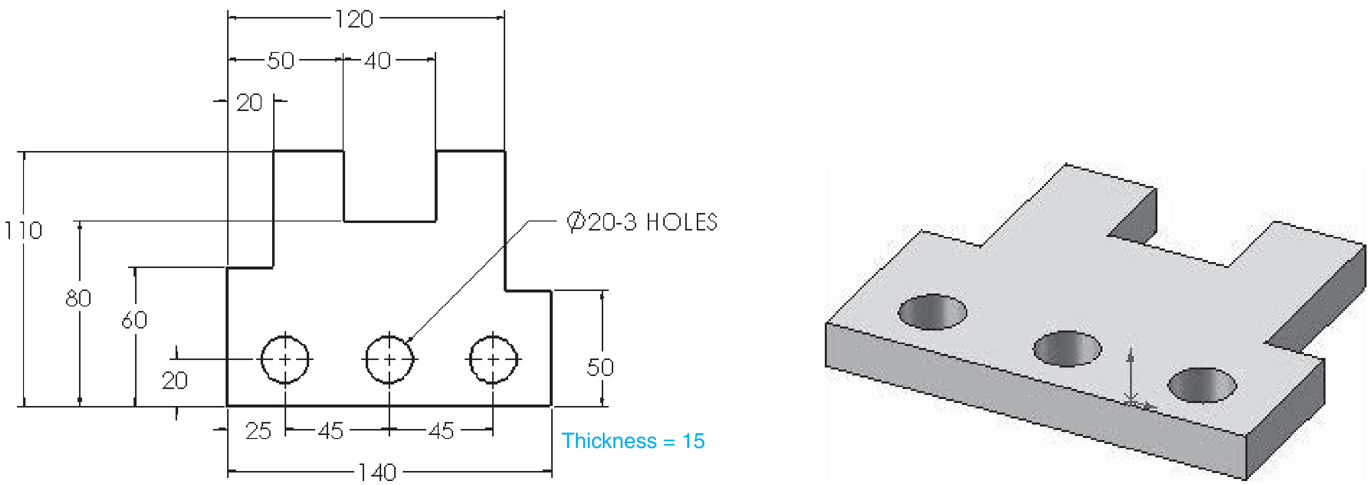



**Figure P1-6**
MILLIMETERS

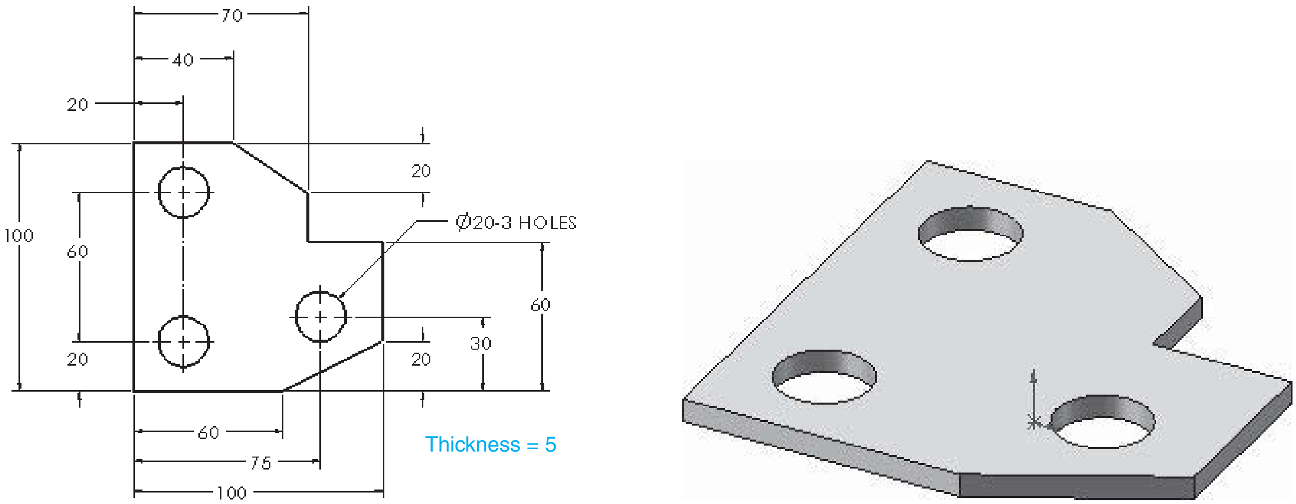



**Figure P1-7**
MILLIMETERS

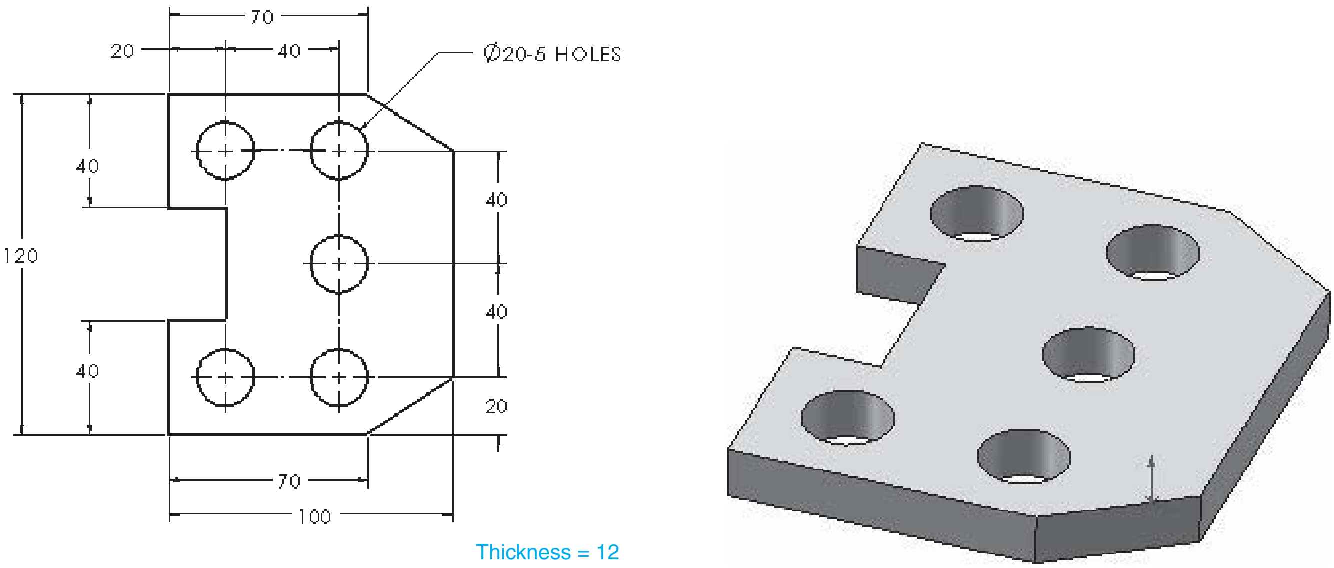



**Figure P1-8**
MILLIMETERS

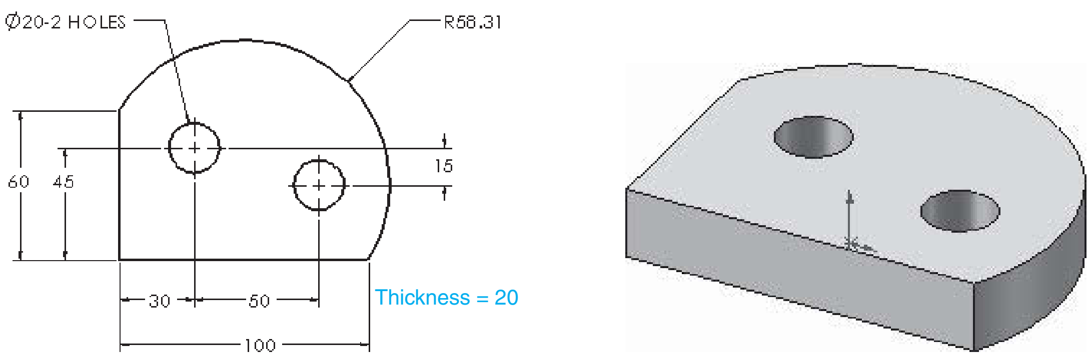
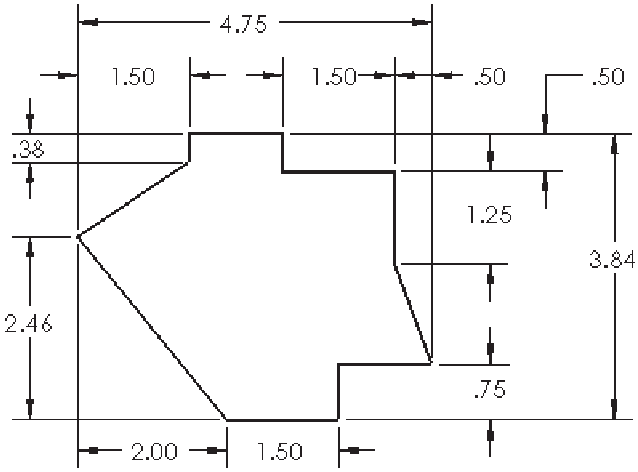
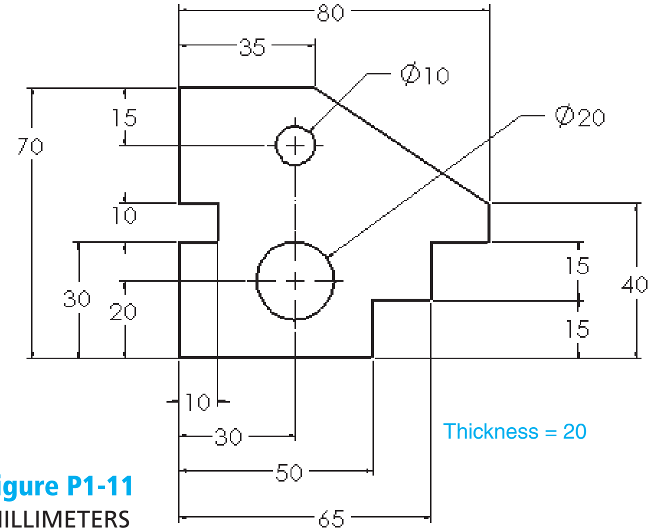



**Figure P1-9**
MILLIMETERS




Thickness = .60

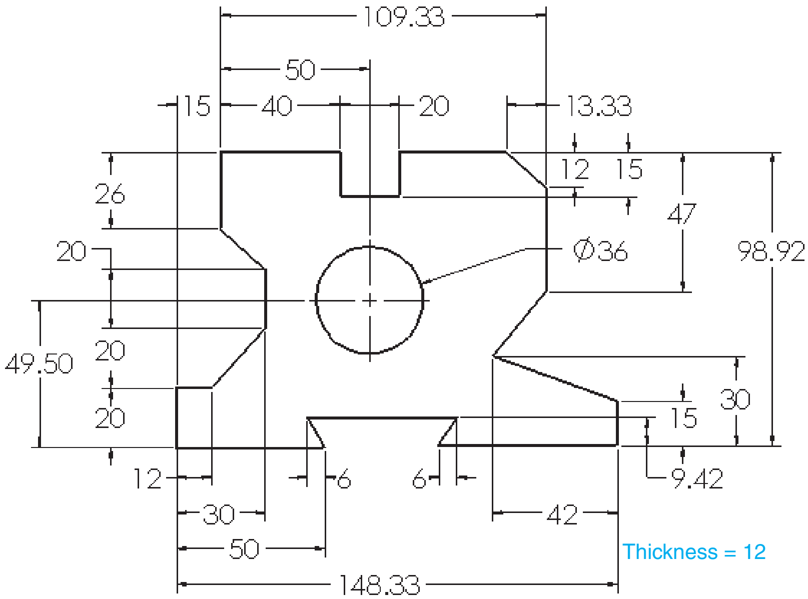
**Figure P1-10**
INCHES




Thickness = 20

**Figure P1-11**
MILLIMETERS

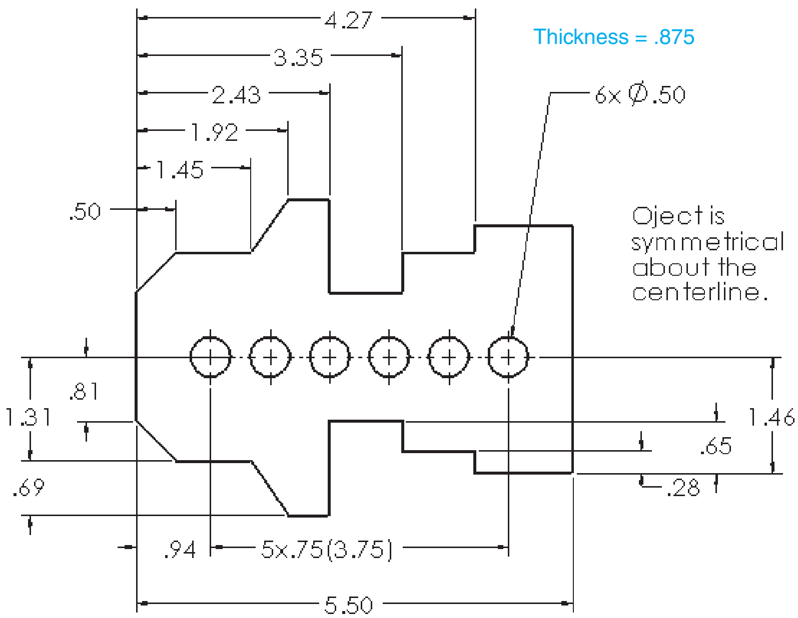
**Figure P1-12**
MILLIMETERS




Thickness = 12

**Figure P1-13**
INCHES

Thickness = .875



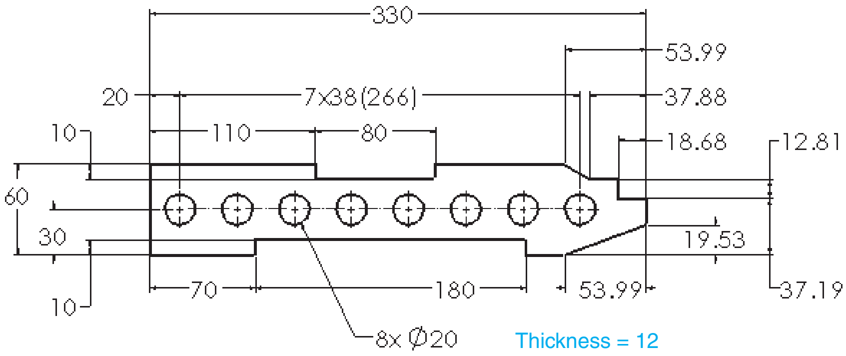
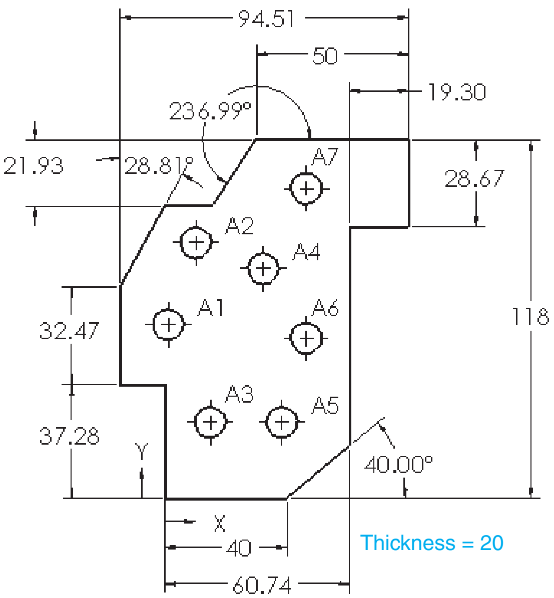



**Figure P1-14**
MILLIMETERS




| 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

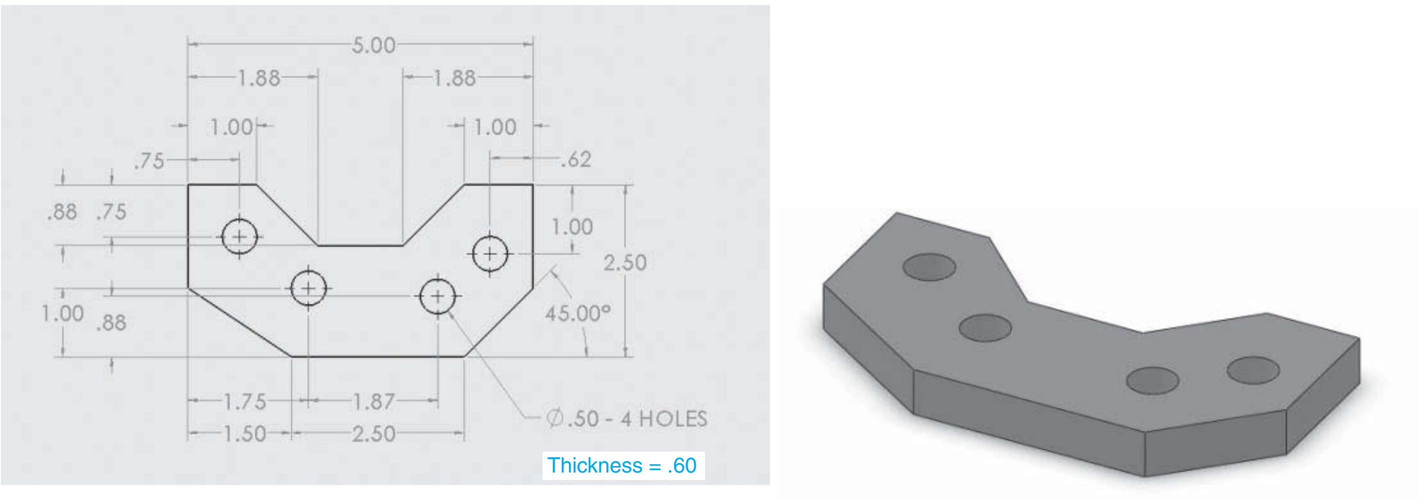
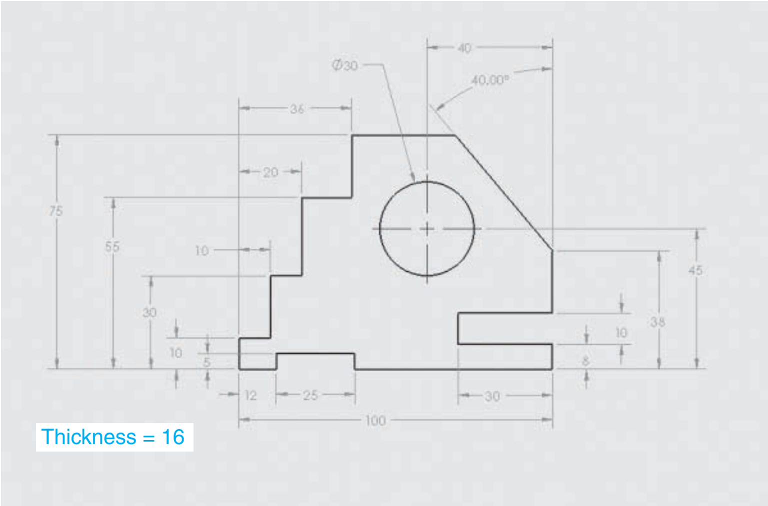



**Figure P1-16**
INCHES



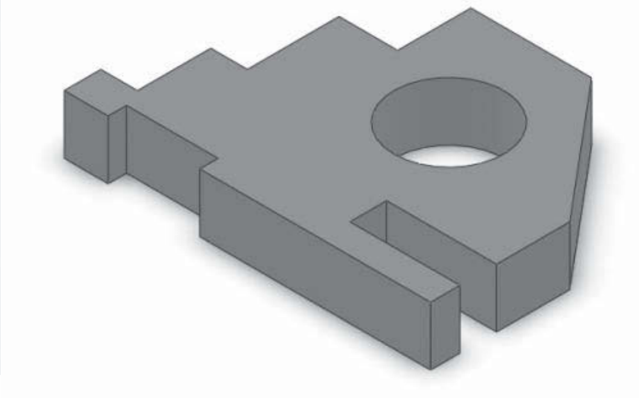
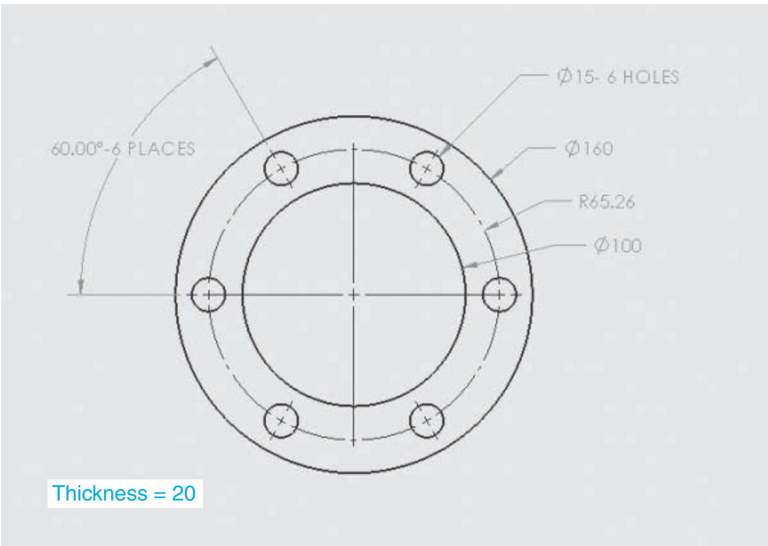



**Figure P1-17**
MILLIMETERS



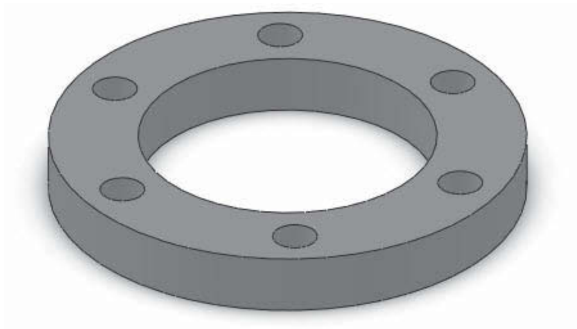



**Figure P1-18**
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