ENGINEERING DESIGN AND GRAPHICS WITH SOLIDWORKS® 2023

JAMES D. BETHUNE
NATHAN BROWN

Engineering Design and Graphics with SolidWorks® 2023

James D. Bethune
Nathan Brown

Pearson

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Preface

This book shows and explains how to use SolidWorks® 2023 to create engineering designs and drawings. 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.

Chapters 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 entities 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 44 exercise problems (18 in Ch1 and 26 in Ch2) using both inches and millimeters so that students can practice 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. Seven exercise problems (P4-142 to P4-149) 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 topic 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.

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.

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Nathan Brown

Contents

7 chapterseven

Dimensioning

CHAPTER OBJECTIVES

- Dimension objects
- Learn ANSI standards and conventions
- Dimension different shapes and features
- Learn fundamentals of 3D dimensioning

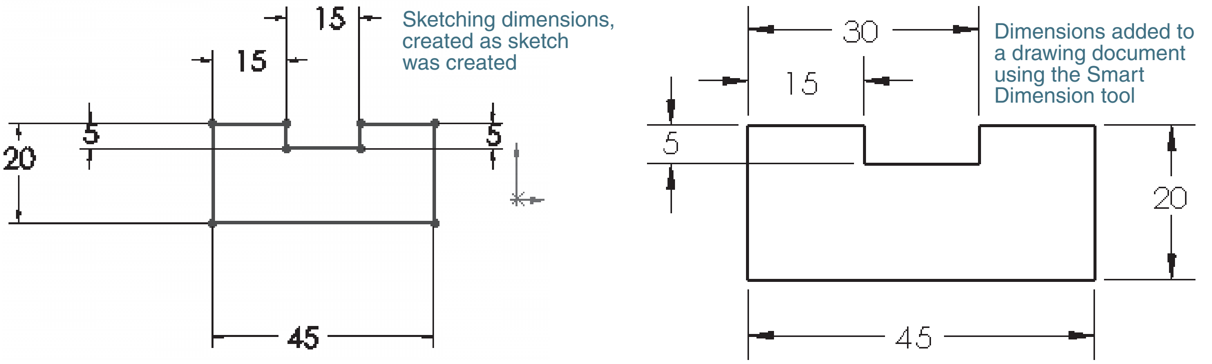
7-1 Introduction

Dimensions are added to SolidWorks on **Drawing** documents. Dimensions will appear in **Part** documents, but these are construction dimensions. These sketch dimensions are used to create a part and are used when a sketch is edited. They may be modified as the part is being created using the **Smart Dimension** tool. They will not appear on the finished model or in **Assembly** documents.

Figure 7-1 shows a dimensioned shape. The drawing on the left in Figure 7-1 shows the sketch dimensions that were used as the part was being created. The drawing on the right in Figure 7-1 shows dimensions that were created using the **Smart Dimension** tool in a **Drawing** document. These are defining dimensions and will appear on the working drawings. This chapter will show how to apply these types of dimensions.

SolidWorks has ANSI Inch and ANSI Metric dimensions available. Other dimensioning systems such as ISO also are available. This text is in compliance with ANSI standards.

Figure 7-1



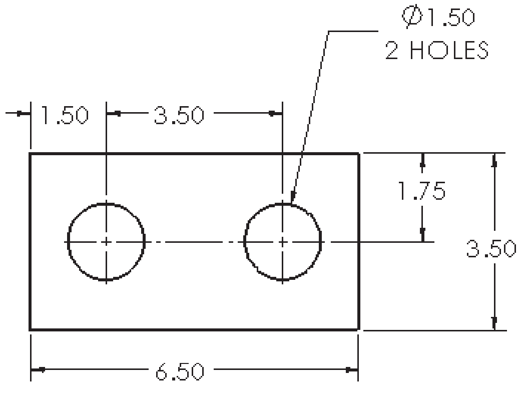
7-2 Terminology and Conventions—ANSI

Dimensions are added to drawings to define the part and guide manufacturing. General rules and conventions are used to dimension a drawing in a complete, orderly, and succinct manner.

Common Terms

Figure 7-2 shows both ANSI- and ISO-style dimensions. The terms apply to both styles.

Figure 7-2



Dimension lines: In mechanical drawings, lines between extension lines that end with an arrowhead and include a numerical dimensional value located within the line.

Extension lines: Lines that extend away from an object and allow dimensions to be located off the surface of an object.

Leader lines: Lines drawn at an angle, not horizontal or vertical, that are used to dimension specific shapes such as holes. The start point of a leader line includes an arrowhead. Numerical values are drawn at the end opposite the arrowhead.

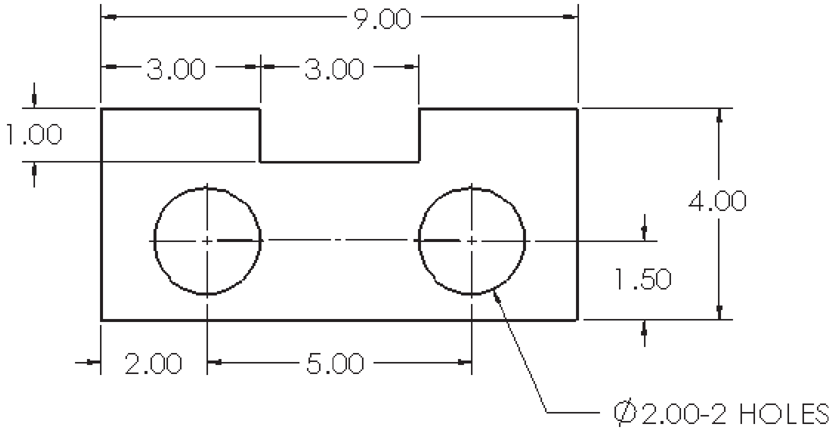
Linear dimensions: Dimensions that define the straight-line distance between two points.

Angular dimensions: Dimensions that define the angular value, measured in degrees, between two straight lines.

Dimensioning Conventions

There are general guidelines you should follow when dimensioning drawings. Figure 7-3 shows some of the following guidelines applied to a dimensioned part.

Figure 7-3



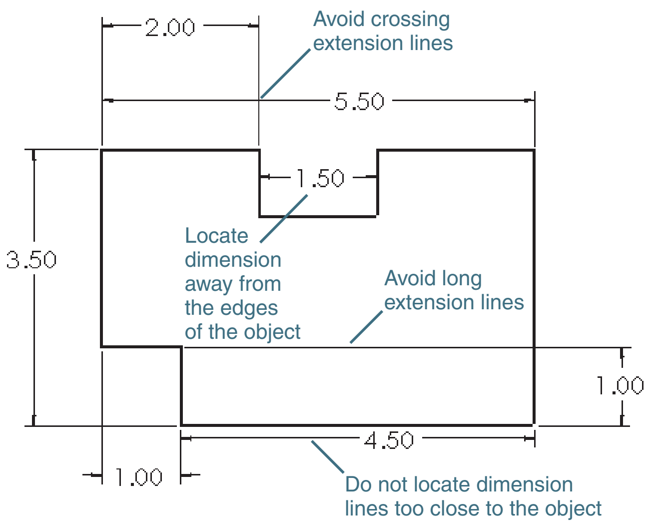
- Dimension lines should be drawn evenly spaced; that is, the distance between dimension lines should be uniform. A general rule of thumb is to locate dimension lines about 1/2in. or 15mm apart.
- There should be a noticeable gap between the edge of a part and the beginning of an extension line. This serves as a visual break between the object and the extension line. The visual difference between the line types can be enhanced by using different colors for the two types of lines.
- Leader lines are used to define the size of holes and should be positioned so that the arrowhead points toward the center of the hole.
- Centerlines may be used as extension lines. No gap is used when a centerline is extended beyond the edge lines of an object.
- Align dimension lines whenever possible to give the drawing a neat, organized appearance.

Common Errors to Avoid

See Figure 7-4.

Figure 7-4

Some common errors

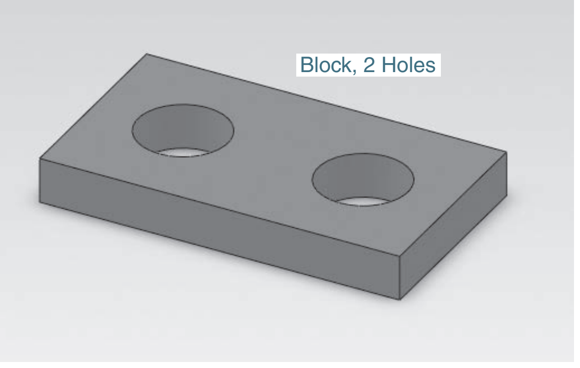


- Avoid crossing extension lines. Place longer dimensions farther away from the object than shorter dimensions.
- Do not locate dimensions within cutouts; always use extension lines.
- Do not locate any dimension close to the object. Dimension lines should be at least 1/2in. or 15mm from the edge of the object.
- Avoid long extension lines. Locate dimensions in the same general area as the feature being defined.

7-3 Adding Dimensions to a Drawing

Figure 7-5 shows a part that includes two holes. This section will explain how to add dimensions to the part. The part was drawn as a **Part** document and saved as **BLOCK, 2 HOLES**. See Figure 7-10 for the part's dimensions. The part is 0.50 thick. Save the part and start a new **Drawing** document.

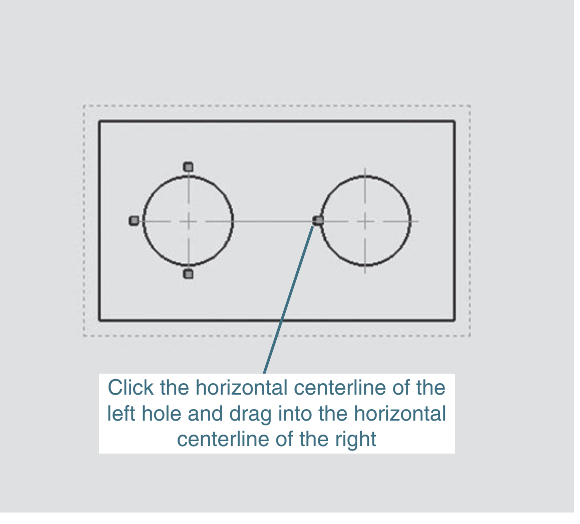
Figure 7-5



1. Click **New**, **Drawing**, and **OK** to start a new drawing. Use a **B (ANSI) Landscape** sheet size.
2. Click the **View Layout** tab, **Model View**, and create a top view of the **BLOCK, 2 HOLES** part.

   In this example we will work with only one view. See Figure 7-6.

Figure 7-6



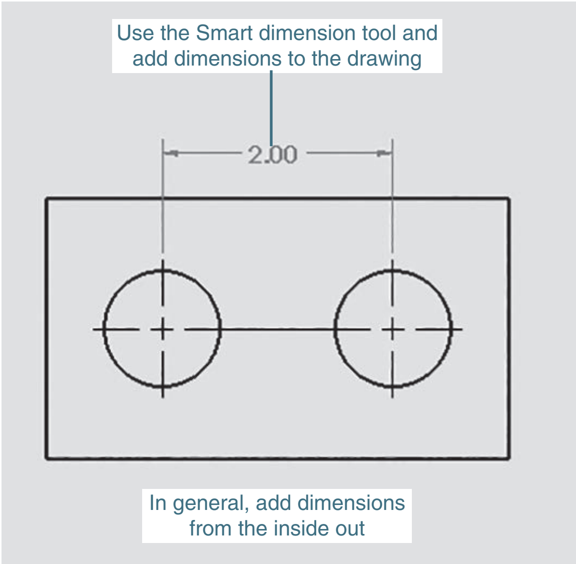
Extend the horizontal center mark from the left hole to the horizontal centerline of the right hole.

3 Click the horizontal centerline of the left hole. Small blue boxes will appear on the center mark.

4 Click and drag the horizontal centerline from the left hole to the right hole.

The holes now have the same horizontal centerline so only one vertical dimension can be used to define the hole's location. See Figure 7-7.

Figure 7-7



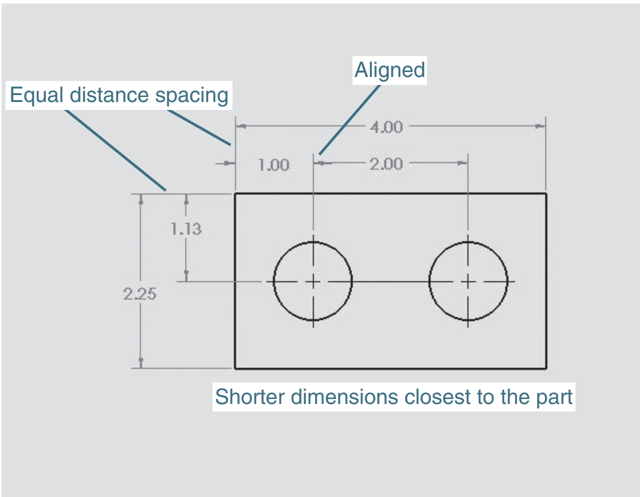
TIP

Centerlines can be extended by first clicking them and then dragging an endpoint to a new location.

5 Use the **Smart Dimension** tool and add the horizontal and vertical dimensions as shown.

See Figure 7-8.

Figure 7-8



Note that the dimension values for the vertical dimensions are written horizontally. This is in compliance with ANSI standards. For this example, the Century Gothic font was made bold with 14-point height.

> **RULE**
> Keep dimension lines aligned and evenly spaced.

6 Click the **Hole Callout** tool located on the **Annotation** panel, click the edge of the left hole, and move the cursor away from the hole.

Note that the leader arrow always points to the center of the hole.

7 Select a location off the surface of the part and click the mouse.

> **RULE**
> Never locate dimensions on the surface of the part.

See Figure 7-9. The word THRU is optional. Some companies require it and some do not.

Figure 7-9

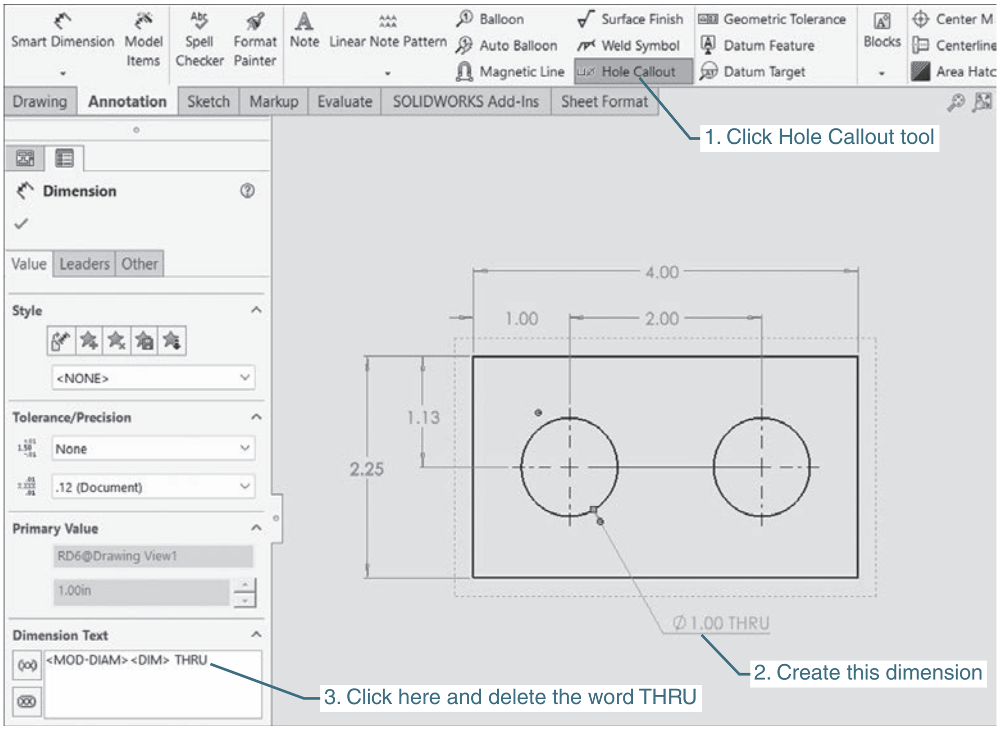
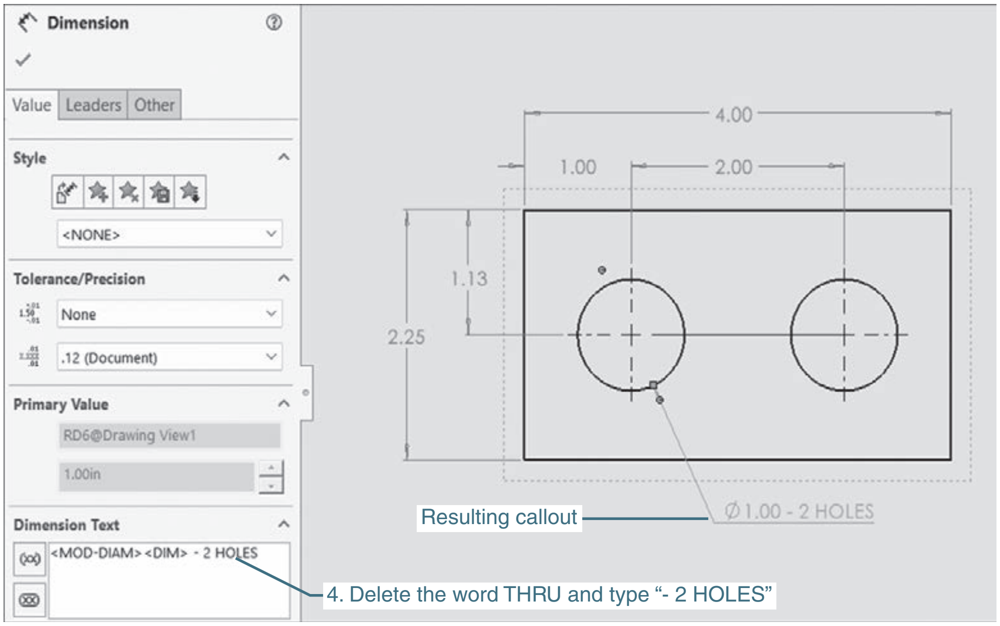


Figure 7-9
(*Continued*)



8. Go to the **Dimension PropertyManager** at the left of the screen, locate the cursor in the **Dimension Text** box, and delete the word THRU.

   The text already in the box defines the hole's diameter.

9. Move the cursor to the end of the existing text line, and type **- 2 HOLES**.

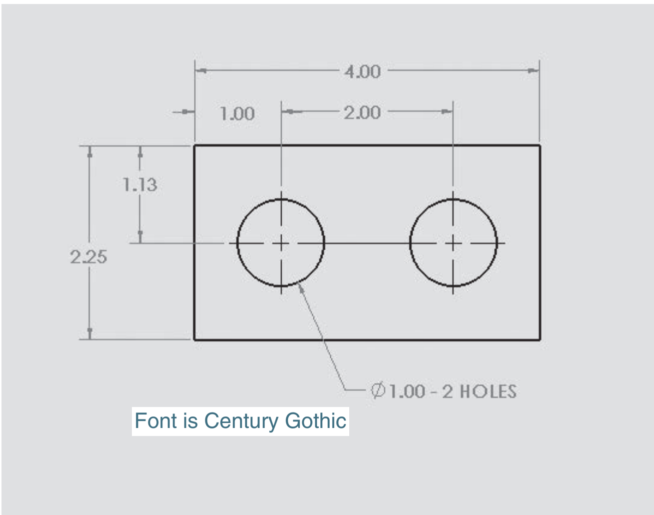
10. Click the green **OK** check mark.

    Move the dimensions if needed to create neat, uniform dimensions.

    See Figure 7-10.

11. Save the drawing.

Figure 7-10



TIP

Dimensions can be relocated by clicking and dragging the dimension text.

Controlling Dimensions

Various aspects of dimensions can be edited, such as text height, arrow location, and text values.

1. Click the **Options** tool at the top of the screen.

The **Documents Properties - Drafting Standard** dialog box will appear. See Figure 7-11.

2. Click the **Document Properties** tab.
3. Click the **Dimensions** option.

The **Document Properties - Dimensions** dialog box can be used to edit the style and form of dimensions. It can also be used to change the way arrows are applied.

4. Click the **Font** option.

The **Choose Font** dialog box will appear. See Figure 7-11. This dialog box can be used to change the font, font style, and height of dimension text. The height of text can be measured in inches, millimeters, or points. A *point* refers to a space that equals about 1/72 of an inch. (There are 12 points to a *pica*.)

Figure 7-11

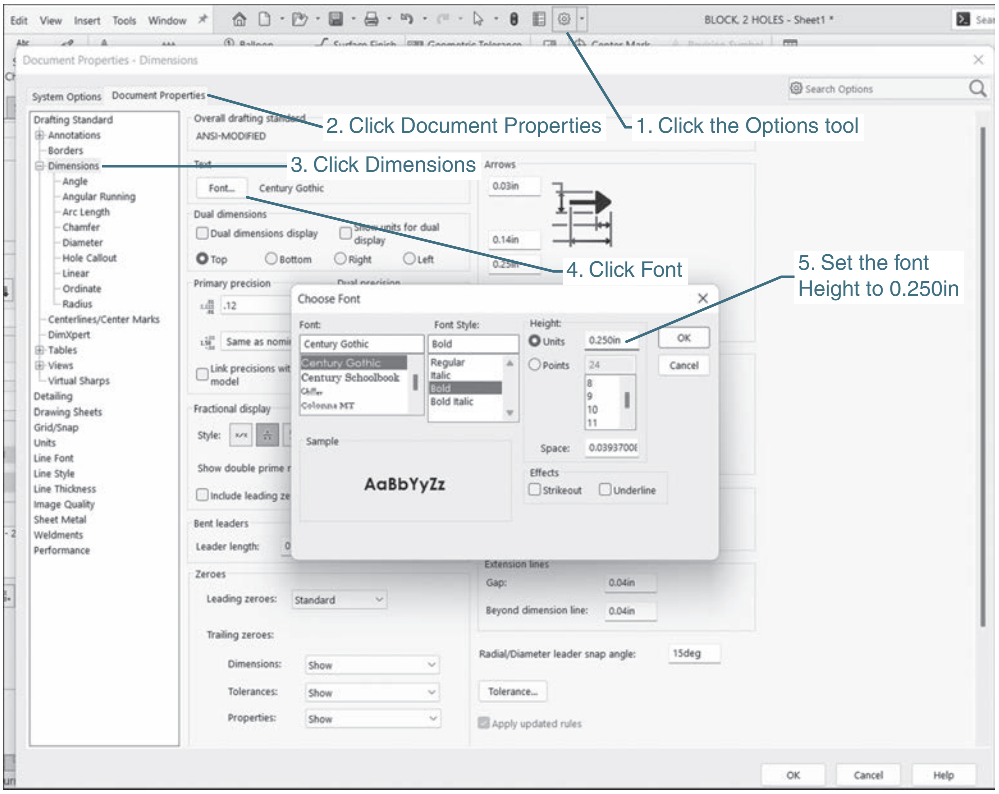
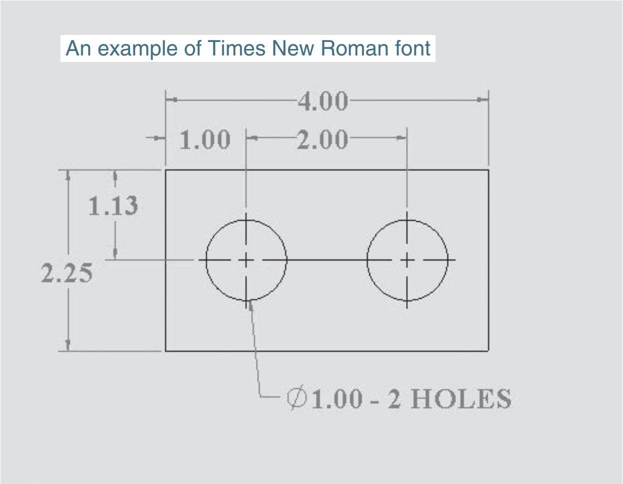


Figure 7-11
(*Continued*)



5 Click the **Height: Units** radio button and change the height to **0.250in**.

Note that the SolidWorks default font is Century Gothic.

6 Click **OK**, then **OK**.

Figure 7-11 shows dimensions created using the Times New Roman font. Fonts for drawings should always be easy to read and not too stylistic.

Dimensioning Short Distances

Figure 7-12 shows an object that includes several short distances. We will start by using the standard dimensions settings and show how to edit them for a particular situation.

1 Use the **Smart Dimension** tool and add dimensions to the drawing.

Note that the arrows for the .50 dimension are aligned with the arrows for the 1.00 dimensions. Dimensions that are aligned in a single row are called *chain dimensions*. Note that the .25 dimension is crowded between the two extension lines.

> **RULE**
>
> Never squeeze dimension values. Dimension values should always be presented clearly and legible.

There are several possible solutions to the crowded .25 value.

2 Click and drag the .25 dimension to the right outside the extension lines.

3 Add the **4.00** overall dimension.

Dimensions that define the total length and width of an object are called *overall dimensions*. In this example the dimension 4.00 defines the total length of the part, so it is an overall dimension. Overall dimensions are located farthest away from the edge of the part.

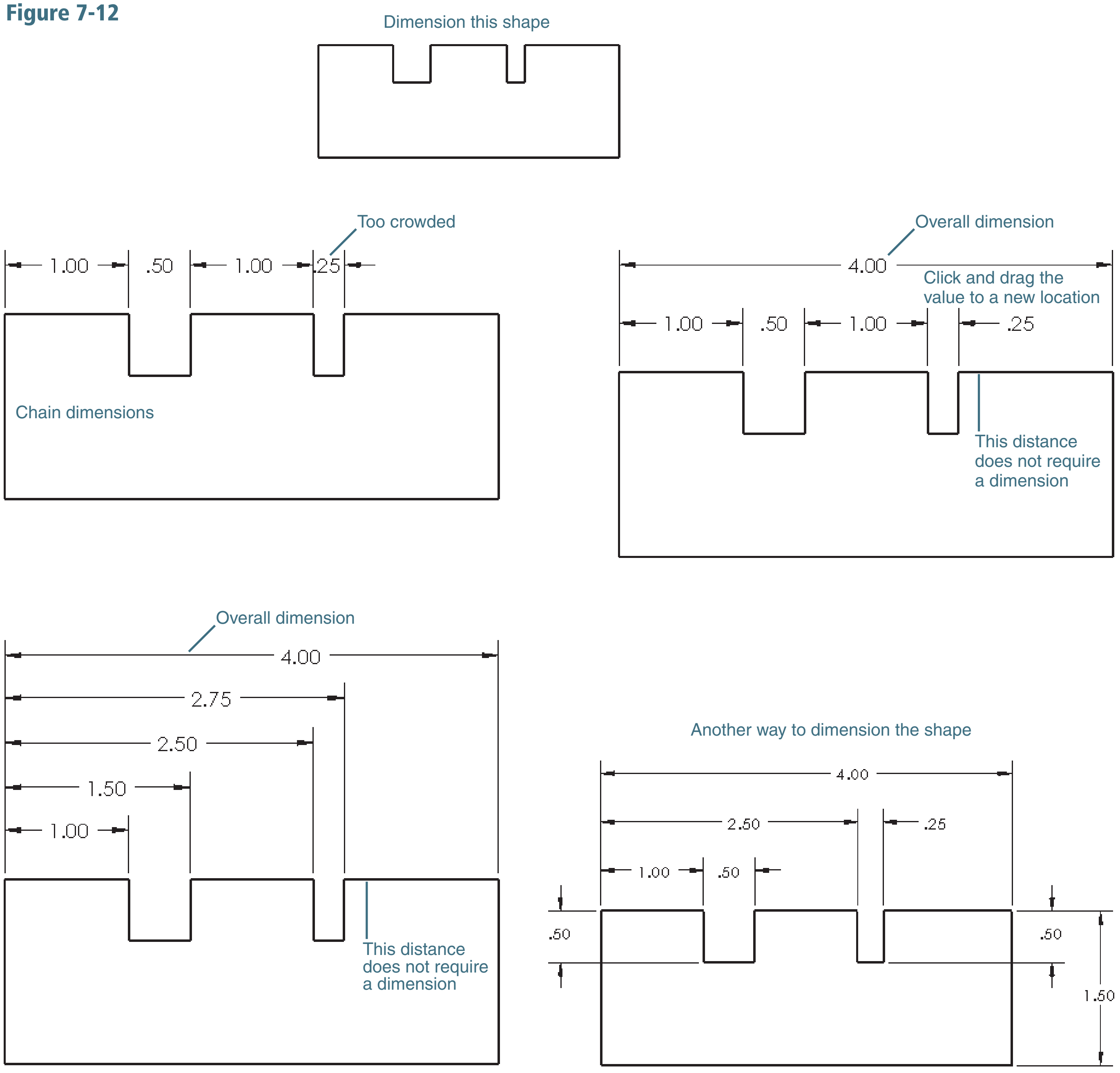
The right edge of the part, the section below the .25, does not need a dimension. The reason for this will be discussed in the next chapter on tolerancing.

TIP

To delete an existing dimension, click the dimension and press the **<Del>** key.

Figure 7-12 shows two other options for dimensioning. The first is the baseline method, in which all dimensions are taken from the same datum line. The second method is a combination of chain and baseline dimensions.

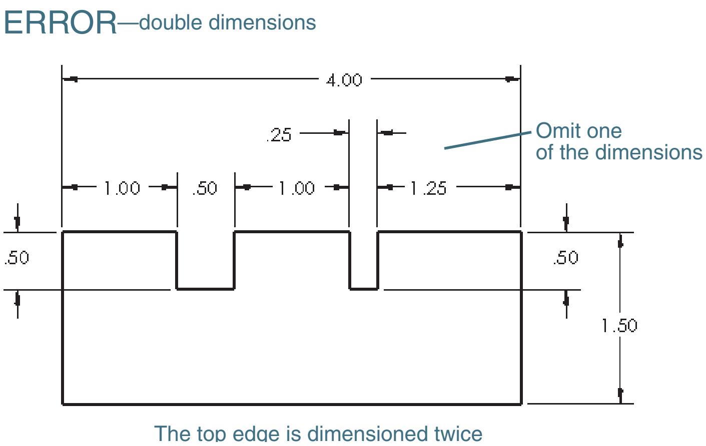
Figure 7-12



> **RULE**
> Never dimension the same distance twice. This is called *double dimensioning*.

Figure 7-13 shows an example of double dimensioning. The top edge distance is dimensioned twice: once using the 1.00 + .50 + 1.00 + .25 + 1.25 dimensions, and a second time using the 4.00 dimension. One of the dimensions must be omitted. Double dimensioning will be explained in more detail in Chapter 8.

Figure 7-13



Autodimension Tool

The **Autodimension** tool will automatically add dimensions to a drawing.

> **WARNING**
> The dimensions created using the **Autodimension** tool are not always in the best locations. The dimensions must be relocated to be in compliance with ANSI conventions.

Figure 7-14 shows a shape to be dimensioned using the **Autodimension** tool.

1. Click the **Annotation** tab, click the **Smart Dimension** tool, and click the **Autodimension** tab.

2. Select the **Chain Scheme**, define **Edge 1** and **Edge 2**, click the **Apply** box, and click the **OK** check mark.

SolidWorks will automatically pick edges 1 and 2. If it does not, or the edges selected are not the ones you want, click the **Edge** box, then click the edge. The word **Edge<1>** should appear in the box.

Figure 7-14 shows the dimensions applied using the **Autodimension** tool. They are not in acceptable positions.

3. Rearrange the dimensions to comply with standard conventions.

Figure 7-14

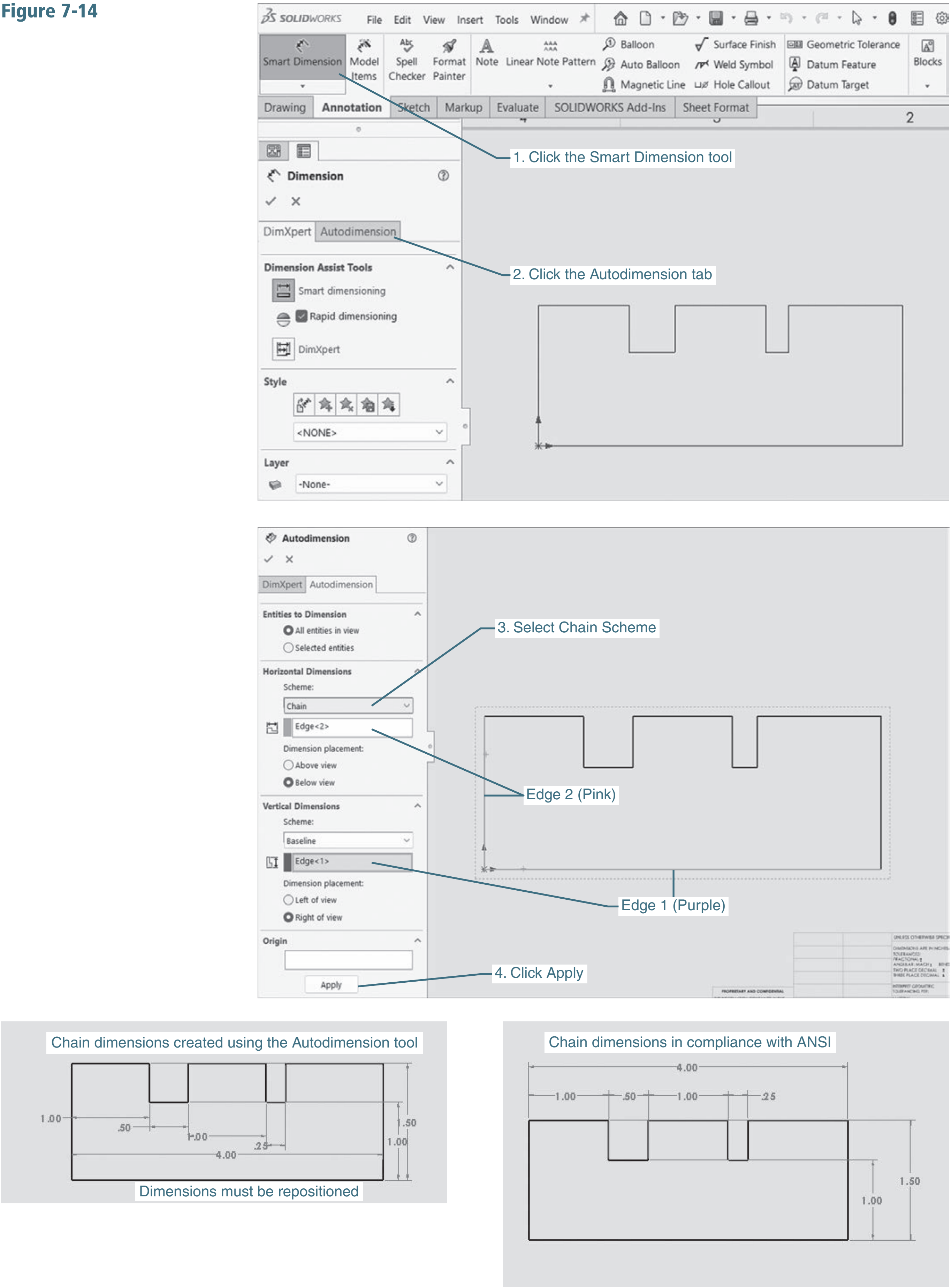
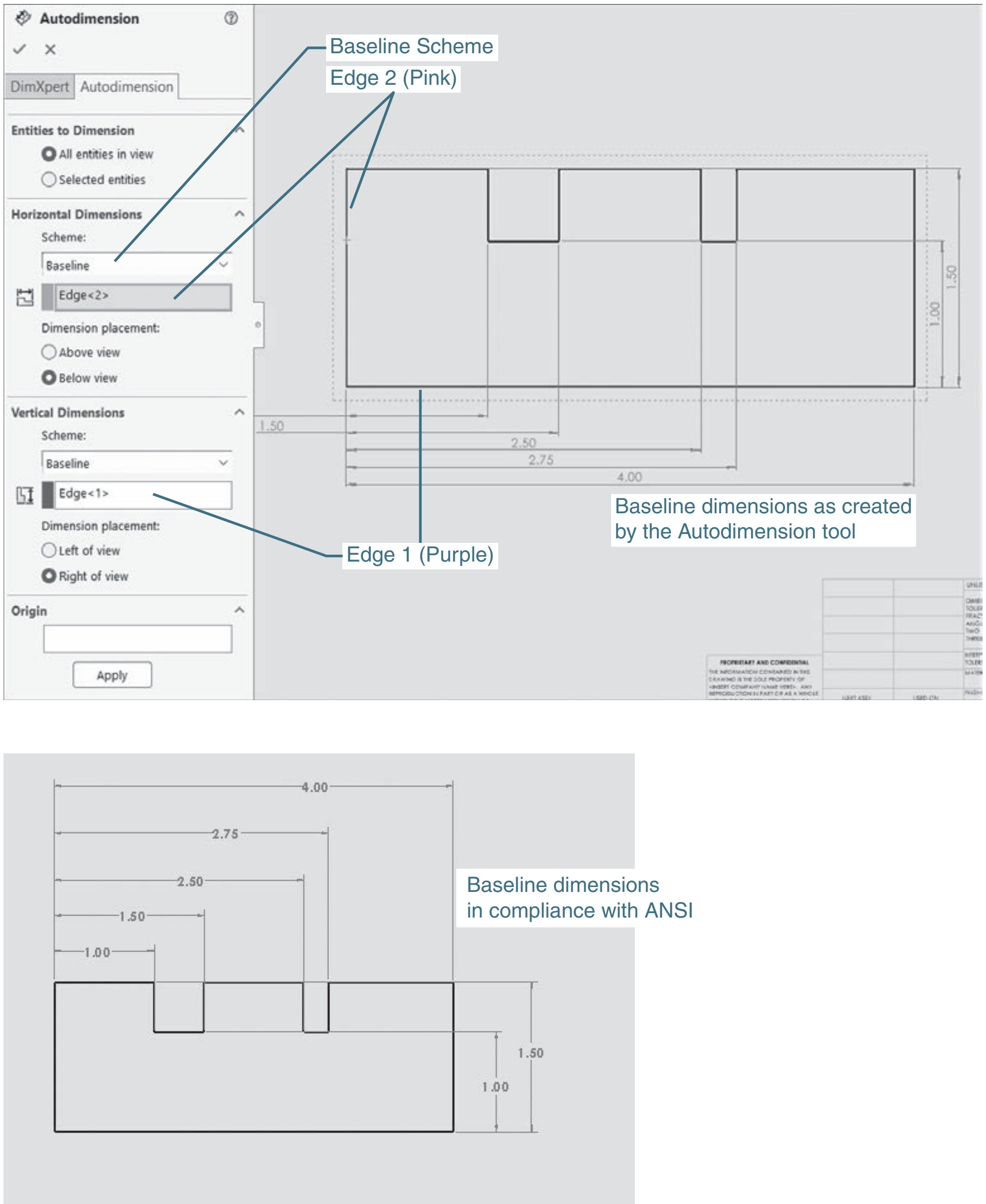

1. Click the Smart Dimension tool
2. Click the Autodimension tab
3. Select Chain Scheme
Edge 2 (Pink)
Edge 1 (Purple)
4. Click Apply
Chain dimensions created using the Autodimension tool
Dimensions must be repositioned
Chain dimensions in compliance with ANSI
4.00
1.00
.50
.25
1.50

Figure 7-15 shows the shape shown in Figure 7-14 dimensioned using the baseline scheme, which is created as follows.

Figure 7-15



Creating Baseline Dimensions

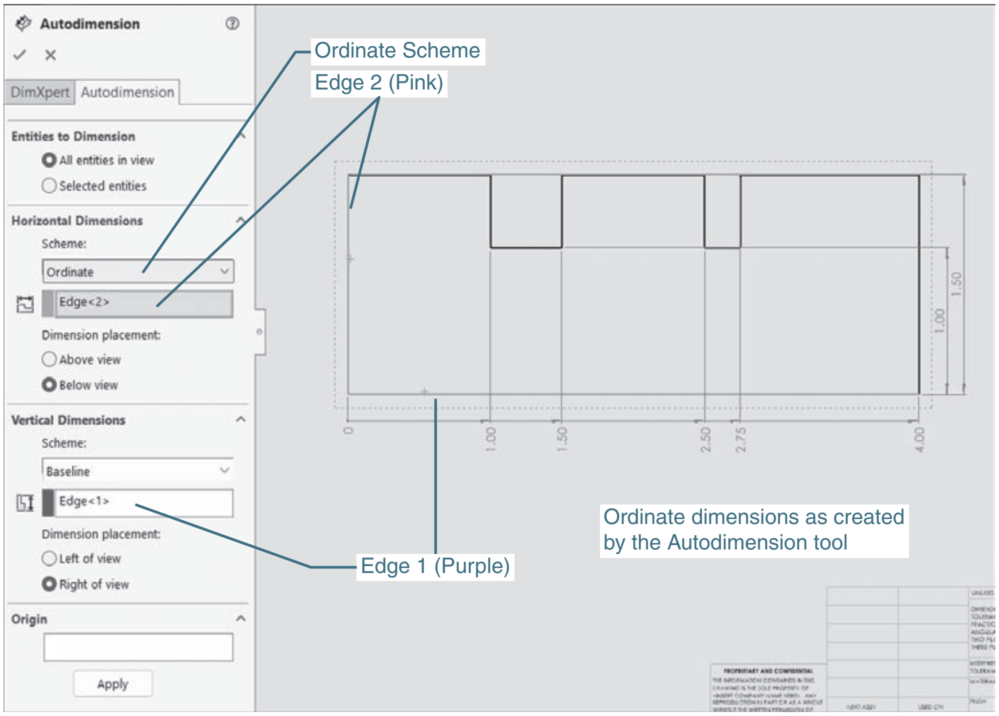
1. Access the **Autodimension** tool and select the **Baseline Scheme**.
2. Select **Edge 1** and **Edge 2**.
3. Click **Apply**.
4. Click the green **OK** check mark.

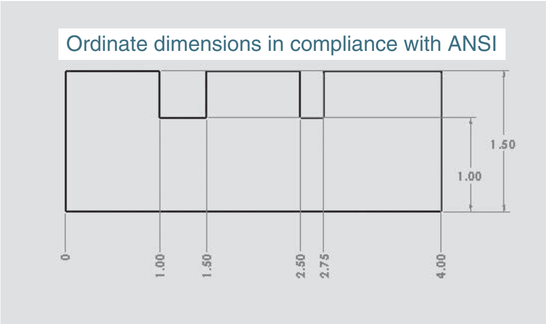
Figure 7-15 shows the dimensions created by the **Autodimension** tool and how the dimensions can be rearranged.

Creating Ordinate Dimensions

Figure 7-16 shows the object dimensioned using the **Ordinate Scheme** of the **Autodimension** tool. The **Autodimension** tool did better placing the ordinate dimensions in this instance, but if some of the created dimensions are located on the surface of the part, this would be a violation of the convention that dimensions should never be located on the surface of the part. Figure 7-16 shows how the ordinate dimensions were rearranged.

Figure 7-16





7-4 Drawing Scale

Drawings are often drawn "to scale" because the actual part is either too big to fit on a sheet of drawing paper or too small to be seen. For example, a microchip circuit must be drawn at several thousand times its actual size to be seen.

Drawing scales are written using the following formats:

SCALE: 1 = 1

SCALE: FULL

SCALE: 1000 = 1

SCALE: .25 = 1

In each example the value on the left indicates the scale factor. A value greater than 1 indicates that the drawing is larger than actual size. A value smaller than 1 indicates that the drawing is smaller than actual size.

Regardless of the drawing scale selected, the dimension values must be true size. Figure 7-17 shows the same rectangle drawn at two different scales. The top rectangle is drawn at a scale of 1 = 1, or its true size. The bottom rectangle is drawn at a scale of 2 = 1, or twice its true size. In both examples the 3.00 dimension remains the same.

Figure 7-17

SCALE: FULL

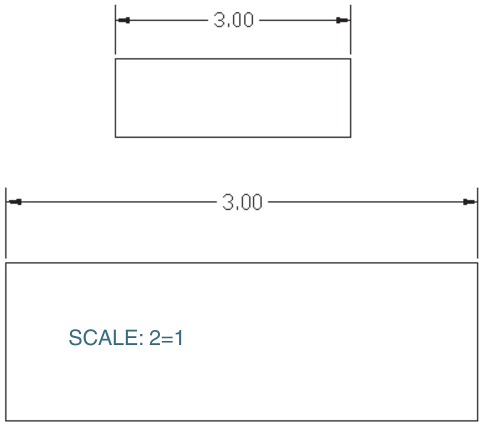


Figure 7-18

TOLERANCES UNLESS OTHERWISE STATED

X	± 1
.X	± .1
.xx	± .01
.XXX	± .005
X°	± 1°
.X°	± .1°

7-5 Units

It is important to understand that dimension values are not the same as mathematical units. Dimension values are manufacturing instructions and always include a tolerance, even if the tolerance value is not stated. Manufacturers use a predefined set of standard dimensions that are applied to any dimensional value that does not include a written tolerance. Standard tolerance values differ from organization to organization. Figure 7-18 shows a chart of standard tolerances.

In Figure 7-19 a distance is dimensioned twice: once as 5.50 and a second time as 5.5000. Mathematically these two values are equal, but they are not the same manufacturing instruction. The 5.50 value could, for example, have a standard tolerance of ±.01, whereas the 5.5000 value could have a standard tolerance of ±.0005. A tolerance of ±.0005 is more difficult and therefore more expensive to manufacture than a tolerance of ±.01.

Figure 7-20 shows examples of units expressed in millimeters and in decimal inches. A zero is not required to the left of the decimal point for decimal inch values less than one. Millimeter values do not require zeros to the right of the decimal point. Millimeter and decimal inch values never include symbols; the units will be defined in the title block of the drawing.

Figure 7-19

These dimensions are not the same. They have different tolerance requirements.

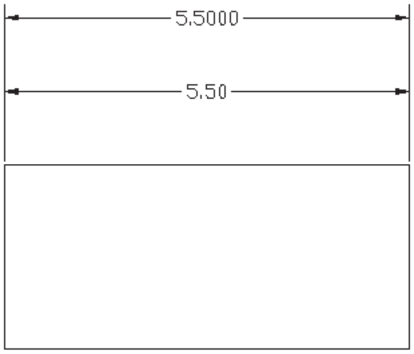


Figure 7-20

Millimeters

0.25	0.5	0.033
32	14.5	3

Zero required

Inches

No zero required

.25	.05	.033
32.00	14.50	3.000

Aligned Dimensions

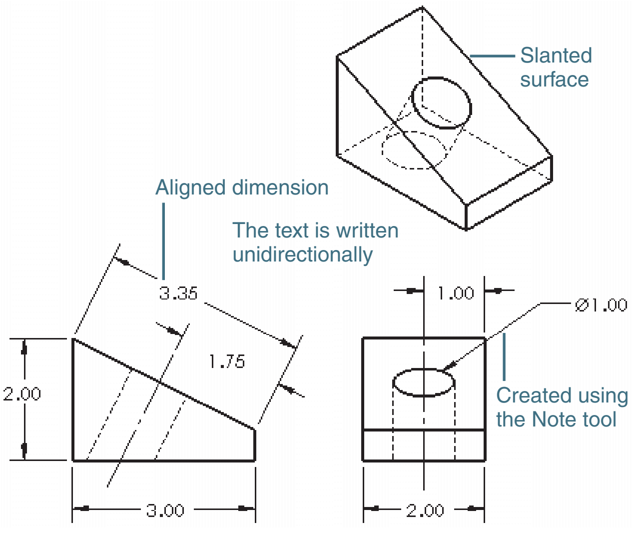
Aligned dimensions are dimensions that are parallel to a slanted edge or surface. They are not horizontal or vertical. The units for aligned dimensions should be written horizontally. This is called *unidirectional dimensioning*.

Figure 7-21 shows the front, right-side, and isometric views of a part with a slanted surface. The dimensions were applied using the **Smart Dimension** tool. Note that the slanted dimension, aligned with the slanted surface, has unidirectional (horizontal) text. The hole dimension was created using the **Note** tool from the **Annotation** tab.

Hole Dimensions

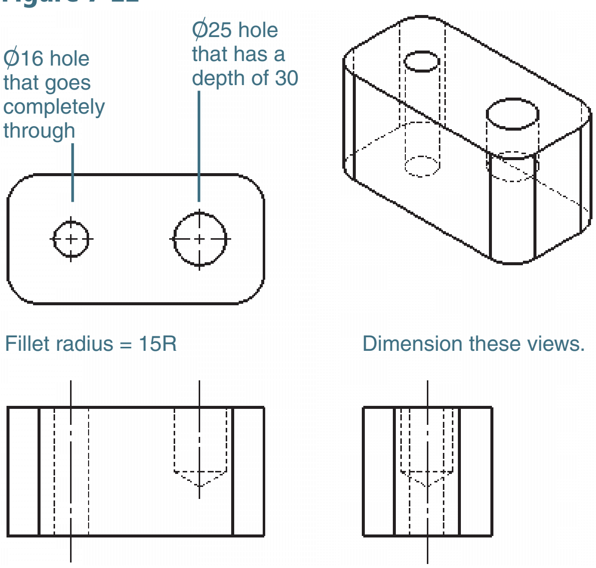
Figure 7-22 shows an object that has two holes, one blind and one completely through. The object has filleted corners. In this section we will add dimensions to the views.

Figure 7-21



Dimensions were created using the Smart Dimension tool

Figure 7-22



The holes were drawn using the **Hole Wizard** tool. The **Hole Wizard** tool will automatically create a conical point to a blind hole.

1 Use the **Smart Dimension** tool and locate the two holes.

See Figure 7-23. In general, dimensions are applied from the inside out; that is, starting with the features in the middle of the part and working out to the overall dimensions. Leader lines are generally applied last, as they have more freedom of location.

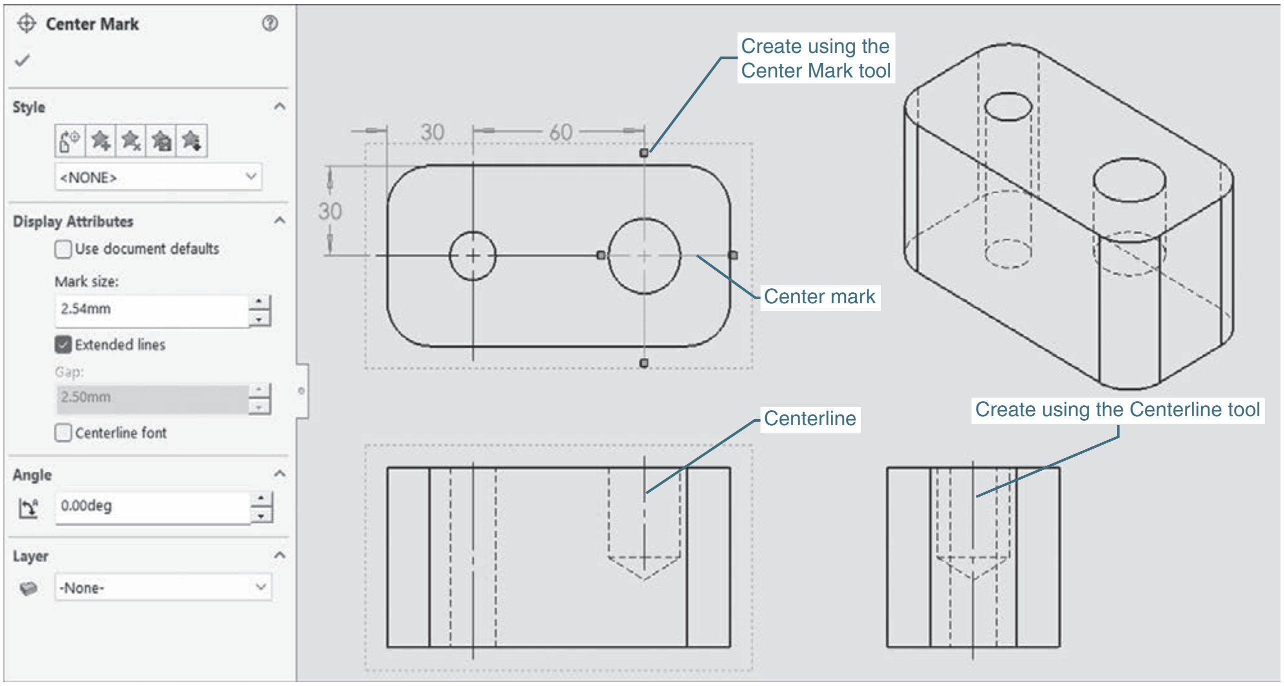
2 Use the **Linear Center Mark** tool to draw a centerline between the two holes and use the **Centerline** tool to add the vertical centerline in the front and side views.

The centerline between the two holes indicates that the vertical 30 dimension applies to both holes.

NOTE

Centerlines should extend beyond the edges of the part. Centerlines can be extended by first clicking the centerline. Blue end boxes will appear. Click and drag the blue end boxes to a point beyond the edges of the part.

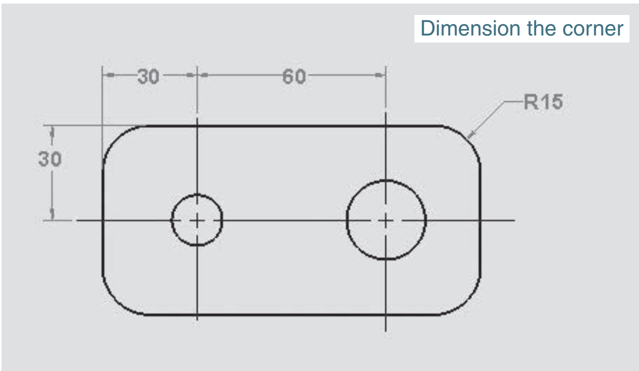
Figure 7-23



3 Use the **Smart Dimension** tool and add a dimension to one of the filleted corners.

See Figure 7-24.

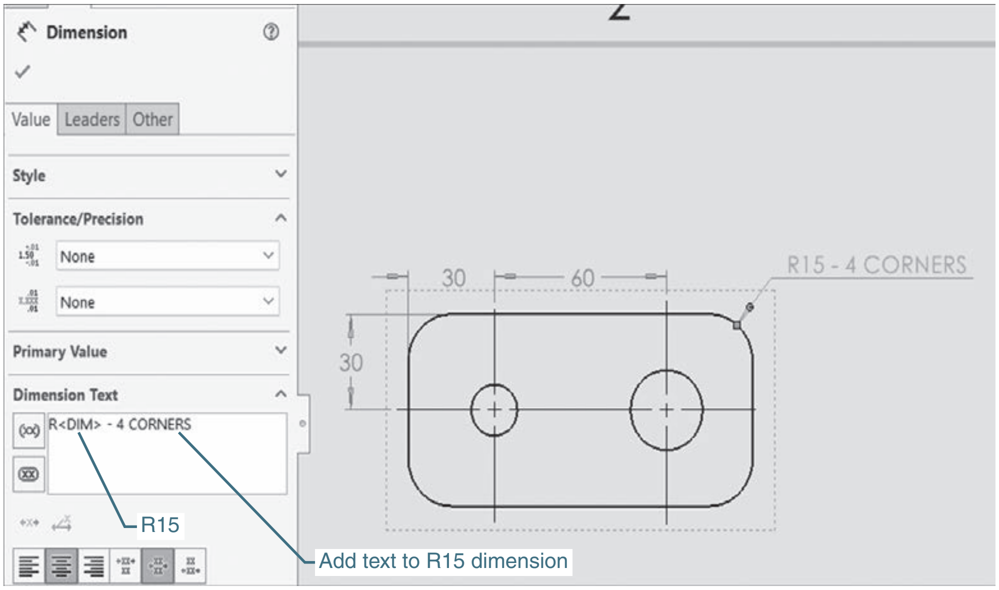
Figure 7-24



> **TIP**
> The dimension options found on the **Document PropertyManager** will change all dimensions. Clicking a dimension and using the **Dimension PropertyManager** allows you to change just that dimension.

4 Click the fillet dimension again, go to the **Dimension Text** block on the **Dimension PropertyManager**, and type **- 4 CORNERS** as shown in Figure 7-25.

Figure 7-25



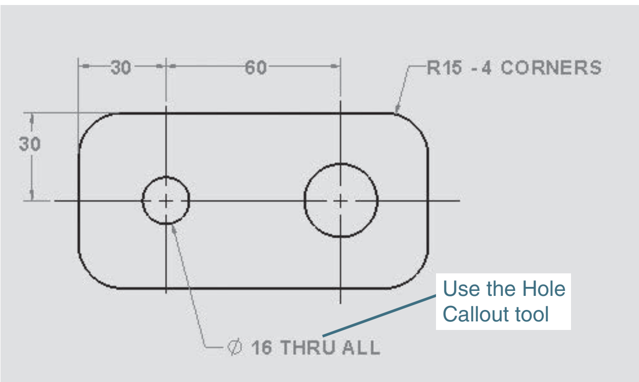
5 Click **OK**, **Apply**, and **OK**.

6 Use the **Hole Callout** tool on the **Annotation** panel and dimension the Ø16 hole.

The Ø16 hole goes completely through the part, so no depth specification is required. See Figure 7-26. The word THRU is optional and may be removed.

7 Use the **Hole Callout** tool and **Dimension** the Ø25 hole.

Figure 7-26



The hole callout will include the depth symbol (see Figure 7-27) and a depth value of **30** (see Figure 7-28).

Complete the dimensions.

See Figure 7-29.

Figure 7-27

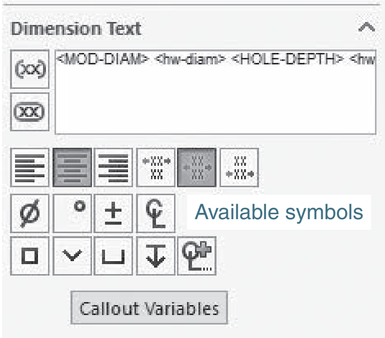


Figure 7-28

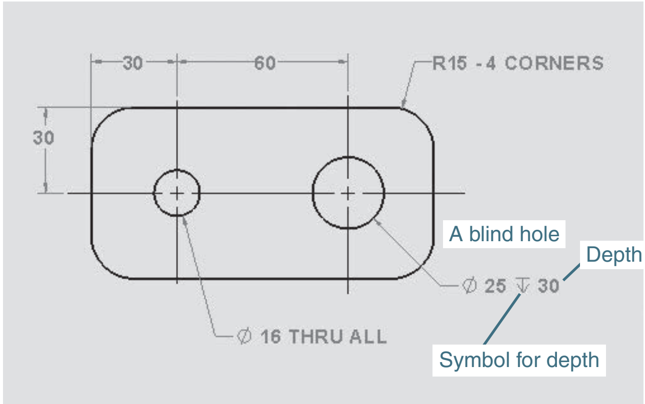
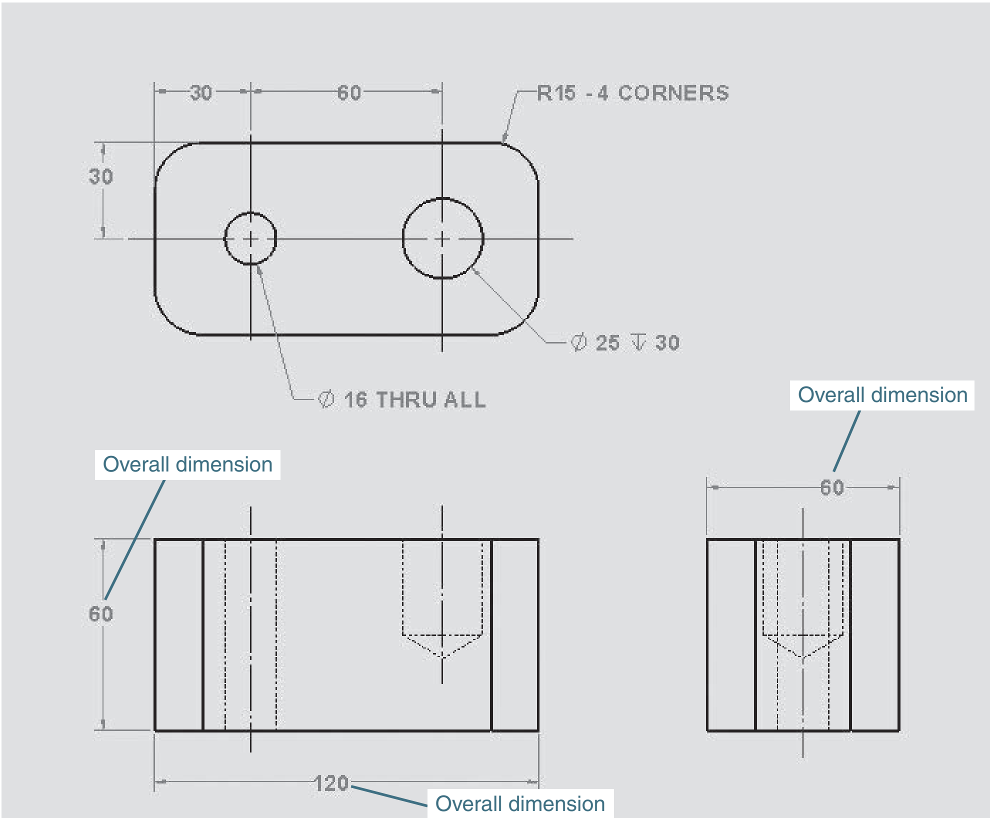


Figure 7-29



NOTE

If the **Smart Dimension** tool had been used, the dimension would have to be edited in the **Dimension Text** area and the depth symbol and a numerical value added.

7-6 Dimensioning Holes and Fillets

A *blind hole* is a hole that does not go completely through an object. It has a depth requirement. Figure 7-30 shows a 2.00 × 2.00 × 2.00 cube with a blind Ø.50 × 1.18 DEEP hole. It was created as follows.

Dimensioning a Blind Hole

1 Draw the block.

2 Click the **Hole Wizard** tool.

See Figure 7-30.

Figure 7-30

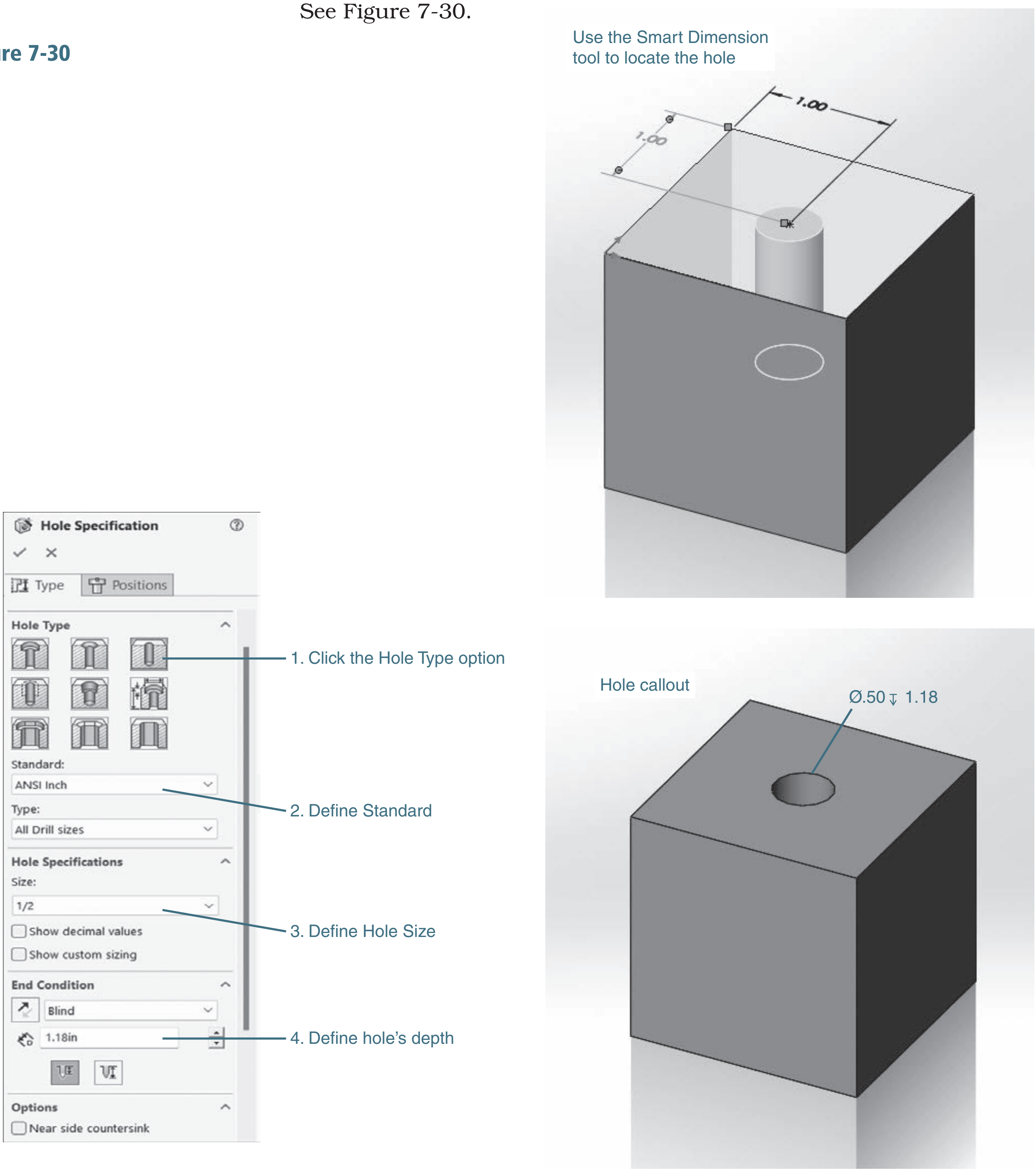
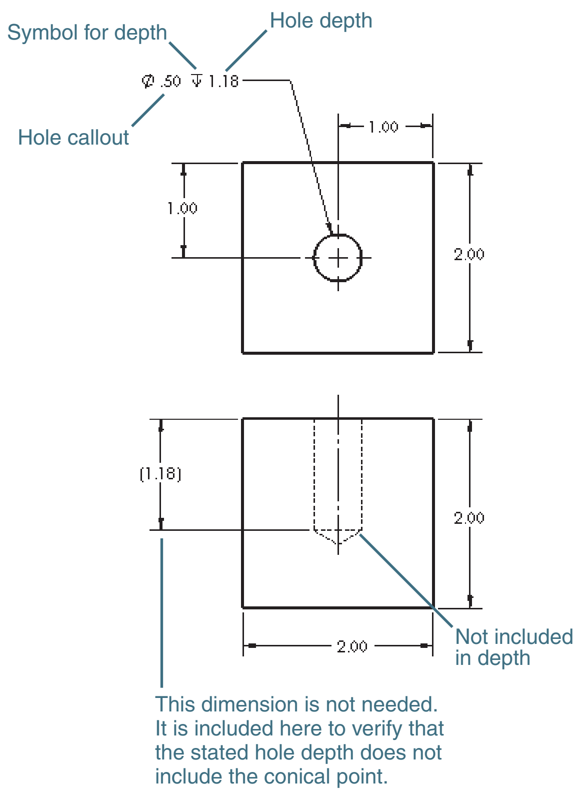


Figure 7-30
(*Continued*)



3 Click the **Hole** tool in the **Hole Type** box. Define the hole using the **ANSI Inch** standard with a **Size** of **1/2** and a **Blind Hole Depth** of **1.18in**.

4 Click the **Positions** tab.

5 Locate the hole as shown.

The initial location is an approximation. Use the **Smart Dimension** tool to specify the exact location of the hole's centerpoint.

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

7 Save the drawing as **Block, Blind**.

8 Start a new **Drawing** document and create a front and a top orthographic view of the **Block, Blind**.

9 Add dimensions to the views.

10 Click the **Annotation** tab and click the **Hole Callout** option.

11 Click the edge of the hole, move the cursor away from the hole, define a location for the hole callout, and click the mouse. The hole callout dimension will initially appear as a rectangular box.

Change the height of the text font if necessary.

12 Save the drawing.

Note that the hole includes a conical point. Holes manufactured using twist drills will have conical points. The conical point is not included in the hole's depth dimension. A special drill bit can be used to create a flat-bottomed hole.

Figure 7-31 shows three different methods that can be used to dimension a blind hole.

Figure 7-31

Dimensions for holes with depth

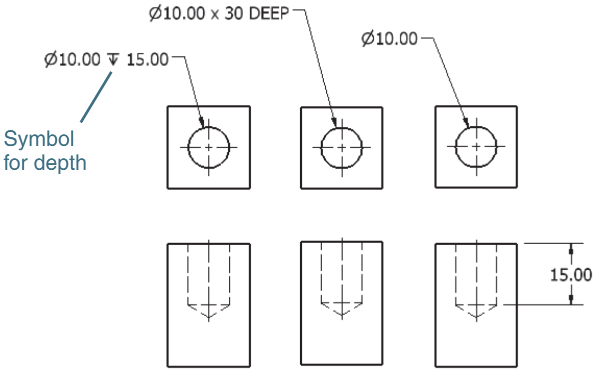


Figure 7-32

Section views of holes with depth

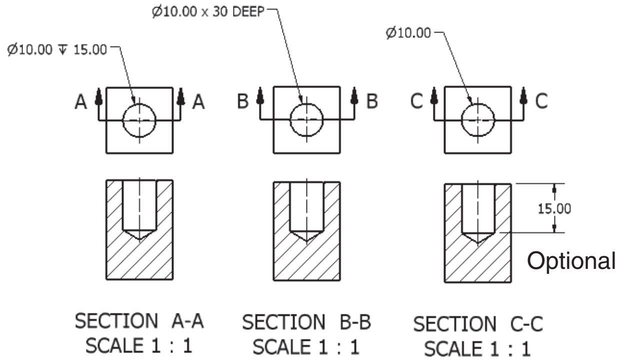


Figure 7-32 shows three methods of dimensioning holes in section views. The single line note version is the preferred method.

Dimensioning Hole Patterns

Figure 7-33 shows two different hole patterns dimensioned. The circular pattern includes the note **Ø10 – 4 HOLES**. This note serves to define all four holes within the object.

Figure 7-33 also shows a rectangular object that contains five holes of equal diameter, equally spaced from one another. The notation **5** × **Ø10** specifies five holes of 10 diameter. The notation **4** × **20 (=80)** means four equal spaces of 20. The notation **(=80)** is a reference dimension and is included for convenience. Reference dimensions are explained in Chapter 9.

Figure 7-33

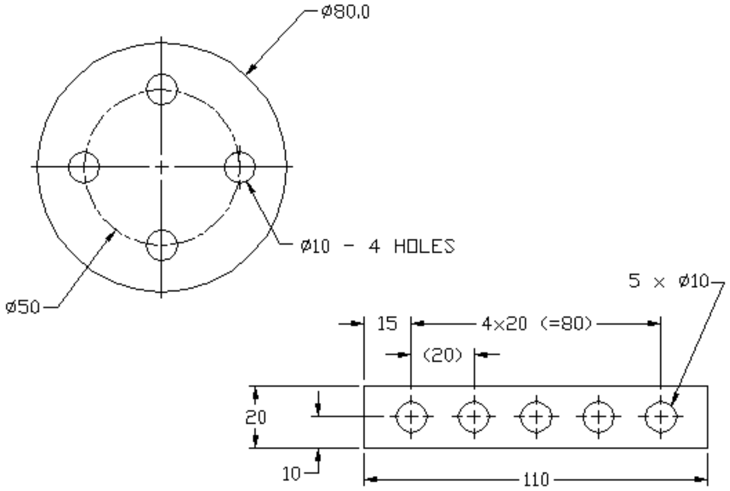


Figure 7-34 shows two additional methods for dimensioning repeating hole patterns. Figure 7-35 shows a circular hole pattern that includes two different hole diameters. The hole diameters are not noticeably different and could be confused. One group is defined by indicating letter **A**; the other is dimensioned in a normal manner.

Figure 7-34

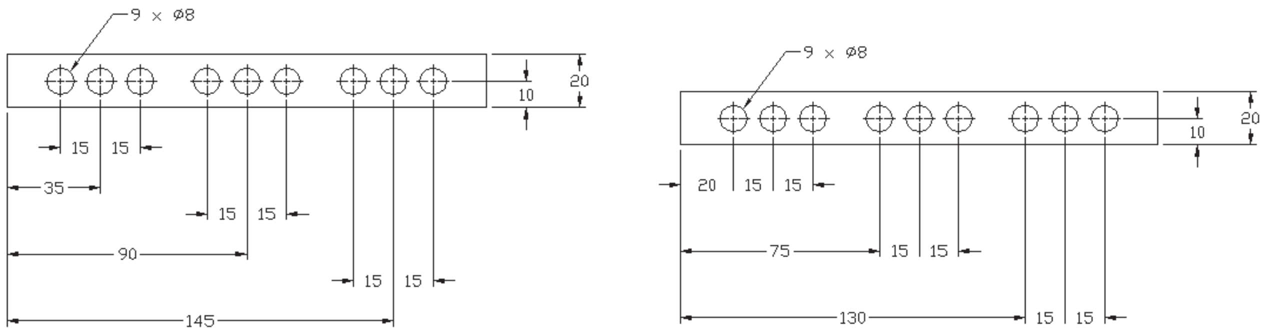
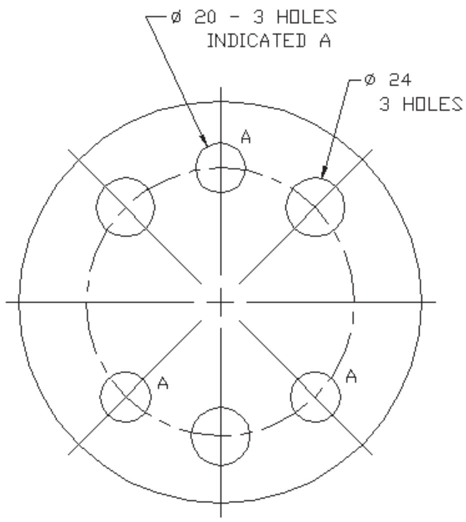


Figure 7-35



7-7 Dimensioning Counterbored and Countersunk Holes

Counterbored holes are dimensioned in the sequence of their manufacture. First the hole's diameter is given, then the counterbore diameter, then the depth of the counterbore.

Figure 7-36 shows a part that contains two counterbored holes; one goes completely through and the other is blind. Dimensions will be applied to both.

1 Create a **3.00** × **4.00** × **1.75** block.

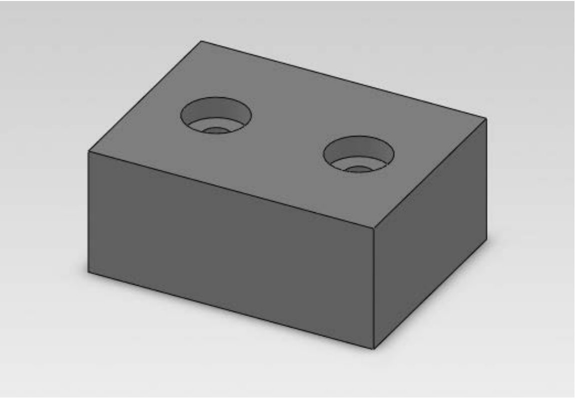
2 Click the **Hole Wizard** tool, click the **Counterbore** option, and insert the counterbored hole that goes completely through.

3 Specify the **Standard** as **ANSI Inch**, the **Type** as **Hex Screw**, the **Size** as a **3/8** diameter, and the **End Condition** as **Through All**.

See Figure 7-37. SolidWorks will automatically select the diameter for the counterbored hole that will accommodate a Ø3/8 Hex Head Screw.

Depending on your default settings, the counterbored hole may have a small chamfer added. The countersink can be removed by removing the check mark in the **Options** box on the **Hole Specification** manager.

Figure 7-36



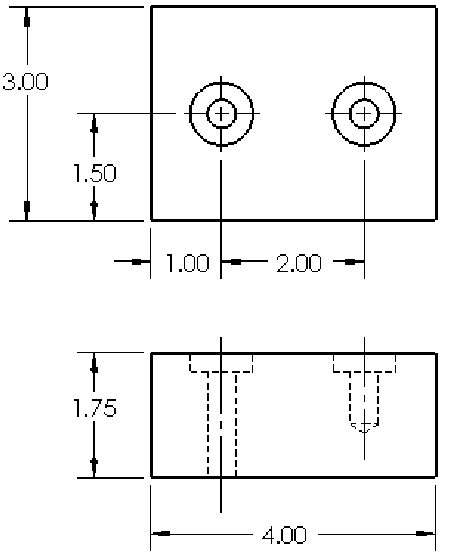


Figure 7-37

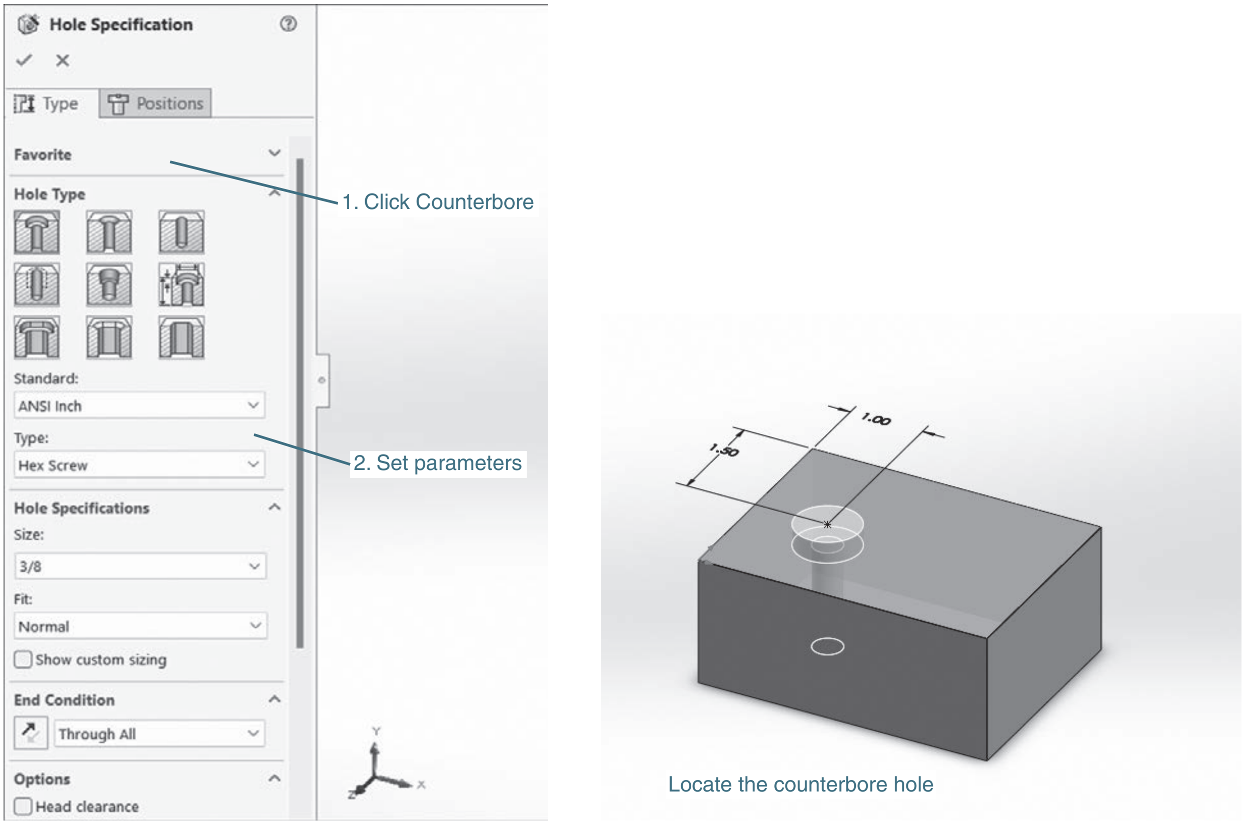
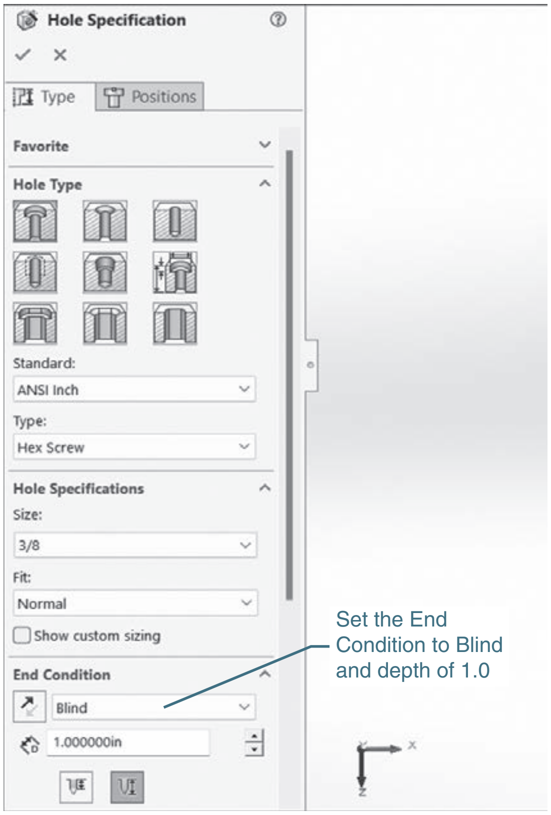
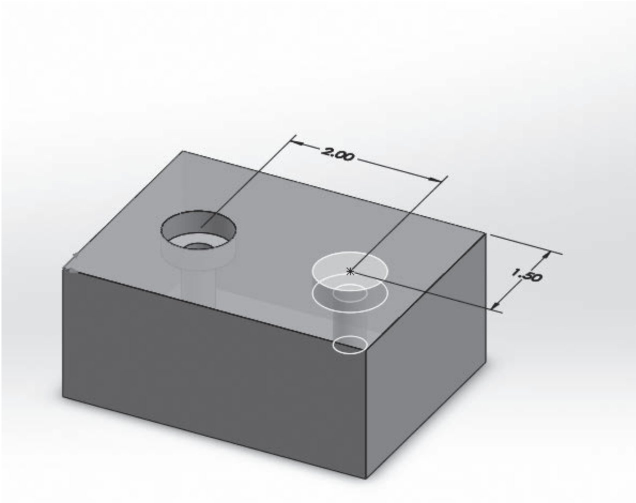


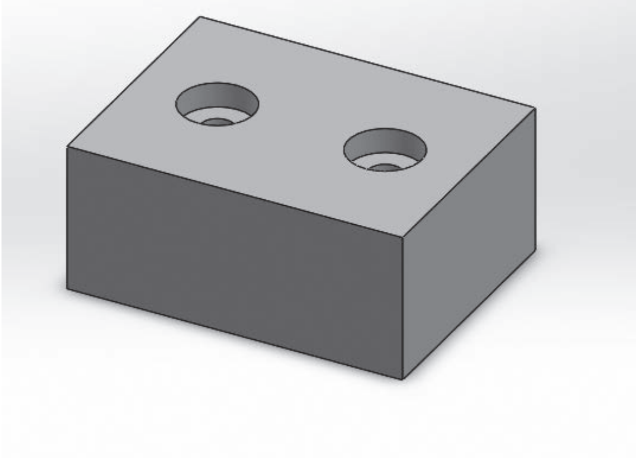
Figure 7-37
(*Continued*)





Position the second hole

Finished block



4 Position the hole using the given dimensions.

5 Add the second hole, setting the **End Condition** to **Blind** and **1.00** deep.

6 Position the hole using the given dimensions.

7 Save the block as **Block, Cbore**.

8 Start a new **Drawing** document and create a front and a top orthographic view of the **Block, Cbore**.

9 Click the **Annotation** tab and add all dimensions and centerlines other than the hole dimensions.

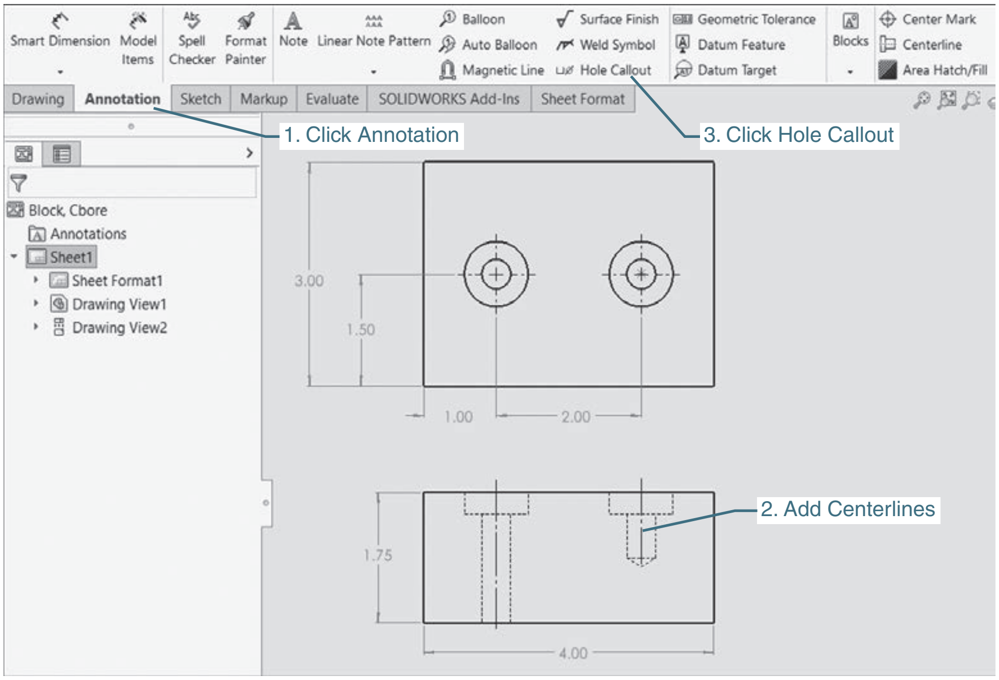
10 Click the **Hole Callout** tool.

See Figure 7-38.

11 Click the edge of each hole, move the cursor away from the hole, and click the mouse when a suitable location is found.

The counterbored hole's dimension note is interpreted as shown in Figure 7-38.

Figure 7-38



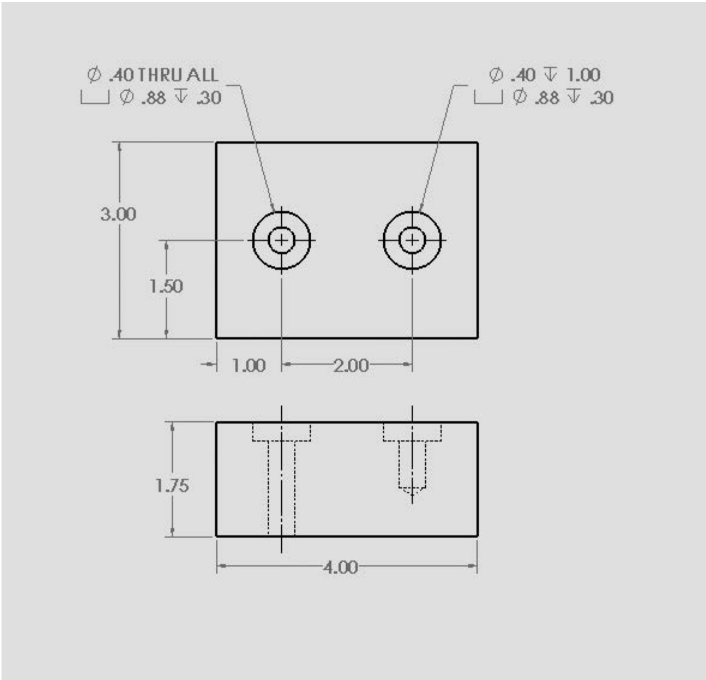


Figure 7-39 shows the **Block, Cbore** assembled with hex head screws inserted into the counterbored holes. SolidWorks will automatically generate the correct size counterbored hole for a specified screw. The counterbore depth will align the top of the screw head with the top surface of the part and will define a hole diameter that includes clearance between the fastener and the hole. In this example a clearance hole with a diameter of **Ø.40** was generated. The hole is .02 larger than the specified .38 fastener diameter.

If clearance is required between the top of the screw and the top surface of the part, check the **Head clearance** box under **Options** in the **Hole Specification** section of the **Hole Wizard PropertyManager**. See Figure 7-40.

Figure 7-39

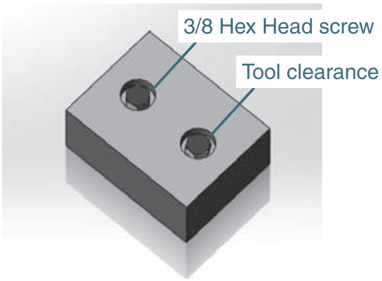
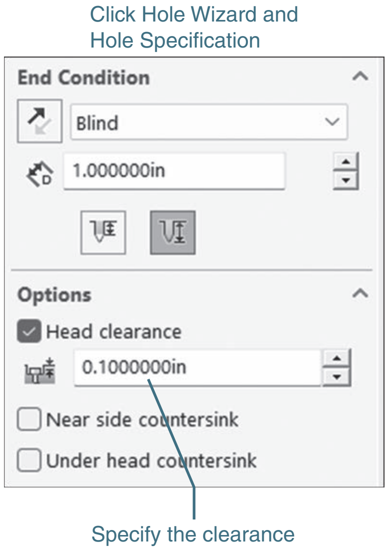


Figure 7-40



The diameter of the counterbored hole can be made larger than the clearance generated by SolidWorks to allow for tool clearance. Tool clearance allowance increases the diameter of the counterbore so that it is large enough to allow a socket wrench to fit over the head of the fastener and still fit within the hole.

Counterbored Hole with Threads

Figure 7-41 shows a 3.00 × 4.00 × 2.00 block with two counterbored holes. Both holes are threaded.

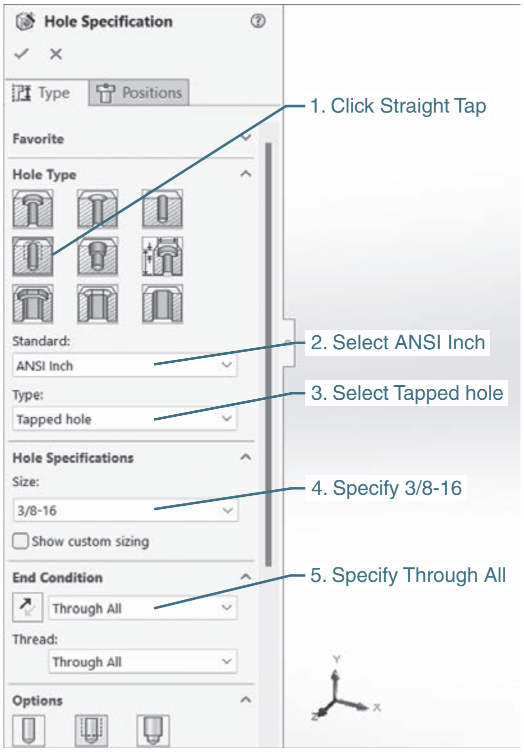
1. Create the block.
2. Click the **Hole Wizard** tool, select the **Straight Tap** option, and specify a **3/8-16 UNC** thread that goes completely through.
3. Click the **Positions** tab and locate the hole.
4. Click the green **OK** check mark.

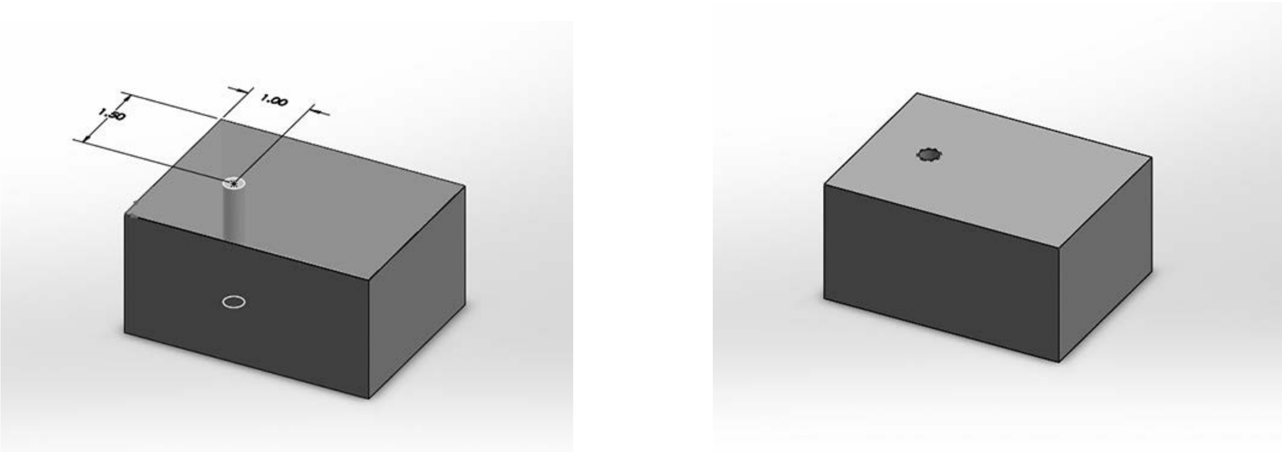
This will locate a 3/8-16 UNC thread hole in the block. Now, we add the counterbore.

5. Click the top surface of the block and click the **Sketch** option.
6. Click the **Circle** tool and draw a **Ø.88** circle on the top surface centered on the same centerpoint as the Ø3/8-16 hole.

The dimensions for this example came from Figure 7-38.

Figure 7-41
(*Continued*)





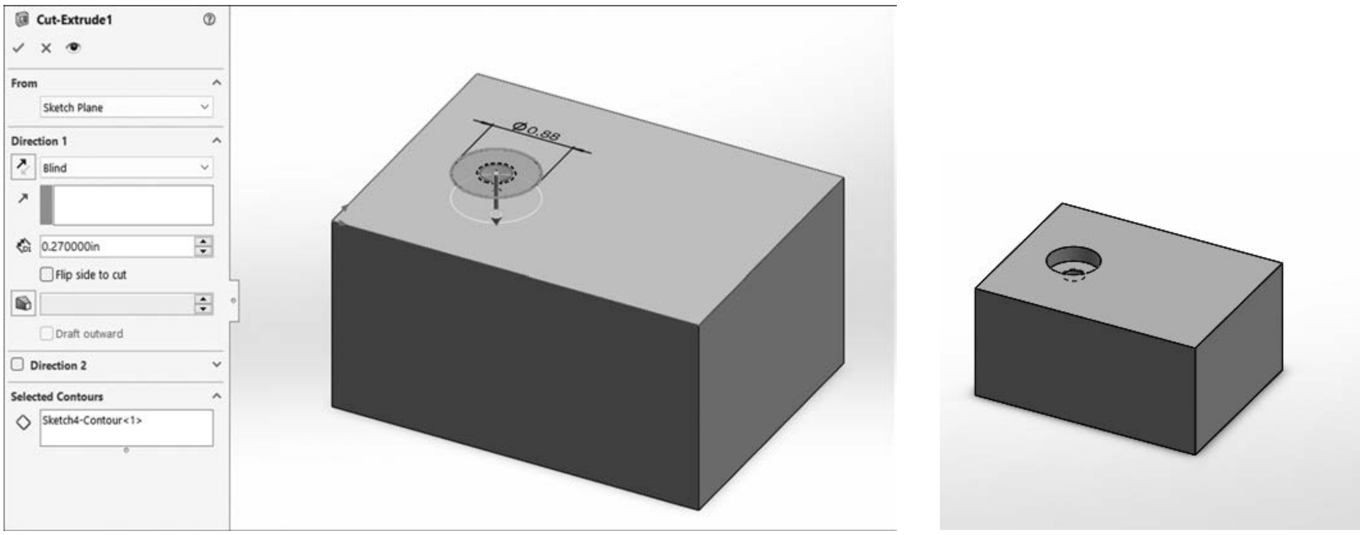
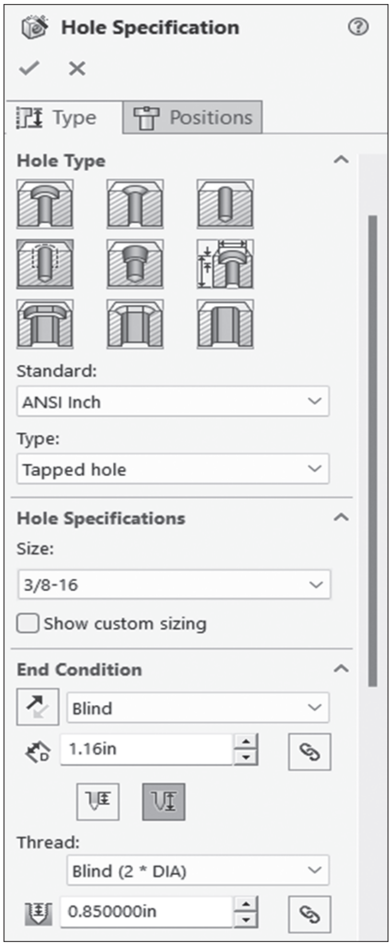
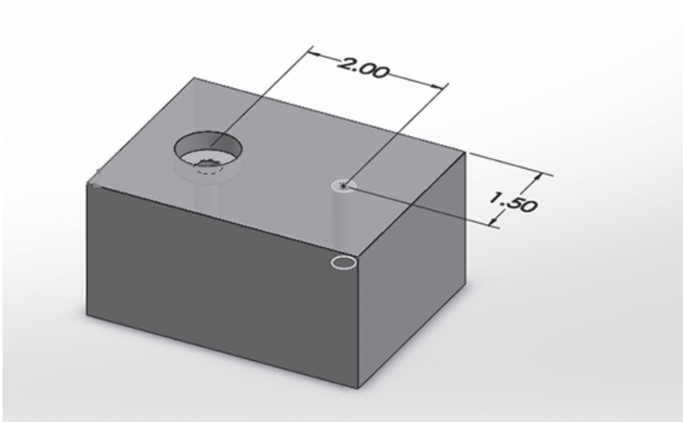
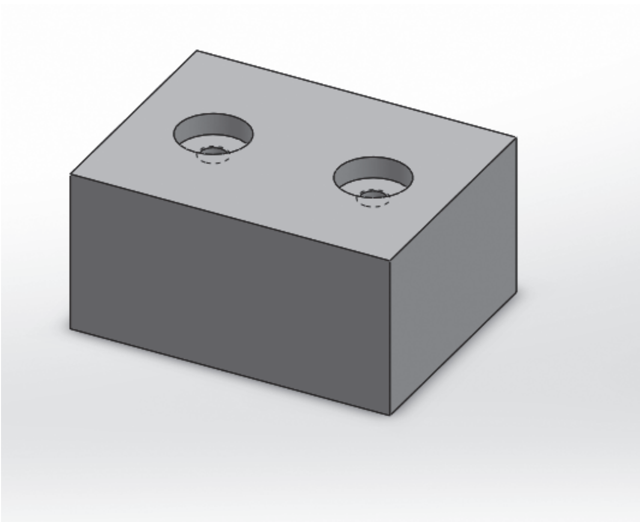


Figure 7-41
(*Continued*)







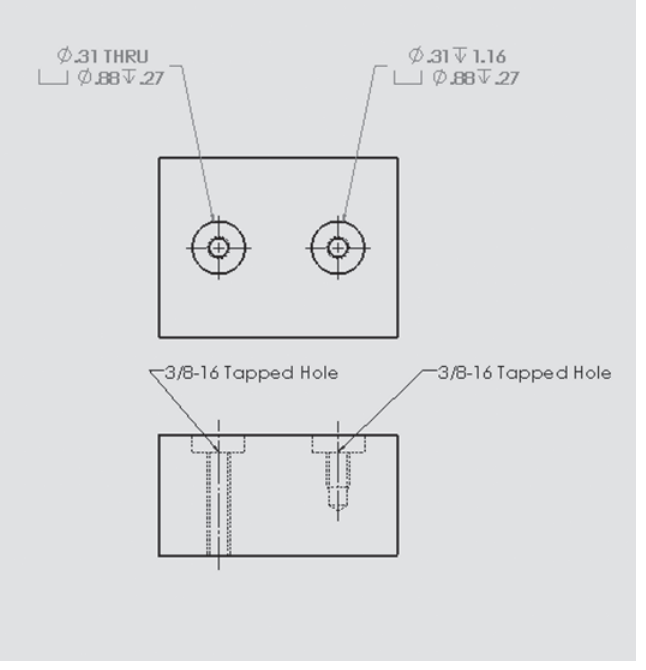
7. Click the **Features** tab, click the **Extruded Cut** tool, and specify a cut depth of **0.27**.
8. Click the green **OK** check mark.
9. Repeat the procedure, adding a second hole with a thread to a depth of **0.85**.

TIP

For an internal thread, the thread depth is measured from the top surface of the part.

See Figure 7-42.

Figure 7-42



10 Save the block as **Block, Threads**.

11 Create a new **Drawing** document and create front and top orthographic views of the **Block, Threads**.

12 Add centerlines to the front view and add dimensions as shown.

See Figure 7-43.

13 Use the **Hole Callout** tool and click the left threaded hole.

Do not click the outside of the counterbored hole. This will generate a note that includes only the counterbore. In Figure 7-42 the callouts 3/8-16 Tapped Hole appear on the front view. Remove the callouts from view by right-clicking the callouts and selecting the **Hide** option. The thread information will be included in the counterbore hole callout.

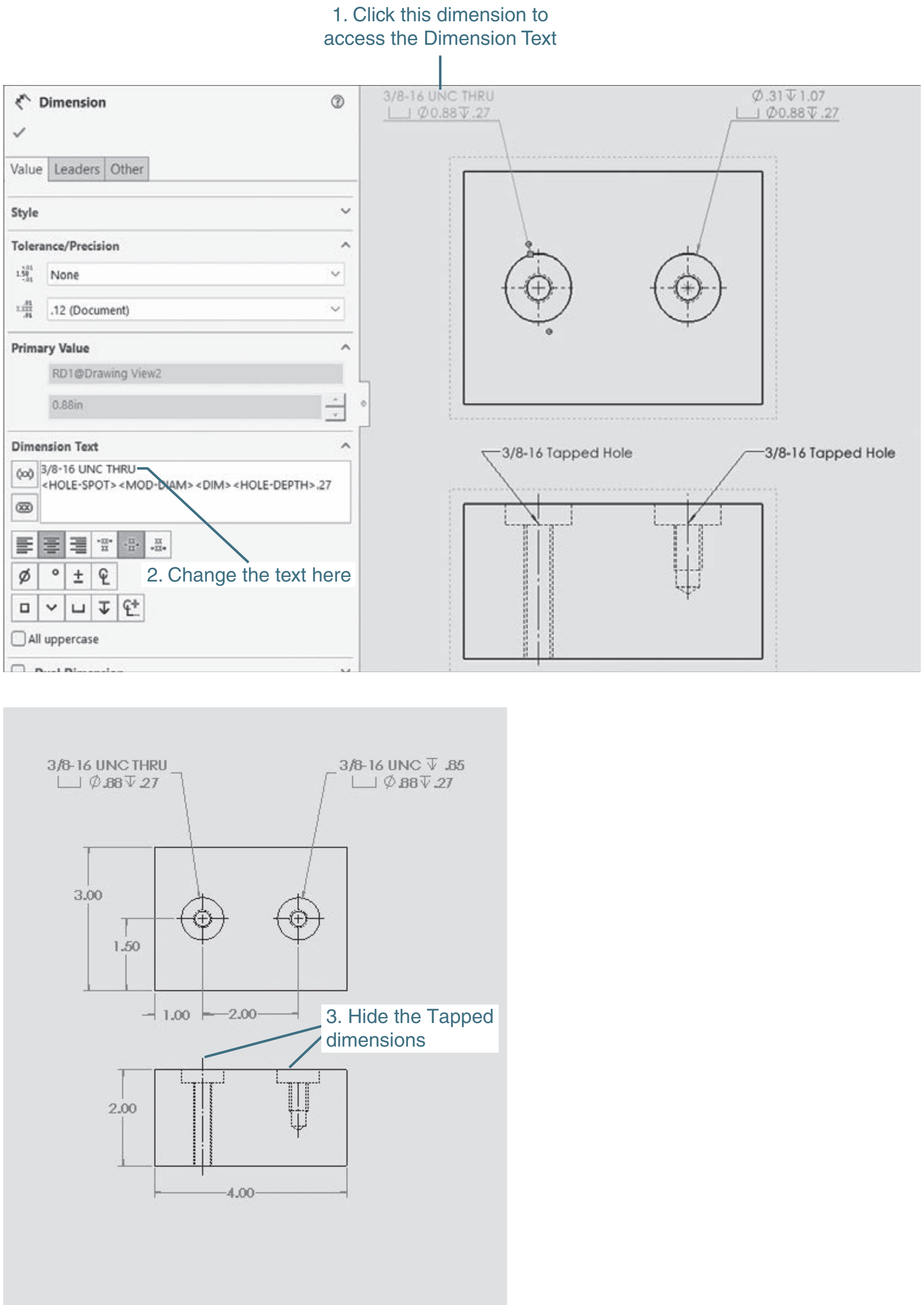
14 Locate the text and click the mouse.

The initial note may show the counterbore callout above the thread callout. Convention calls for the note to read in the sequence of manufacture. The threaded hole is cut first and then the counterbore is added; therefore, the thread callout should come before the counterbore callout.

15 Modify the callout to list the thread callout above the counterbore callout.

Access the **Dimension Text** box on the **Dimension Manager** by clicking the **.31 THRU ALL** dimension, delete the first line of text, and replace the dimension with a new dimension, **3/8-16 UNC THRU**.

Figure 7-43



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

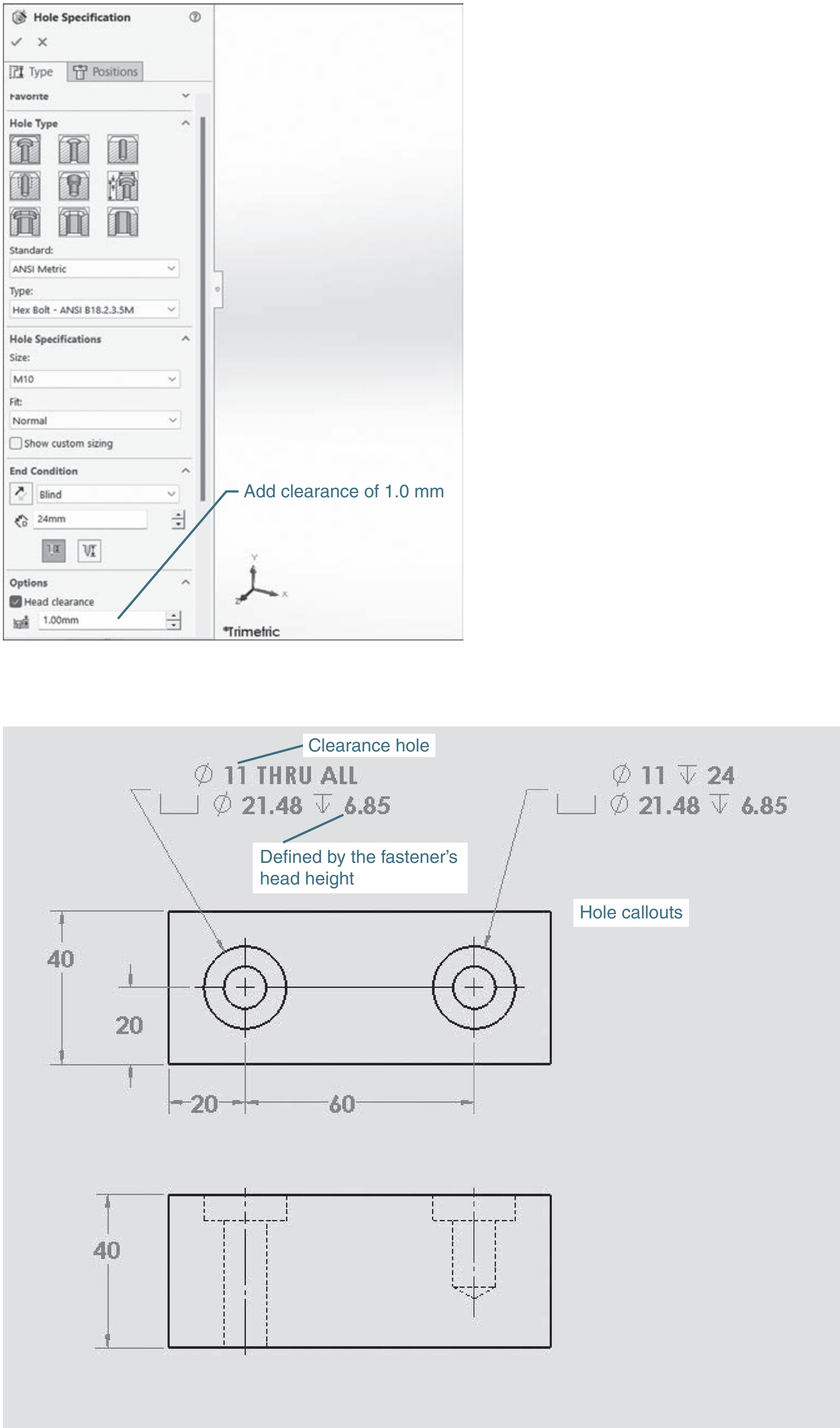
17 Click the threaded portion of the right hole.

18 Locate the text and click the mouse.

19 Modify the callout as shown.

Figure 7-44 shows dimensioned counterbored holes using metric units. The **Hole Callout** tool was used to dimension the counterbored holes. Note that the hole's diameter is listed as Ø11. The fastener size was specified as M10, and the Ø11 hole is a clearance hole.

Figure 7-44

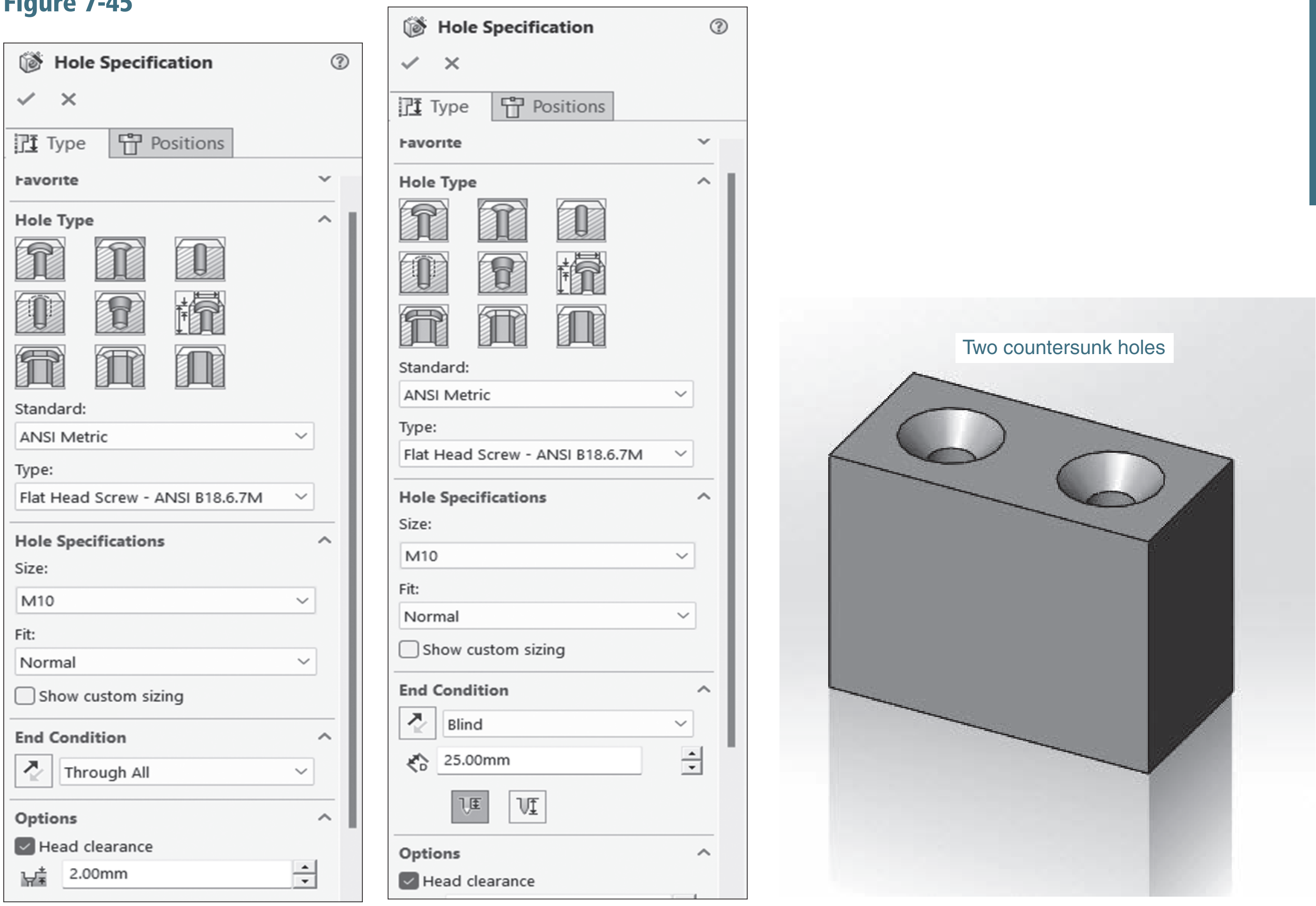

Hole Specification
Type
Positions
Favorite
Hole Type
Standard:
ANSI Metric
Type:
Hex Bolt - ANSI B18.2.3.5M
Hole Specifications
Size:
M10
Fit:
Normal
Show custom sizing
End Condition
Blind
24mm
Options
Head clearance
1.00mm
*Trimetric
Add clearance of 1.0 mm
Clearance hole
⌀ 11 THRU ALL
⌴ ⌀ 21.48 ↧ 6.85
⌀ 11 ↧ 24
⌴ ⌀ 21.48 ↧ 6.85
Defined by the fastener's head height
Hole callouts
40
20
20
60
40

Dimensioning Countersink Holes

Countersink holes are used with flat head screws to create assemblies in which the fasteners do not protrude above the surfaces.

Figure 7-45 shows a part with two countersunk holes; one goes completely through, the other has a depth specification.

Figure 7-45



1. Draw a 40 × 80 × 60 block.
2. Use the **Hole Wizard** tool, click the **Countersink** type, specify the **ANSI Metric** standard, select an **M10** size for a flat head screw, and specify a hole that goes all the way through. Define a head clearance of **2.00mm**.
3. Click the **Positions** tab and position the countersunk hole's center-point as shown using the **Smart Dimension** tool.
4. Click the green **OK** check mark.
5. Click the **Hole Wizard** tool, click the **Countersink** type, specify the **ANSI Metric** standard, select an **M10** size for a flat head screw, and specify a depth requirement of **25.00mm** for a **Blind** hole. Define a head clearance of **2.00mm**.

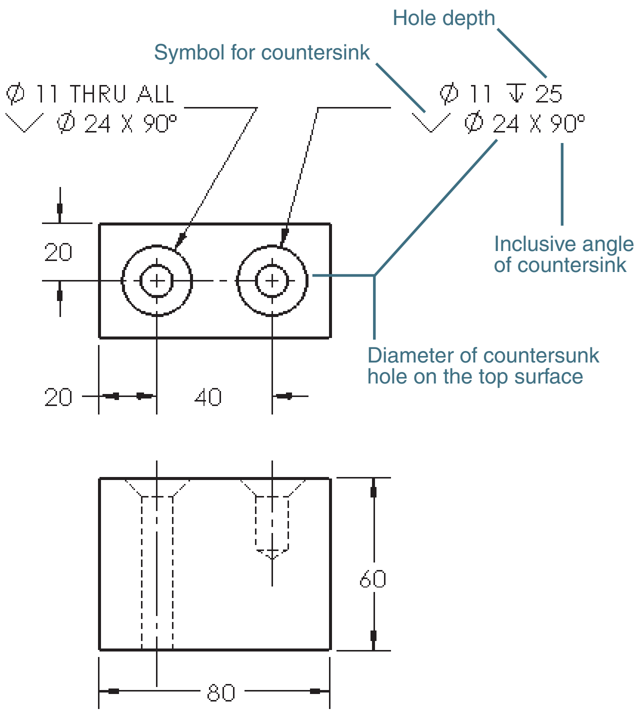
6. Click the **Positions** tab and locate the hole as shown.
7. Click the green **OK** check mark.
8. Save the drawing as **Block, CSink**.

Dimensioning the Block

1. Create a new **Drawing** document with a front and a top orthographic view of the **Block, CSink**.
2. Use the **Smart Dimension** tool and add the appropriate dimensions.
3. Use the **Center Mark** tool to add a centerline between the two holes indicating they are aligned.
4. Click the **Annotation** tab, click the **Hole Callout** tool, and dimension the two countersunk holes.

   See Figure 7-46.

Figure 7-46



Completed dimensions

7-8 Angular Dimensions

Figure 7-47 shows a model that includes a slanted surface and dimensioned orthographic views of the model. The dimension values are located beyond the model between two extension lines. Locating dimensions between extension lines is preferred to locating the value between an extension line and the edge of the model.

Figure 7-47

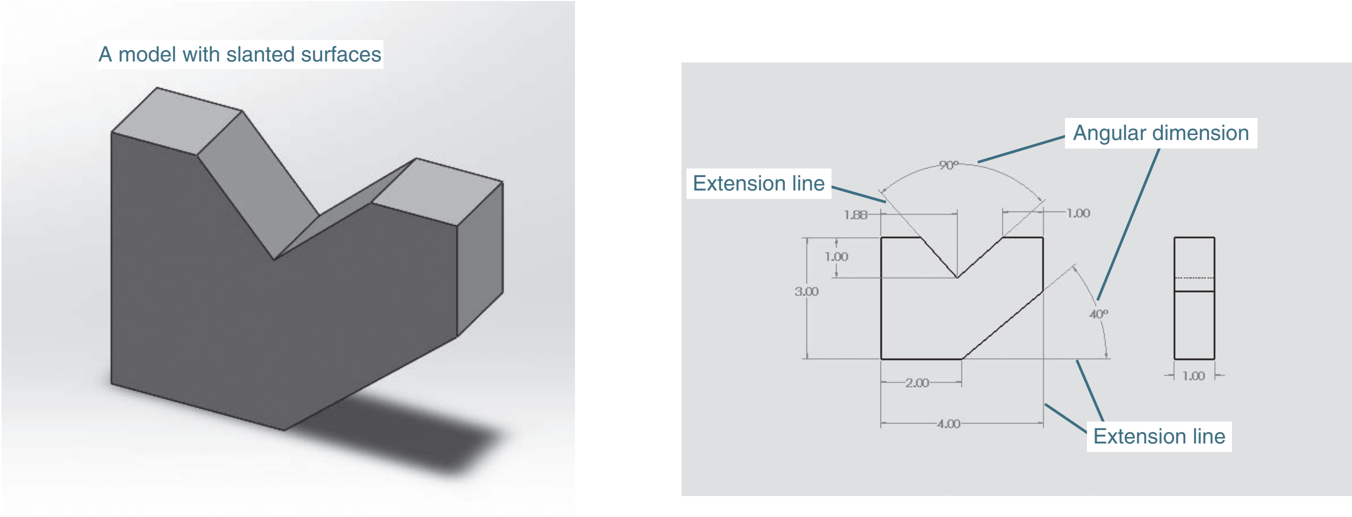


Figure 7-48 shows a shape that includes a slanted surface dimensioned in two different ways. The shape on the left uses an angular dimension; the one on the right does not. Both are acceptable.

Figure 7-48

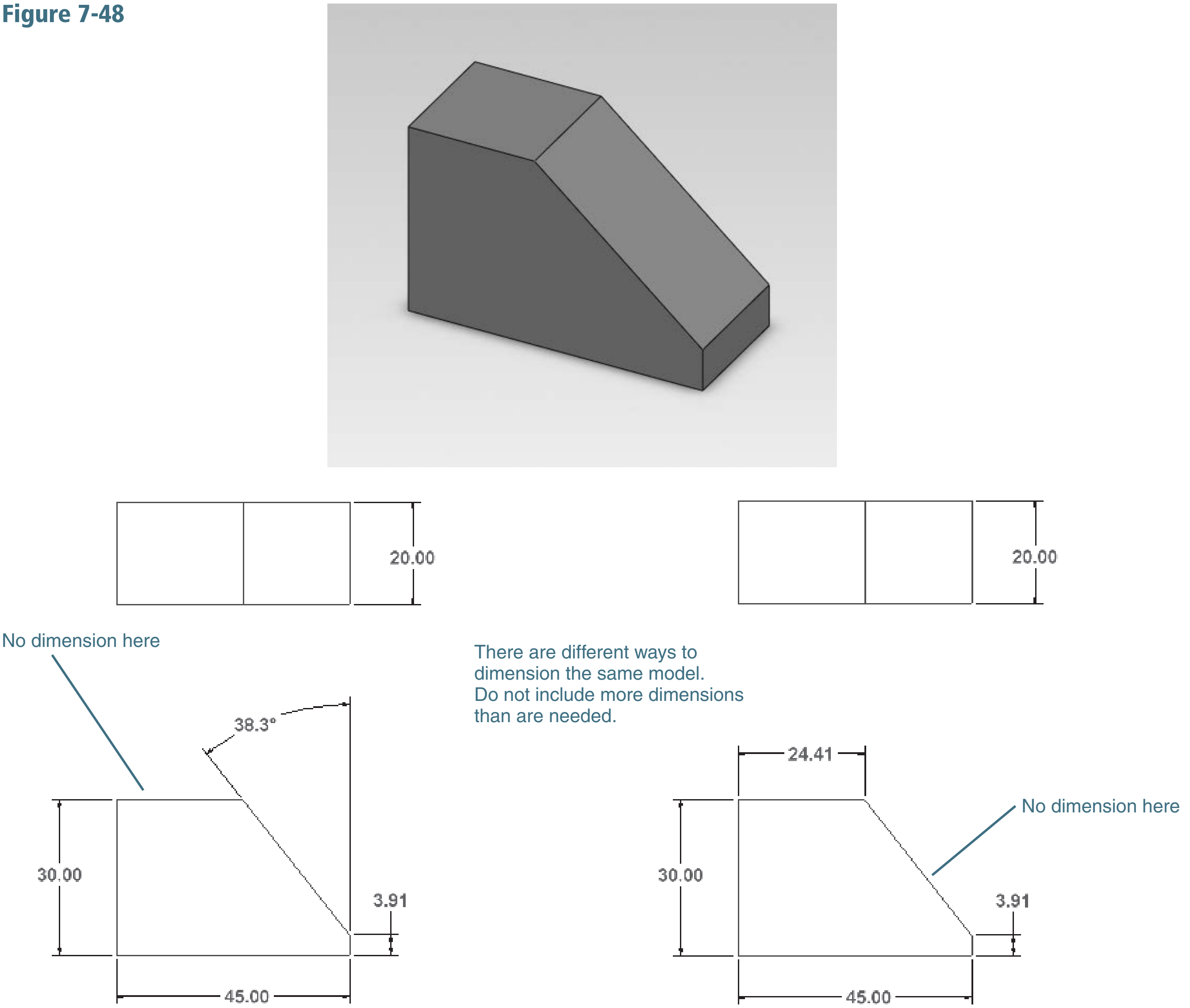
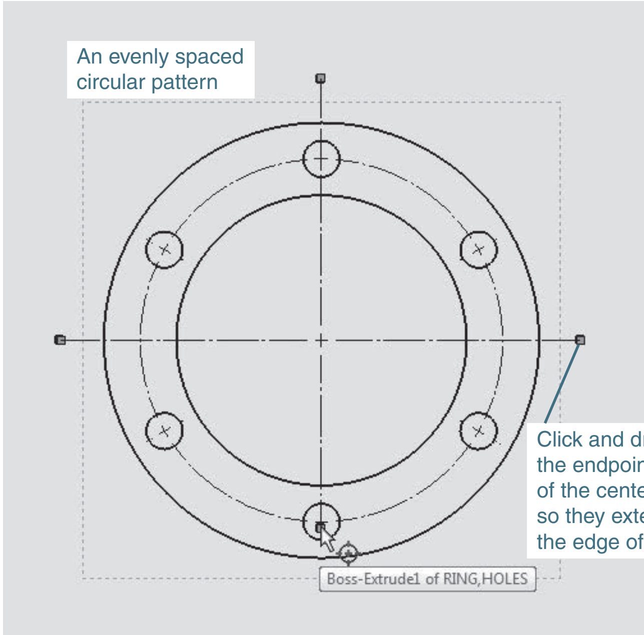


Figure 7-49 shows two objects dimensioned using angular dimensions. One has an evenly spaced hole pattern; the other has an uneven hole pattern.

Figure 7-49



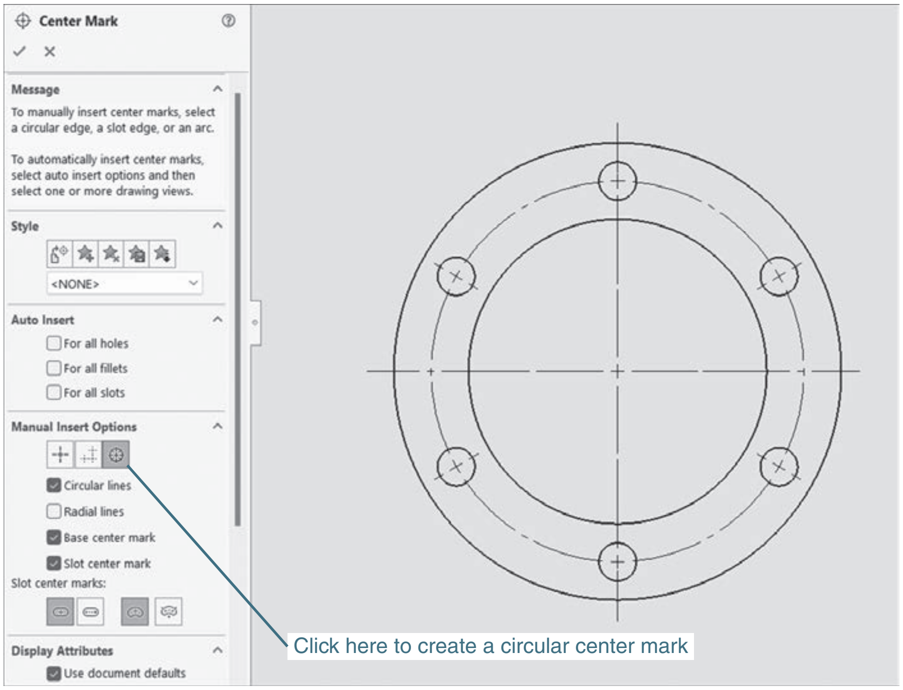
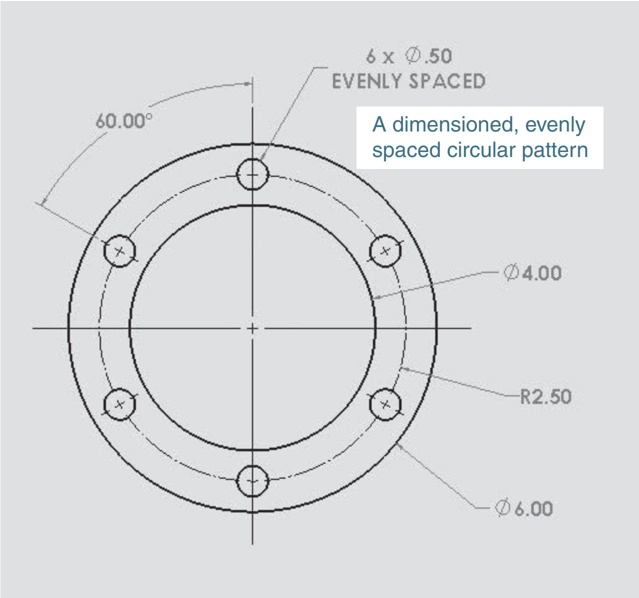
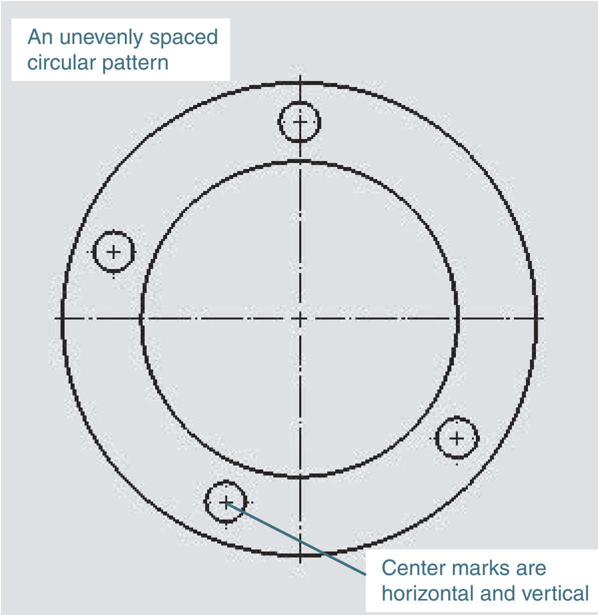


Figure 7-49
(*Continued*)





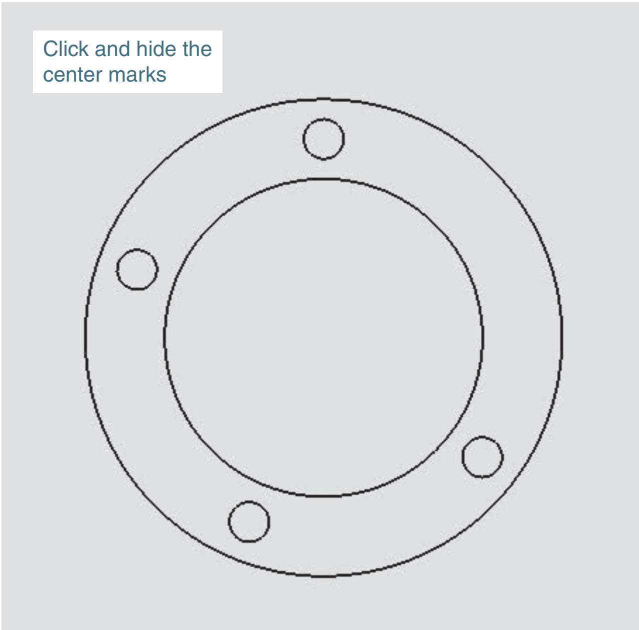
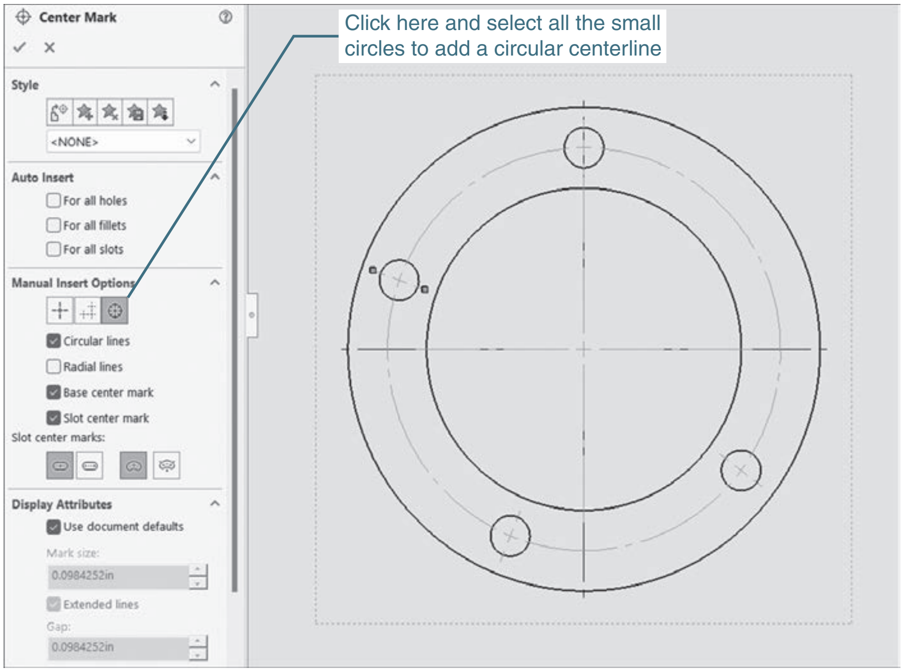
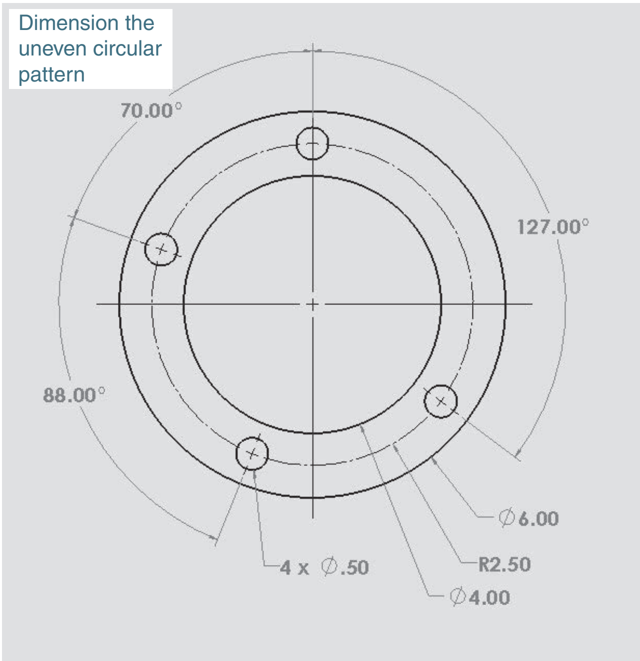


Figure 7-49
(*Continued*)





Dimensioning an Evenly Spaced Hole Pattern

1 Start a new drawing of the object and create a view as shown in Figure 7-49.

The object will automatically include circular centerlines. The circular centerline is called a *bolt circle*. Note that the center marks are not horizontal and vertical but point at the centerpoint of the pattern.

Circular centerlines and center marks can be created using the **Manual Insert Options** located on the **Center Mark PropertyManager**.

2 Add dimensions to the pattern and the object.

The six holes are evenly spaced and are all the same size, so only one angular dimension and a note are needed, as shown. All the holes are the same distance from the centerpoint, so the circular centerline needs only one dimension that will include the six holes.

The size and text position of the angular dimension can be edited using the **System** tool, **Document Properties**, **Dimensions**, **Angle**, and entering edits.

Figure 7-49 shows a similar object but with an uneven hole pattern. Each hole must be dimensioned separately.

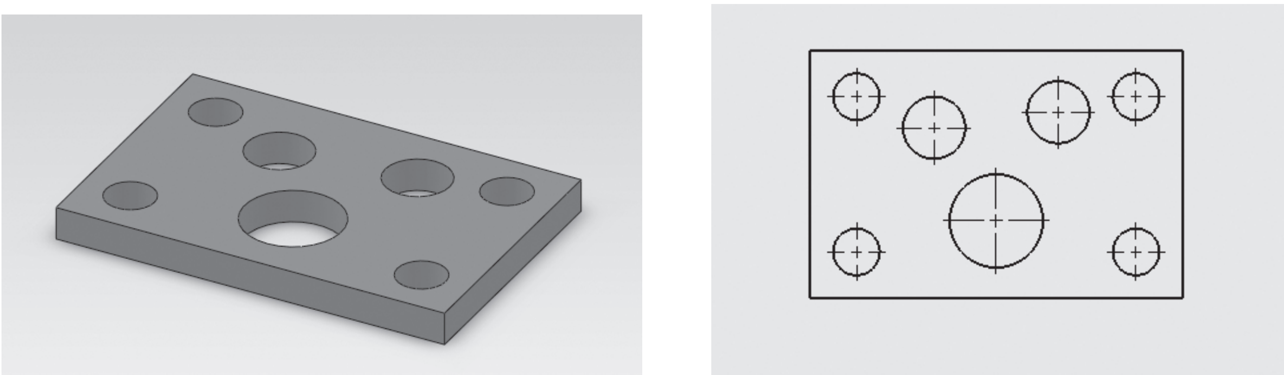
When the drawing view first appears on the screen, all the center marks are horizontal and vertical. A circular centerline pattern is preferred. Click each center mark and **Hide** the mark. Click the **Center Mark** tool and the **Circular Center Mark** tool located under the **Manual Insert Options**, and click each hole. A circular centerline pattern will appear. The shape can then be dimensioned using the circular pattern.

7-9 Ordinate Dimensions

Ordinate dimensions are dimensions based on an XY coordinate system. Ordinate dimensions do not include extension lines, dimension lines, or arrowheads but simply horizontal and vertical leader lines drawn directly from the features of the object. Ordinate dimensions are particularly useful when dimensioning an object that includes many small holes.

Figure 7-50 shows a part that is to be dimensioned using ordinate dimensions. Ordinate dimensions' values are calculated from the XY origin, which, in this example, is the lower-left corner of the front view of the model.

Figure 7-50



Creating Ordinate Dimensions

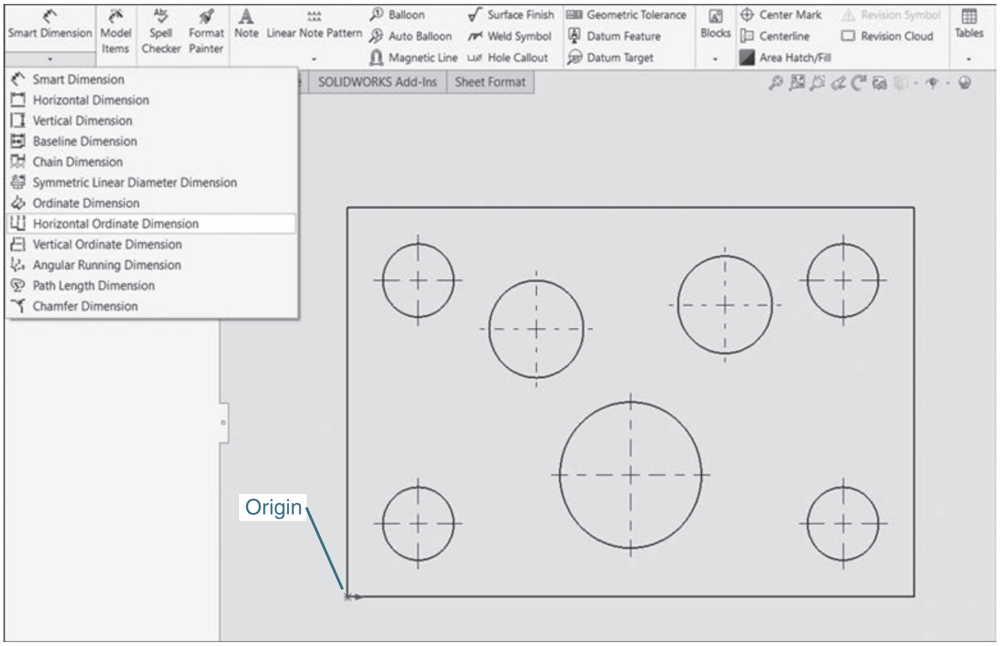
See Figure 7-51.

1 Start a new **Drawing** document and create a top orthographic view of the part.

Use the dimensions shown in Figure 7-52 to create the drawing.

2 Click and extend the center marks and draw centerlines between the four corner holes.

Figure 7-51



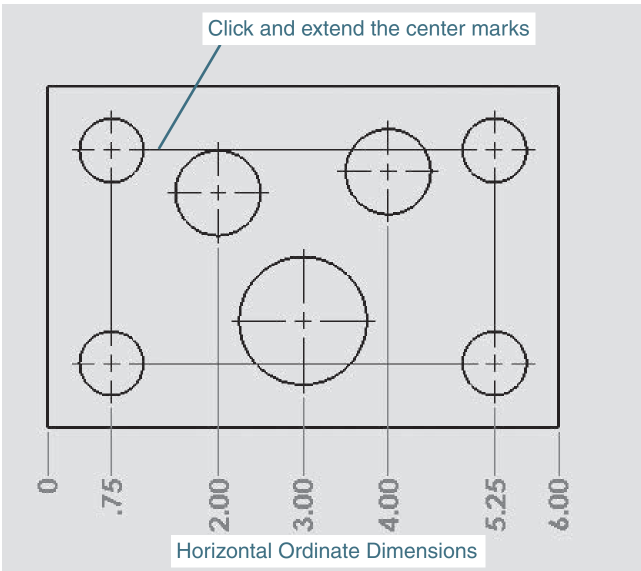
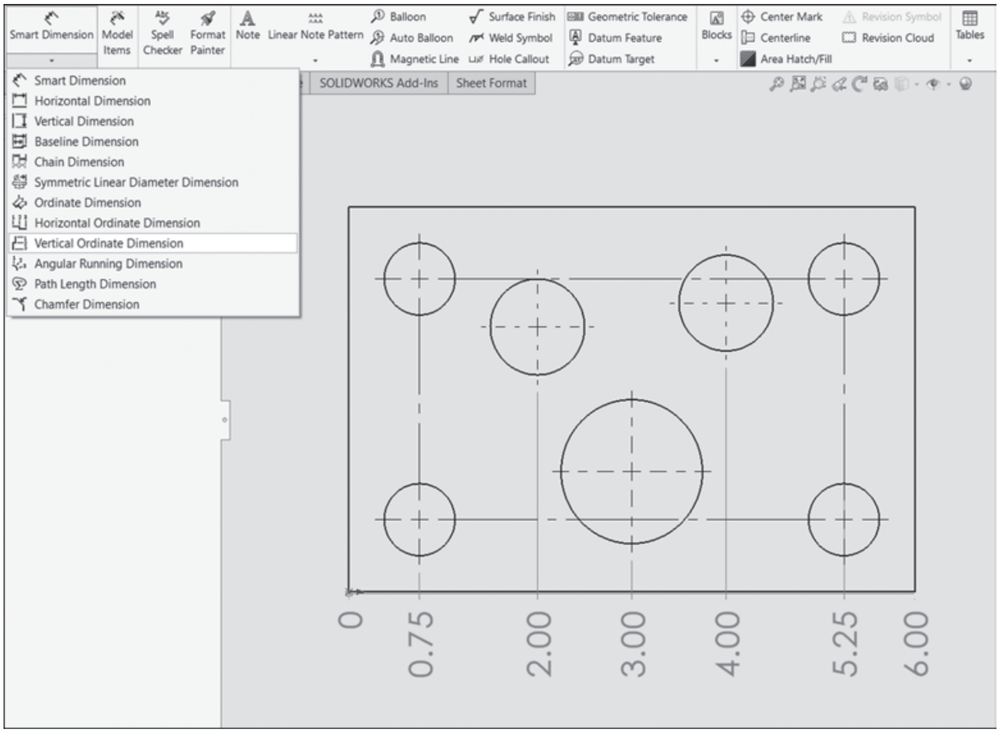
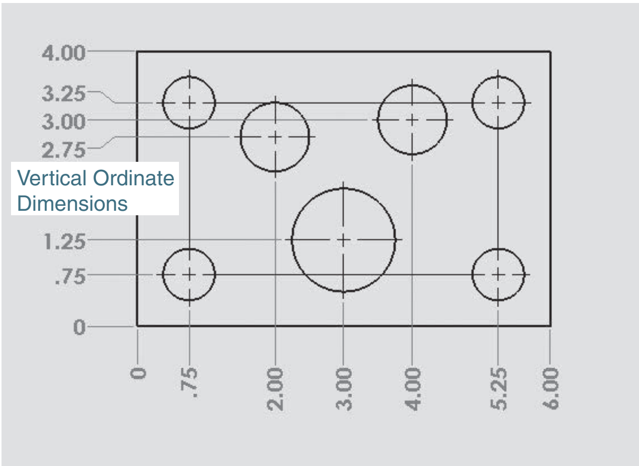


Figure 7-51
(Continued)





3 Click the arrowhead located under the **Smart Dimension** tool and click the **Horizontal Ordinate Dimension** option.

4 Click the lower-left corner of the part to establish the origin for the dimensions.

5 Move the cursor away from the origin and define a location for the "0" dimension.

All other horizontal dimensions will align with this location.

6 Click the lower portion of each hole's vertical centerline and the lower-right corner of the part.

7 Click the arrowhead located under the **Smart Dimension** tool and click the **Vertical Ordinate Dimension** option.

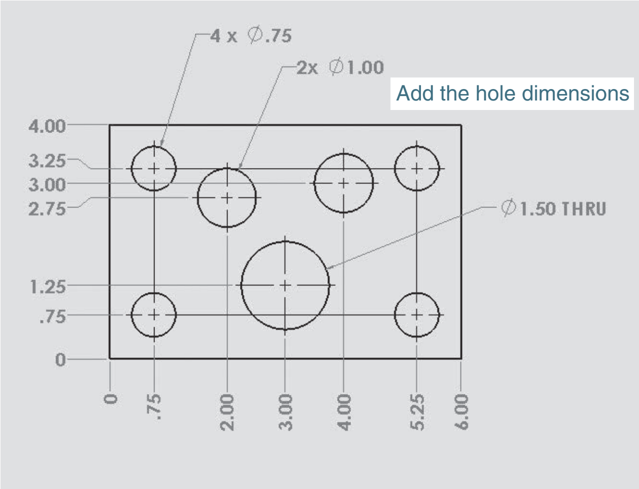
8 Click the lower-left corner of the part to establish the origin for the dimensions.

9 Click the left portion of each hole's horizontal centerline and the upper-left corner of the part.

10 Add dimensions for the holes.

Figure 7-52 shows the dimensioned part.

Figure 7-52



7-10 Baseline Dimensions

Baseline dimensions are a series of dimensions that originate from a common baseline or datum line. Baseline dimensions are very useful because they help eliminate the tolerance buildup that is associated with chain-type dimensions.

Creating Baseline Dimensions

See Figure 7-53.

1 Start a new **Drawing** document and create a top orthographic view of the part.

2 Use the **Linear Center Mark** tool and add connection centerlines between the four corner holes.

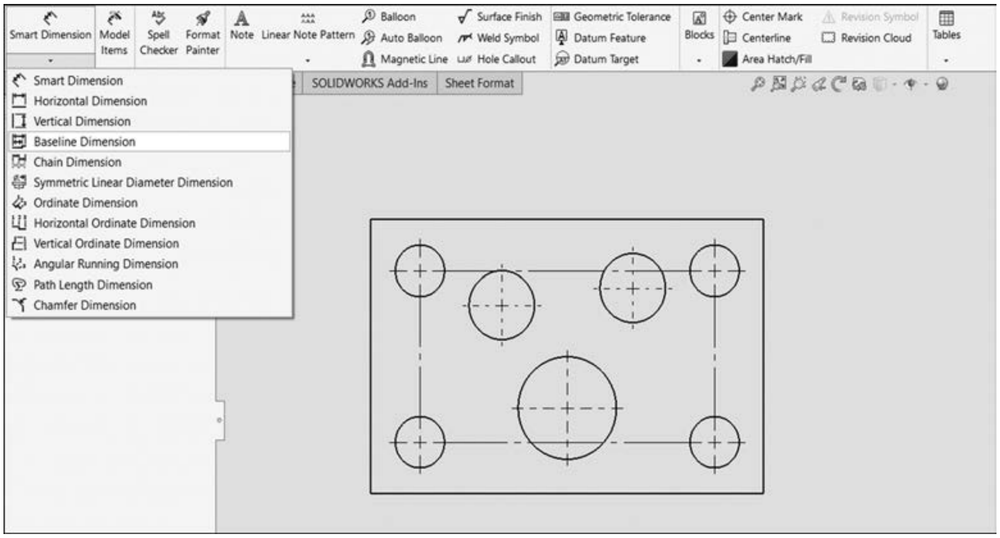
3 Click the arrowhead under the **Smart Dimension** tool and click the **Baseline Dimension** option.

4 Click the left vertical edge of the part and the lower portion of the first vertical centerline.

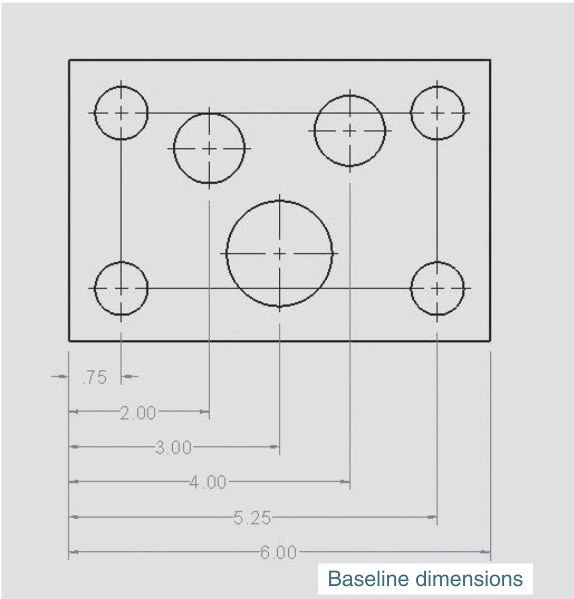
This will establish the baseline.

5 Click the lower portion of each vertical centerline and the right vertical edge line and locate the dimensions.

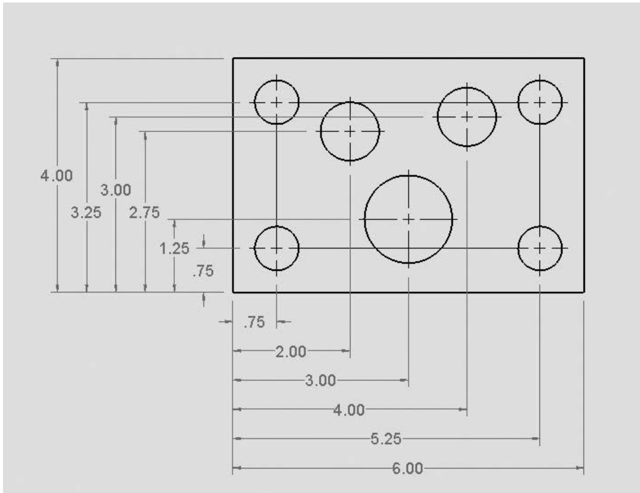
Figure 7-53



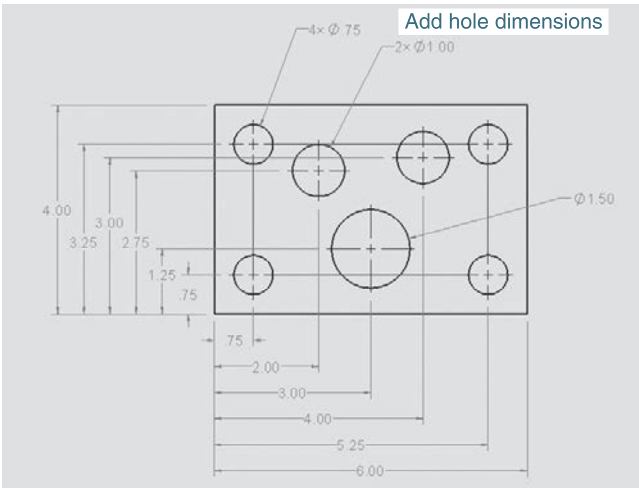
Smart Dimension
Model Items
Spell Checker
Format Painter
Note
Linear Note Pattern
Balloon
Auto Balloon
Magnetic Line
Surface Finish
Weld Symbol
Hole Callout
Geometric Tolerance
Datum Feature
Datum Target
Blocks
Center Mark
Centerline
Area Hatch/Fill
Revision Symbol
Revision Cloud
Tables
SOLIDWORKS Add-Ins
Sheet Format
Smart Dimension
Horizontal Dimension
Vertical Dimension
Baseline Dimension
Chain Dimension
Symmetric Linear Diameter Dimension
Ordinate Dimension
Horizontal Ordinate Dimension
Vertical Ordinate Dimension
Angular Running Dimension
Path Length Dimension
Chamfer Dimension



.75
2.00
3.00
4.00
5.25
6.00
Baseline dimensions



4.00
3.00
3.25
2.75
1.25
.75
.75
2.00
3.00
4.00
5.25
6.00



Add hole dimensions
4× Ø.75
2× Ø1.00
Ø1.50
4.00
3.00
3.25
2.75
1.25
.75
.75
2.00
3.00
4.00
5.25
6.00

> **NOTE**
> The distance between the dimension lines can be changed in the **Offset distances** box under **Dimensions** on the **Document Properties** tab of the **Options** tool.

6 Click the arrowhead under the **Smart Dimension** tool and click the **Baseline Dimension** option.

7 Click the lower horizontal edge of the part and the left end of the first horizontal centerline.

8 Click the left end of each horizontal centerline and the right top horizontal edge line.

The alignment of the vertical dimension lines can be changed by right-clicking the individual dimension and selecting the **Break Alignment** option.

9 Add the hole dimensions.

Hole Tables

Hole tables are a method for dimensioning parts that have large numbers of holes where standard dimensioning may be cluttered and difficult to read. See Figure 7-54.

Figure 7-54

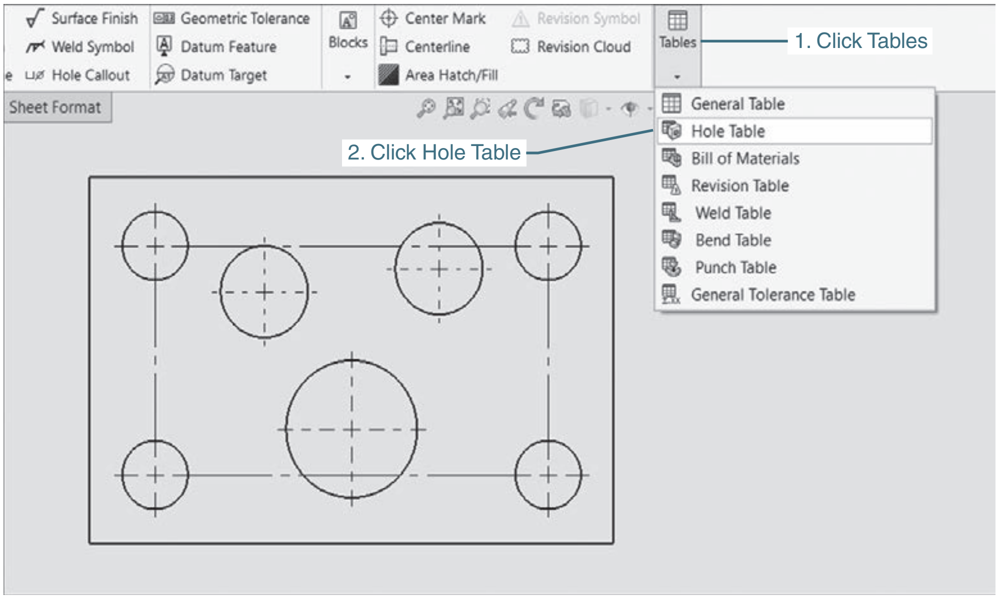
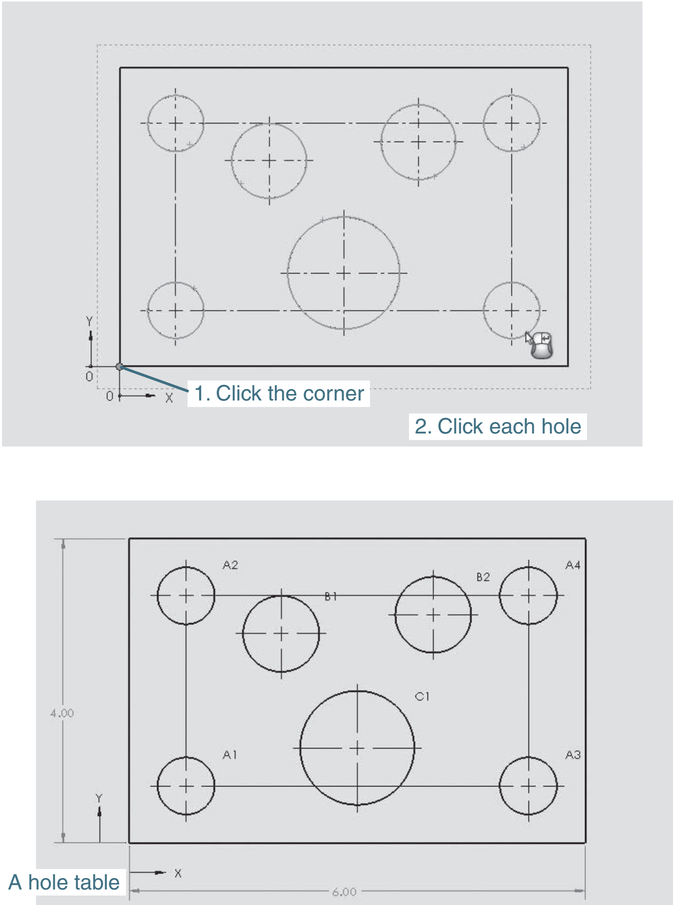


Figure 7-54
(Continued)



TAG	X LOC	Y LOC	SIZE
A1	.75	.75	Ø.75 THRU
A2	.75	3.25	Ø.75 THRU
A3	5.25	.75	Ø.75 THRU
A4	5.25	3.25	Ø.75 THRU
B1	2.00	2.75	Ø1.00 THRU
B2	4.00	3.00	Ø1.00 THRU
C1	3.00	1.25	Ø1.50 THRU

1. Start a new **Drawing** document and create a top orthographic view of the part.
2. Use the **Linear Center Mark** tool and add connection centerlines between the four corner holes.
3. Click the **Annotation** tab, click **Tables**, and click **Hole Table**.
4. Click the lower-left corner of the part to establish an origin.
5. Click each hole.

As the holes are clicked they should be listed in the **Holes** box located in the **Hole Table PropertyManager**.

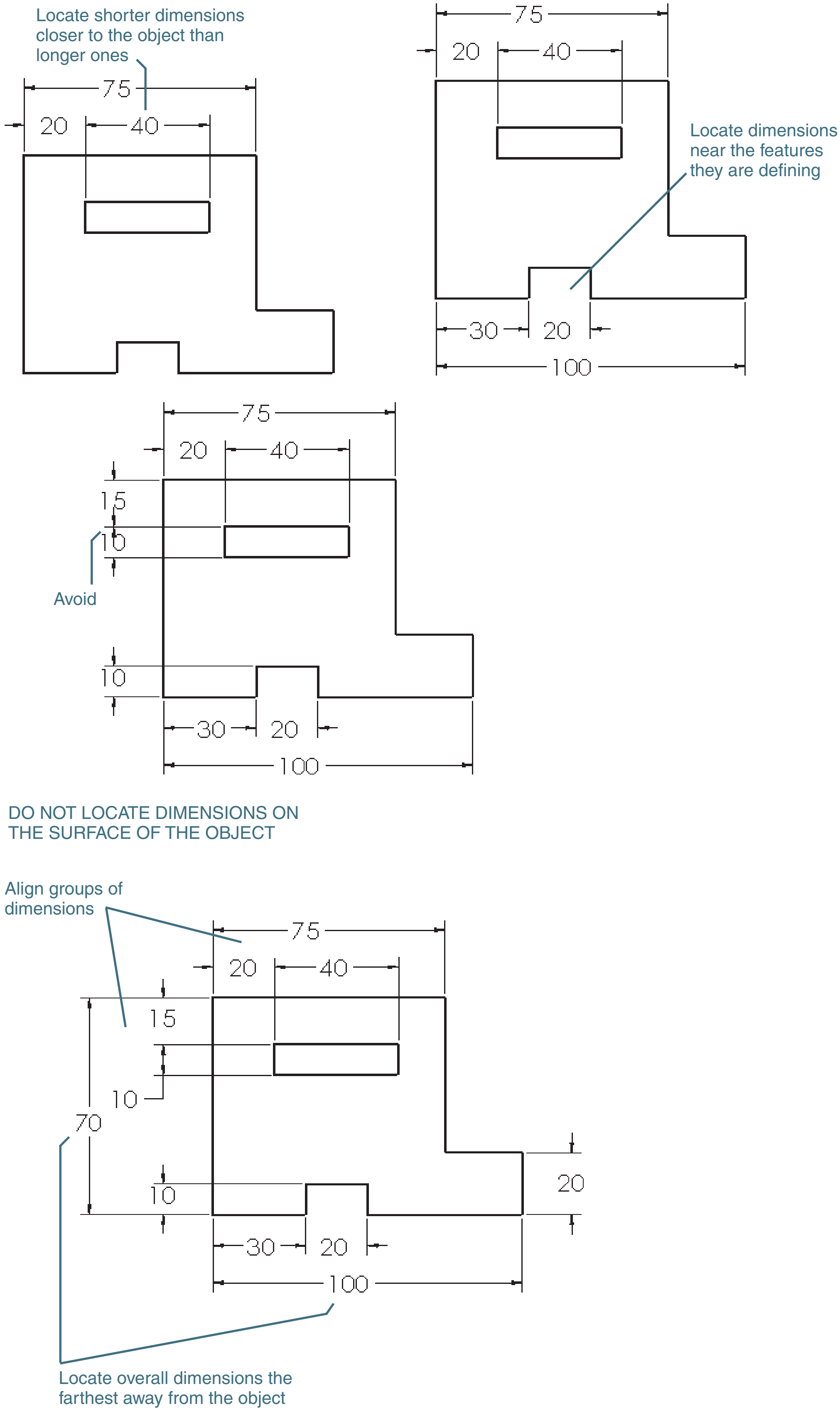
6. Click the green **OK** check mark and locate the hole table.
7. Add the overall dimensions.
8. Move the hole tags as needed to present a clear, easy-to-read drawing.

In this example all tags were located to the upper-right of the holes they define. Tables can be edited using the instructions presented in Section 5-11 for BOMs.

7-11 Locating Dimensions

There are eight general rules concerning the location of dimensions. See Figure 7-55.

Figure 7-55



1 Locate dimensions near the features they are defining.

2 Do not locate dimensions on the surface of the object.

3 Align and group dimensions so that they are neat and easy to understand.

4 Avoid crossing extension lines.

Sometimes it is impossible not to cross extension lines because of the complex shape of the object, but whenever possible, avoid crossing extension lines.

5 Do not cross dimension lines.

6 Locate shorter dimensions closer to the object than longer dimensions.

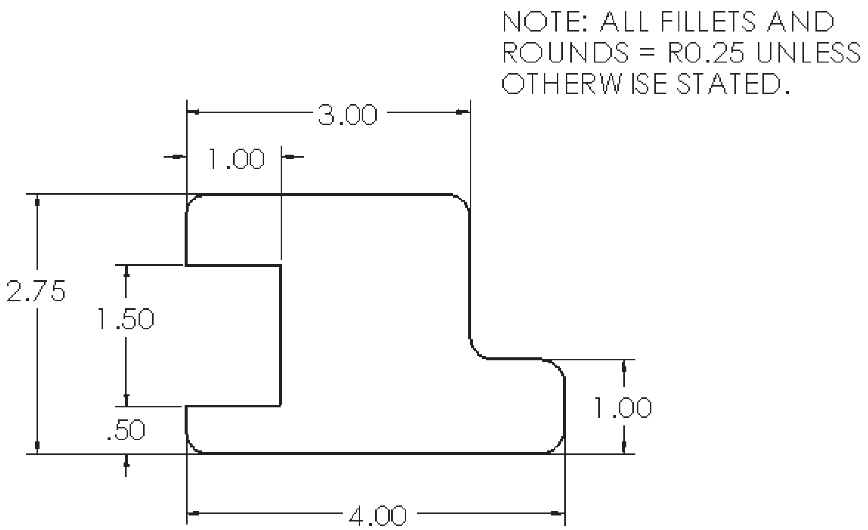
7 Always locate overall dimensions the farthest away from the object.

8 Do not dimension the same distance twice. This is called *double dimensioning* and will be discussed in Chapter 8 in association with tolerancing.

7-12 Fillets and Rounds

Fillets and rounds may be dimensioned individually or by a note. In many design situations all the fillets and rounds are the same size, so a note as shown in Figure 7-56 is used. Any fillets or rounds that have a different radius from that specified by the note are dimensioned individually.

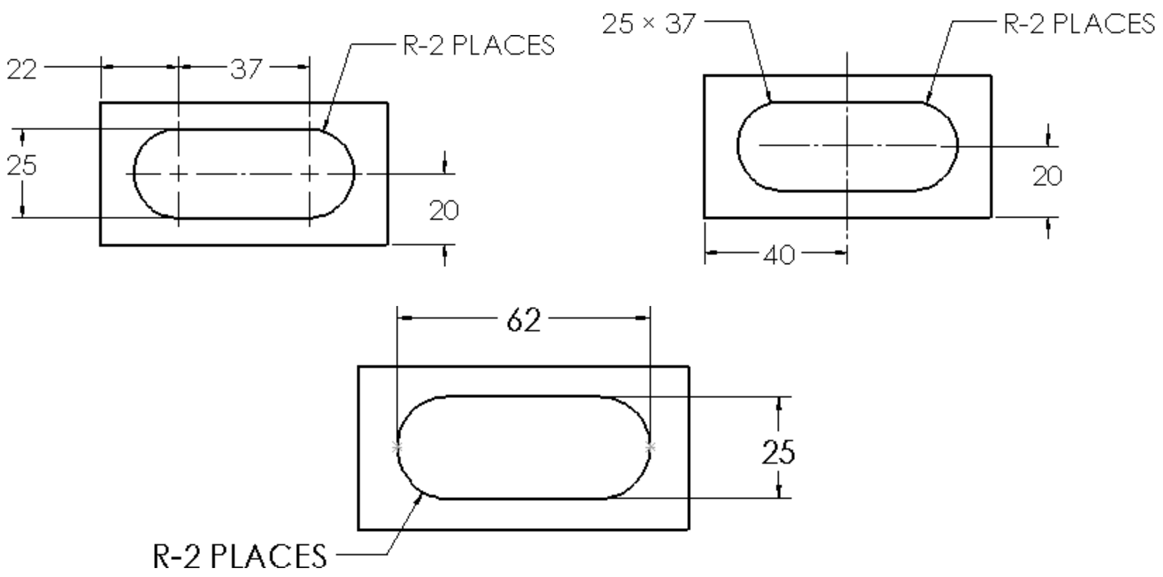
Figure 7-56



7-13 Rounded Shapes—Internal

Internal rounded shapes are called *slots*. Figure 7-57 shows three different methods for dimensioning slots. The end radii are indicated by the note **R-2 PLACES**, but no numerical value is given. The width of the slot is dimensioned, and it is assumed that the radius of the rounded ends is exactly half of the stated width.

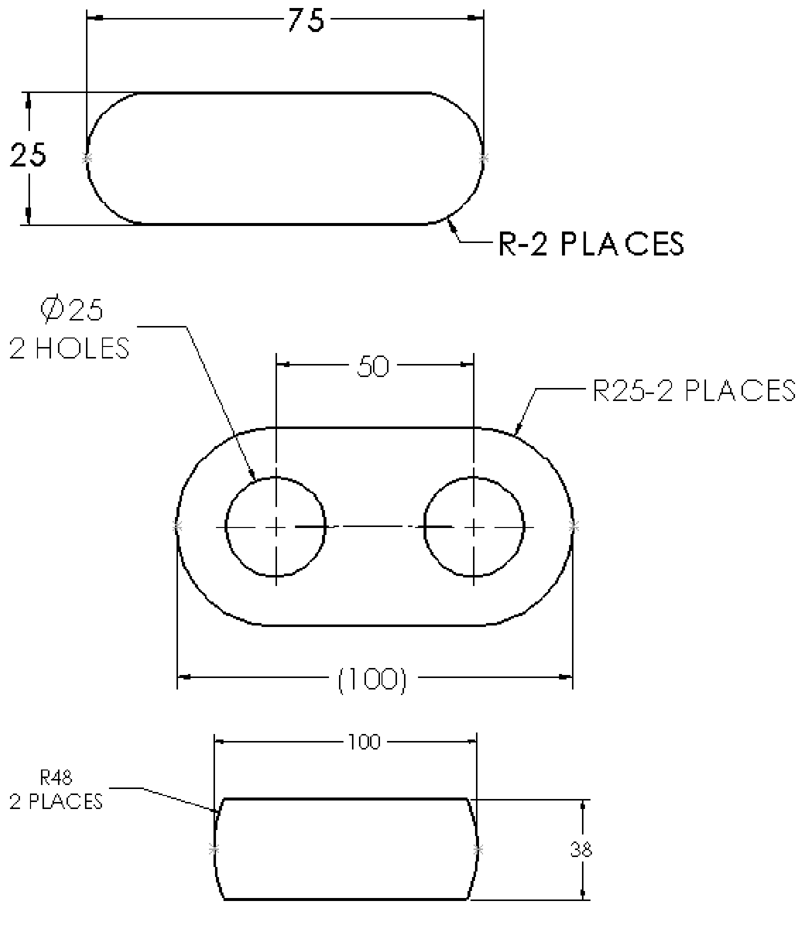
Figure 7-57



7-14 Rounded Shapes—External

Figure 7-58 shows two shapes with external rounded ends. As with internal rounded shapes, the end radii are indicated, but no value is given. The width of the object is given, and the radius of the rounded end is assumed to be exactly half of the stated width.

Figure 7-58



The second example shown in Figure 7-58 shows an object dimensioned using the object's centerline. This type of dimensioning is done when the distance between the holes is more important than the overall length of the object; that is, the tolerance for the distance between the holes is more exact than the tolerance for the overall length of the object.

The overall length of the object is given as a reference dimension (100). This means the object will be manufactured based on the other dimensions, and the 100 value will be used only for reference.

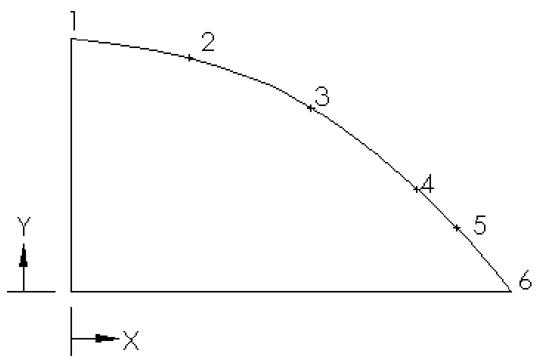
Objects with partially rounded edges should be dimensioned as shown in Figure 7-58. The radii of the end features are dimensioned. The centerpoint of the radii is implied to be on the object centerline. The overall dimension is given; it is not referenced unless specific radii values are included.

7-15 Irregular Surfaces

There are three different methods for dimensioning irregular surfaces: tabular, baseline, and baseline with oblique extension lines. Figure 7-59 shows an irregular surface dimensioned using the tabular method. An XY axis is defined using the edges of the object. Points are then defined relative to the XY axis. The points are assigned reference numbers, and the reference numbers and XY coordinate values are listed in chart form as shown.

Figure 7-59

STATION	1	2	3	4	5	6
X	0	20	40	55	62	70
Y	40	38	30	16	10	0



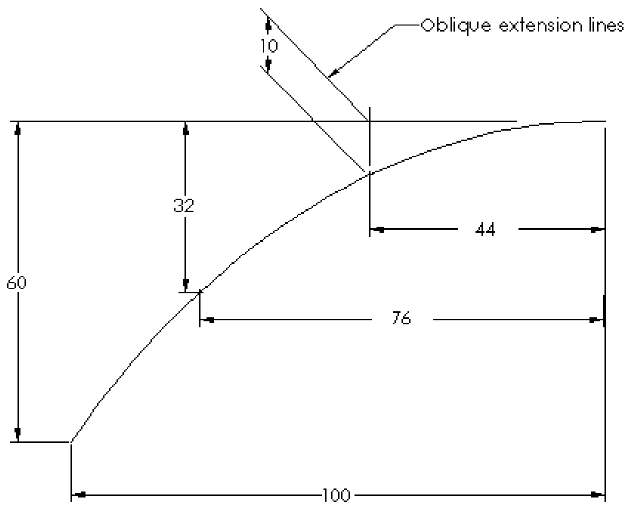
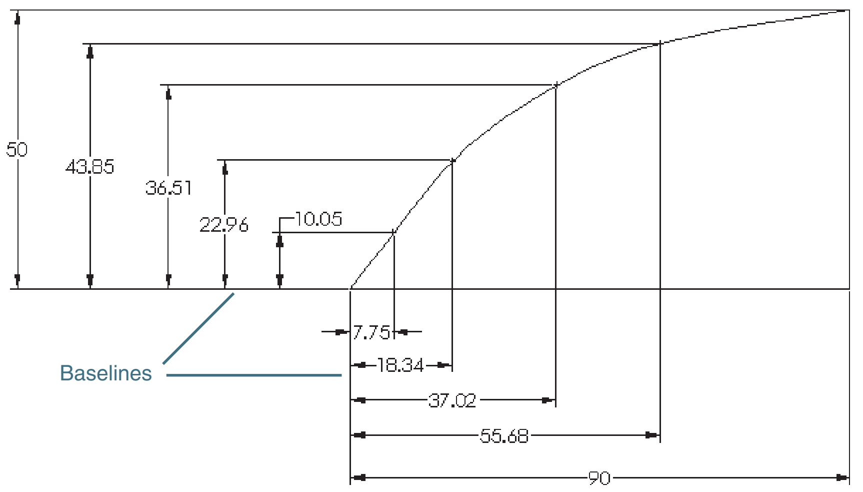


Figure 7-60 shows an irregular curve dimensioned using baseline dimensions. The baseline method references all dimensions to specified baselines. Usually there are two baselines, one horizontal and one vertical.

Figure 7-60



It is considered poor practice to use a centerline as a baseline. Centerlines are imaginary lines that do not exist on the object and would make it more difficult to manufacture and inspect the finished objects.

Baseline dimensioning is very common because it helps eliminate tolerance buildup and is easily adaptable to many manufacturing processes.

7-16 Polar Dimensions

Polar dimensions are similar to polar coordinates. A location is defined by a radius (distance) and an angle. Figure 7-61 shows an object that includes polar dimensions. The holes are located on a circular centerline, and their positions from the vertical centerline are specified using angles.

Figure 7-61

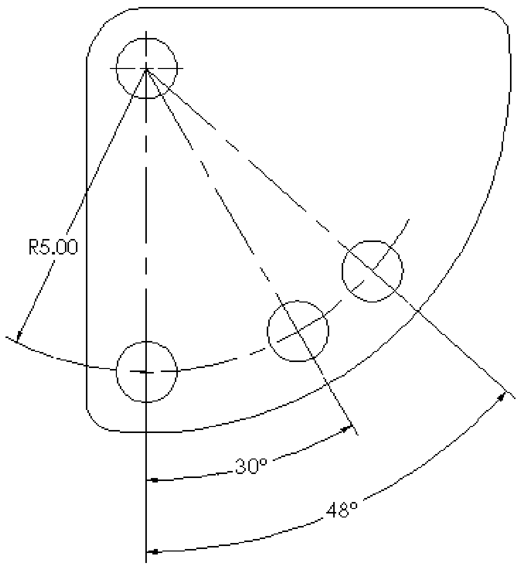
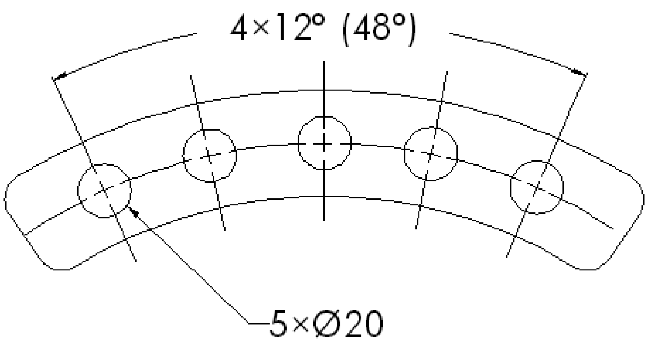


Figure 7-62 shows an example of a hole pattern dimensioned using polar dimensions.

Figure 7-62



7-17 Chamfers

Chamfers are angular cuts made on the edges of objects. They are usually used to make it easier to fit two parts together. They are most often made at 45° angles but may be made at any angle. Figure 7-63 shows two objects with chamfers between surfaces 90° apart and two examples between surfaces that are not 90° apart. Either of the two types of dimensions shown for the 45° dimension may be used. If an angle other than 45° is used, the angle and setback distance must be specified.

Figure 7-63

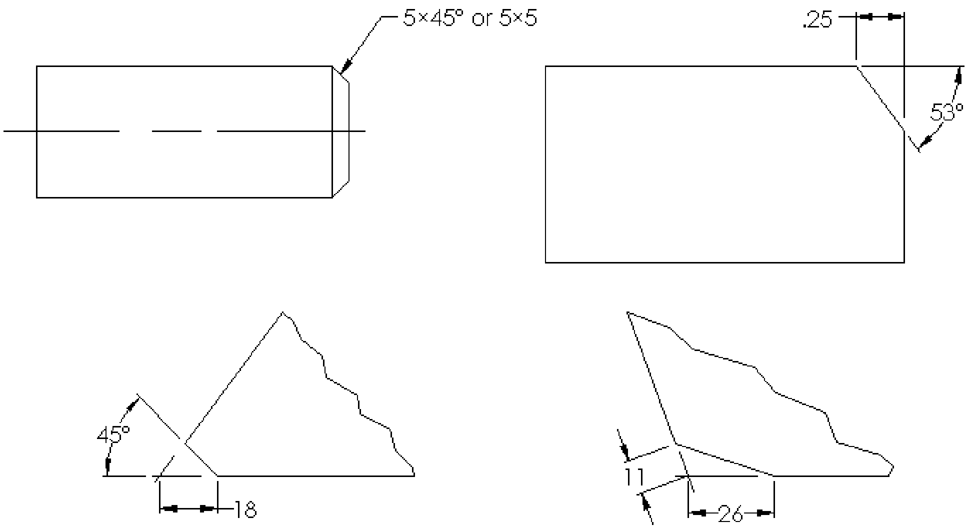
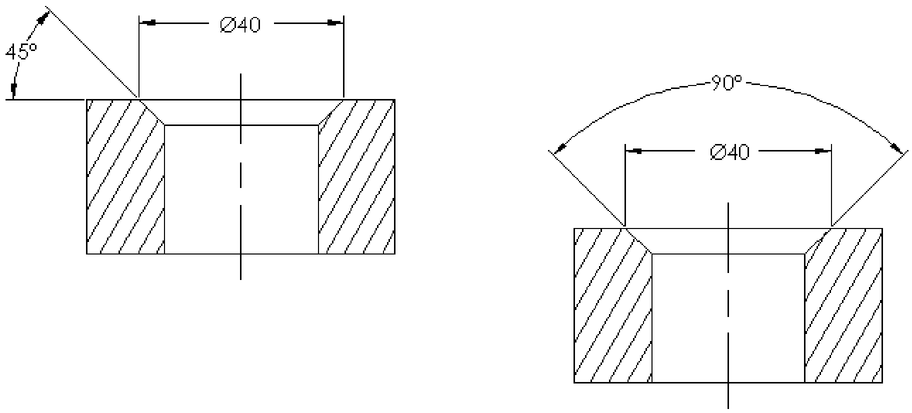


Figure 7-64 shows two examples of internal chamfers. Both define the chamfer using an angle and diameter. Internal chamfers are very similar to countersunk holes.

Figure 7-64



7-18 Symbols and Abbreviations

Symbols are used in dimensioning to help accurately display the meaning of the dimension. Symbols also help eliminate language barriers when reading drawings.

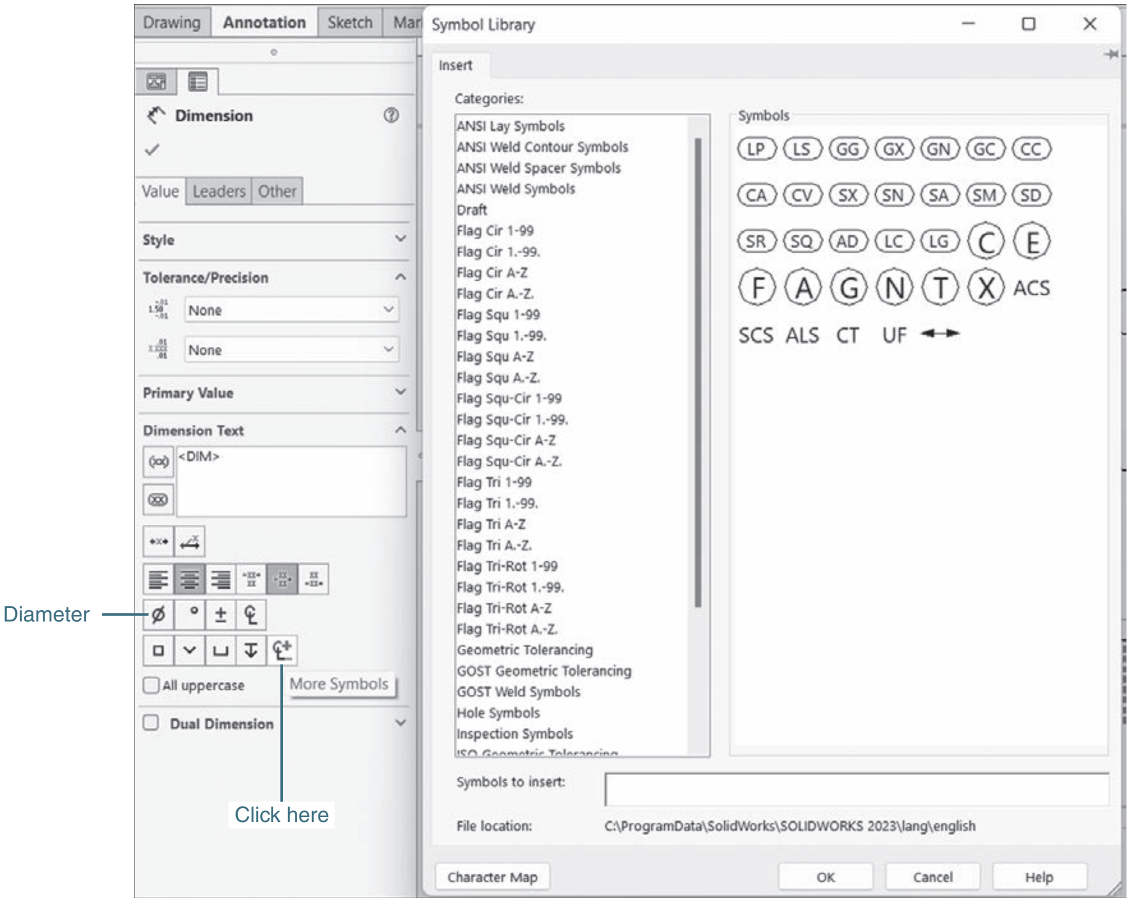
Abbreviations should be used very carefully on drawings. Whenever possible, write out the full word, including correct punctuation. The **Dimension PropertyManager Value** tab includes a group of symbols and words commonly used on technical drawings. Figure 7-65 lists several standard abbreviations used on technical drawings.

Figure 7-65

AL	=	Aluminum
CBORE	=	Counterbore
CRS	=	Cold Rolled Steel
CSK	=	Countersink
DIA	=	Diameter
EQ	=	Equal
HEX	=	Hexagon
MAT'L	=	Material
R	=	Radius
SAE	=	Society of Automotive Engineers
SFACE	=	Spotface
ST	=	Steel
SQ	=	Square
REQD	=	Required

Figure 7-66 shows a list of symbols available in the **Dimension PropertyManager Value** tab.

Figure 7-66



TIP

To access the **Dimension PropertyManager Value** tab, click an existing dimension.

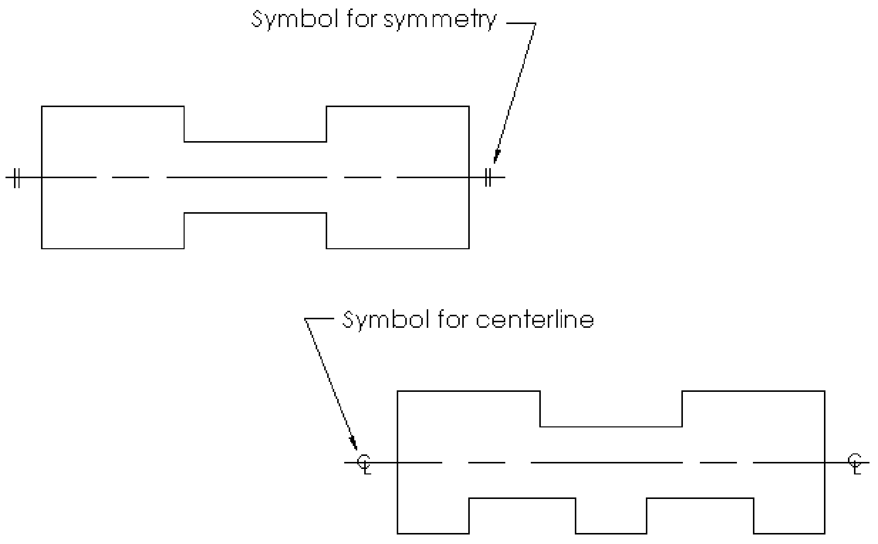
More symbols are available by clicking the **More** box. A list of available symbols will appear. Click a new symbol. A preview of the selected symbol will appear. Click **OK** and the symbol will appear on the drawing next to the existing symbol.

7-19 Symmetrical and Centerline Symbols

An object is symmetrical about an axis when one side is the exact mirror image of the other. Figure 7-67 shows a symmetrical object. The symbol of two short parallel lines or the note OBJECT IS SYMMETRICAL ABOUT THIS AXIS (centerline) may be used to designate symmetry.

If an object is symmetrical, only half the object needs to be dimensioned. The other dimensions are implied by the symmetry note or symbol.

Figure 7-67



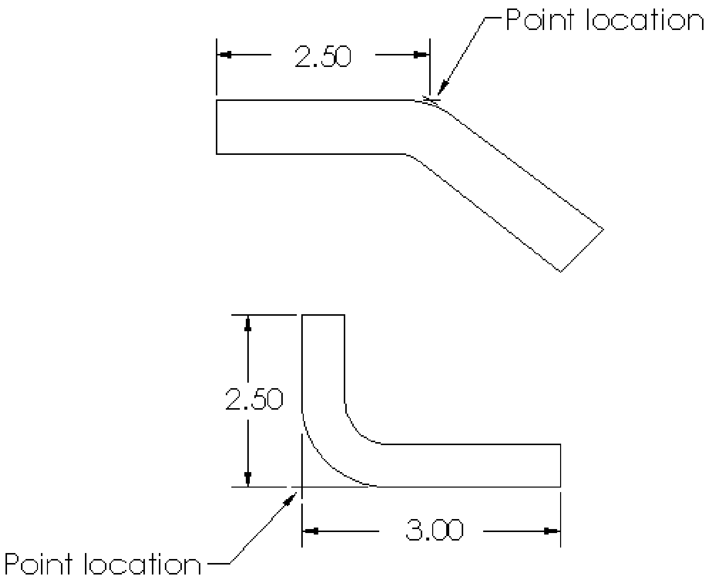
The centerline is slightly different from the axis of symmetry. An object may or may not be symmetrical about its centerline. See Figure 7-67. Centerlines are used to define the center of both individual features and entire objects. Use the centerline symbol when a line is a centerline, but do not use it in place of the symmetry symbol.

7-20 Dimensioning to a Point

Curved surfaces can be dimensioned using theoretical points. See Figure 7-68. There should be a small gap between the surface of the object and the lines used to define the theoretical point. The point should be defined by the intersection of at least two lines.

There should also be a small gap between the extension lines and the theoretical point used to locate the point.

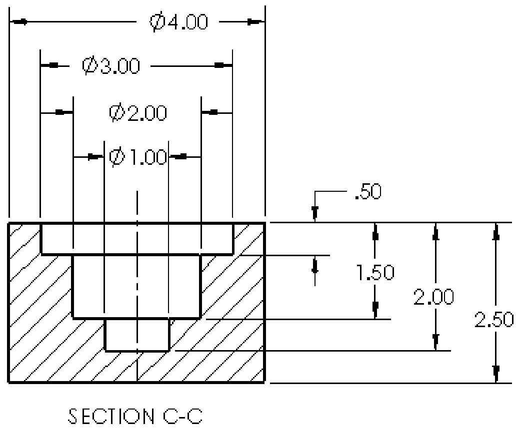
Figure 7-68



7-21 Dimensioning Section Views

Section views are dimensioned. See Figure 7-69. The section lines should be drawn at an angle that allows the viewer to clearly distinguish between the section lines and the extension lines.

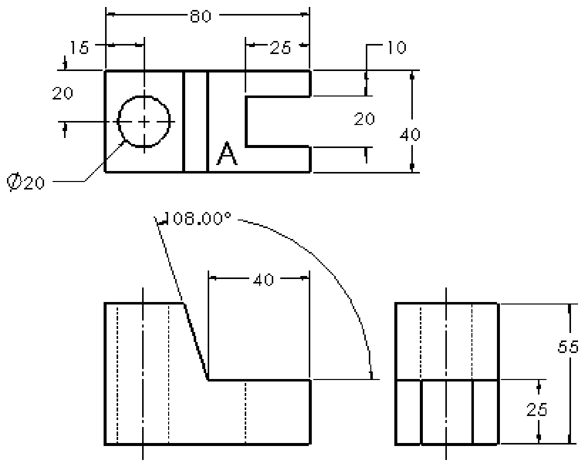
Figure 7-69



7-22 Dimensioning Orthographic Views

Dimensions should be added to orthographic views where the features appear in contour. Holes should be dimensioned in their circular views. Figure 7-70 shows three views of an object that has been dimensioned.

Figure 7-70



The hole dimensions are added to the top view, where the hole appears circular. The slot is also dimensioned in the top view because it appears in contour. The slanted surface is dimensioned in the front view.

The height of surface A is given in the side view rather than run along extension lines across the front view. The length of surface A is given in the front view. This is a contour view of the surface.

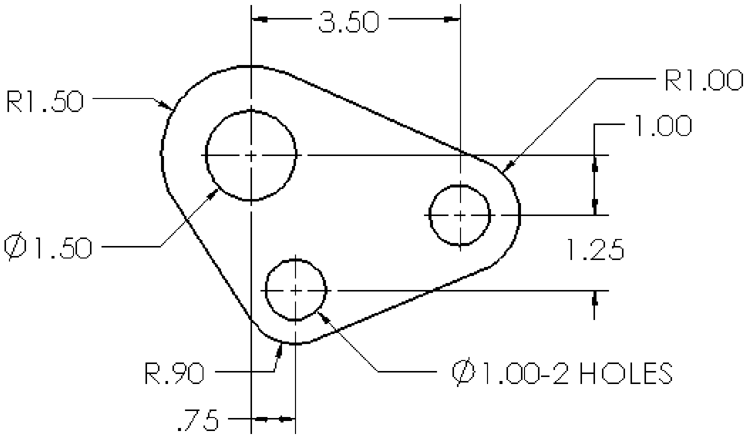
It is considered good practice to keep dimensions in groups. This makes it easier for the viewer to find dimensions.

Be careful not to double dimension a distance. A distance should be dimensioned only once. If a 30 dimension were added above the 25 dimension on the right-side view, it would be an error. The distance would be double dimensioned: once with the 25 + 30 dimension, and again with the 55 overall dimension. The 25 + 30 dimensions are mathematically equal to the 55 overall dimension, but there is a distinct difference in how they affect the manufacturing tolerances. Double dimensions are explained more fully in Chapter 8.

Dimensions Using Centerlines

Figure 7-71 shows an object dimensioned from its centerline. This type of dimensioning is used when the distance between the holes relative to each other is critical.

Figure 7-71



Chapter Projects

Project 7-1:

Measure and redraw the shapes in Figures P7-1 through P7-24. The dotted grid background has either .50in. or 10mm spacing. All holes are through holes. Specify the units and scale of the drawing. Use the **Part** template to create a model. Use the grid background pattern to determine the dimensions. Use the **Drawing** template to create the orthographic view shown. Use the **Smart Dimension** tool to dimension the view.

A. Measure using millimeters.

B. Measure using inches.

All dimensions are within either .25in. or 5mm. All fillets and rounds are R.50in., R.25in. or R10mm, R5mm.

Figure P7-1

THICKNESS:
40mm
1.50in.

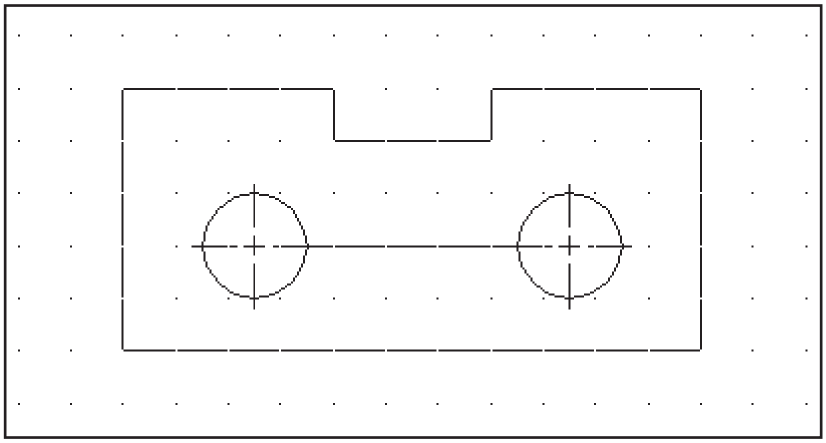


Figure P7-2

THICKNESS:
20mm
.75in.

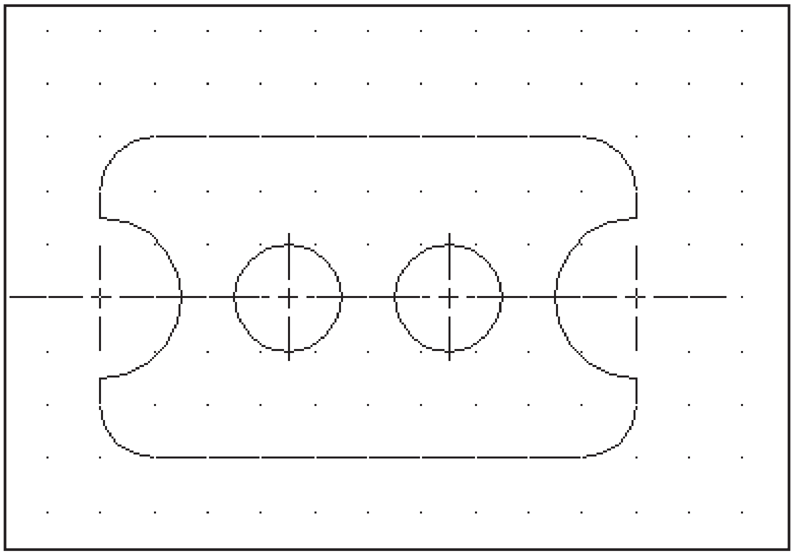
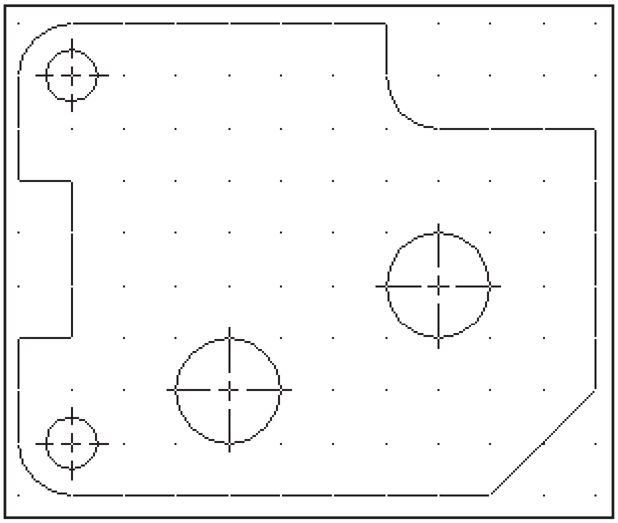
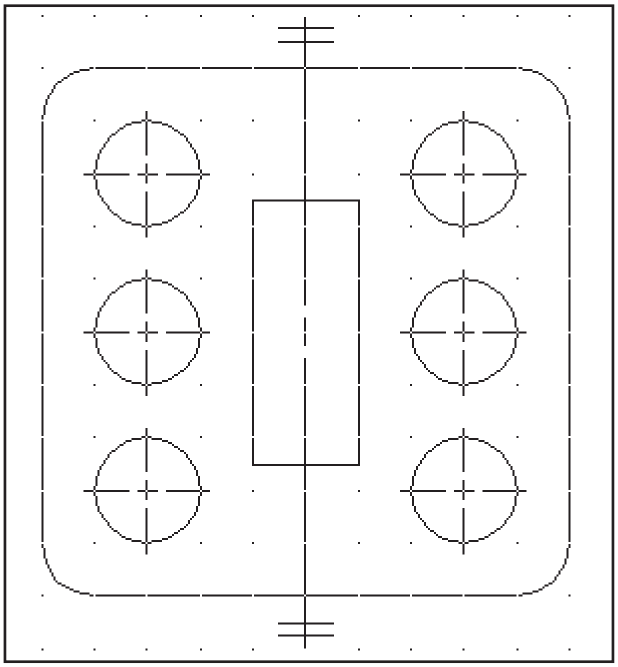


Figure P7-3



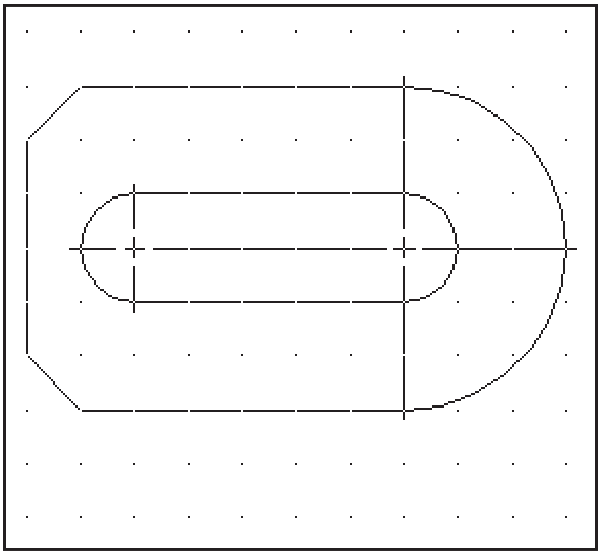
THICKNESS:
35mm
1.25in.

Figure P7-4



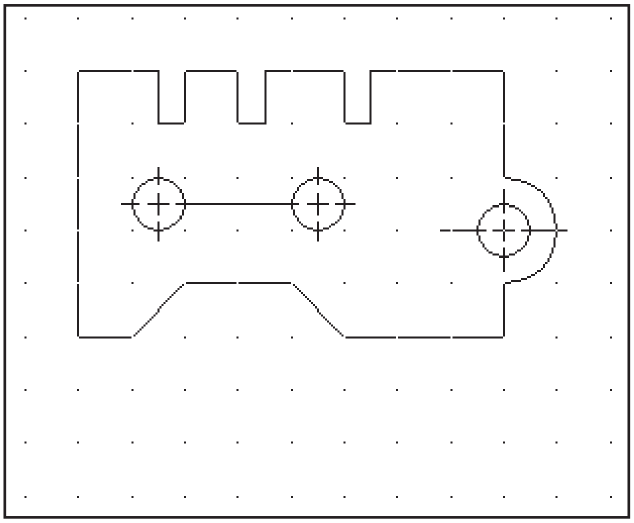
THICKNESS:
15mm
.50in.

Figure P7-5



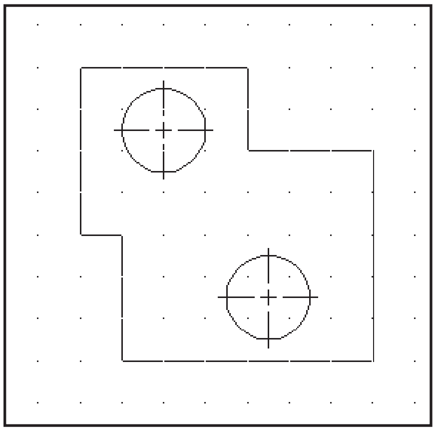
THICKNESS:
10mm
.50in.

Figure P7-6



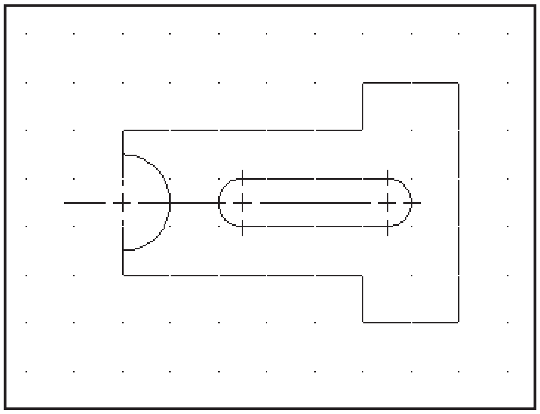
THICKNESS:
5mm
.25in.

Figure P7-7



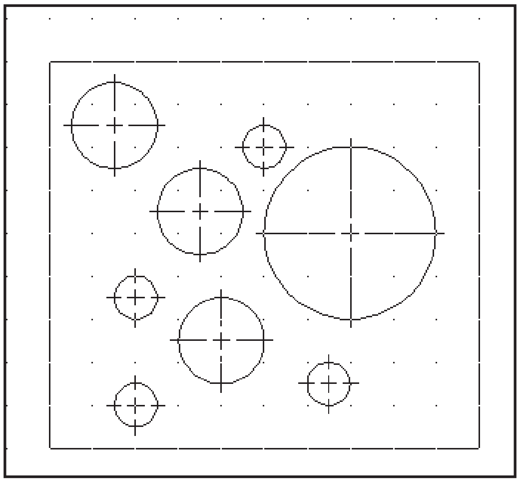
THICKNESS:
12mm
.50in.

Figure P7-8



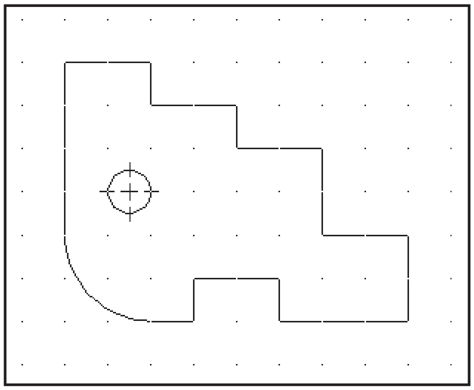
THICKNESS:
25mm
1.00in.

Figure P7-9



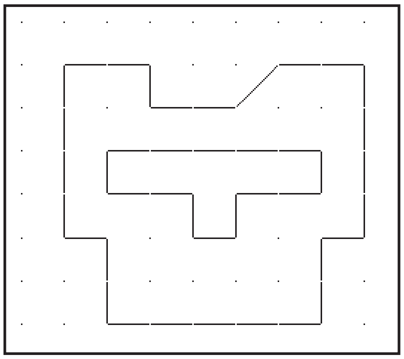
THICKNESS:
5mm
.25in.

Figure P7-10



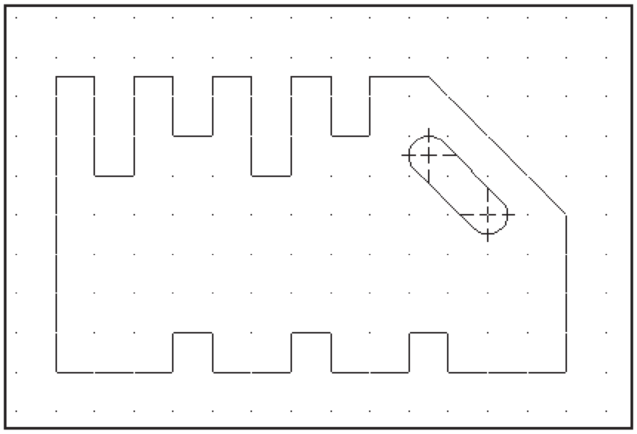
THICKNESS:
20mm
.75in.

Figure P7-11



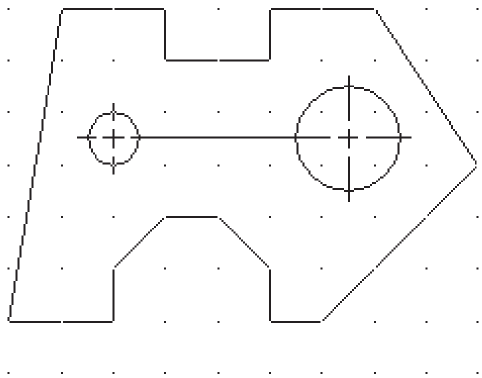
THICKNESS:
18mm
.625in.

Figure P7-12



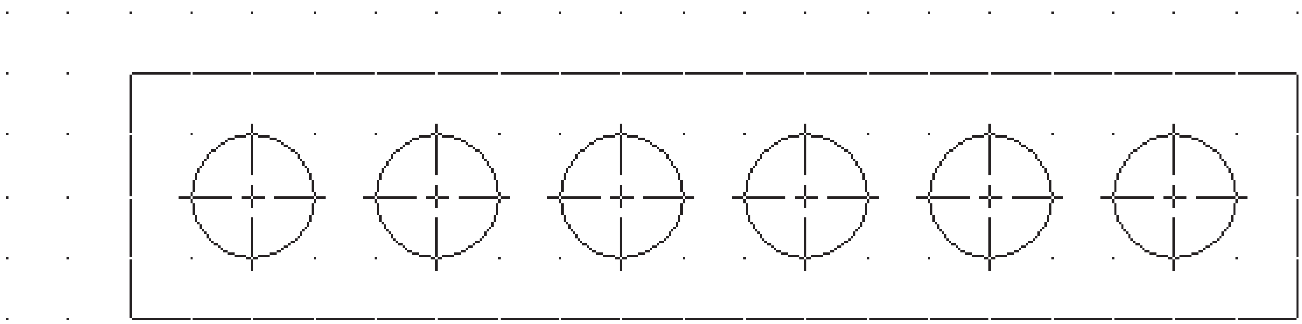
THICKNESS:
24mm
1.00in.

Figure P7-13



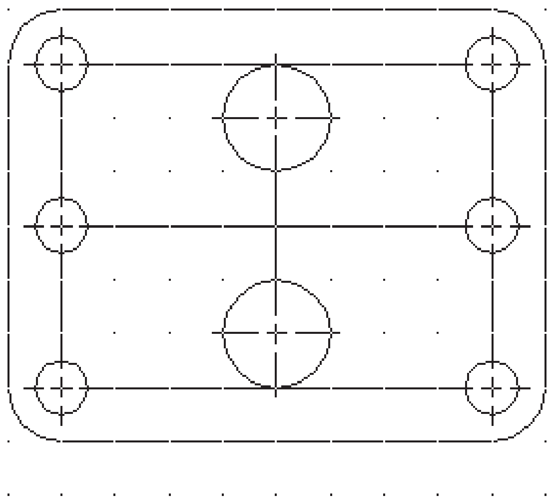
THICKNESS:
10mm
.25in.

Figure P7-14



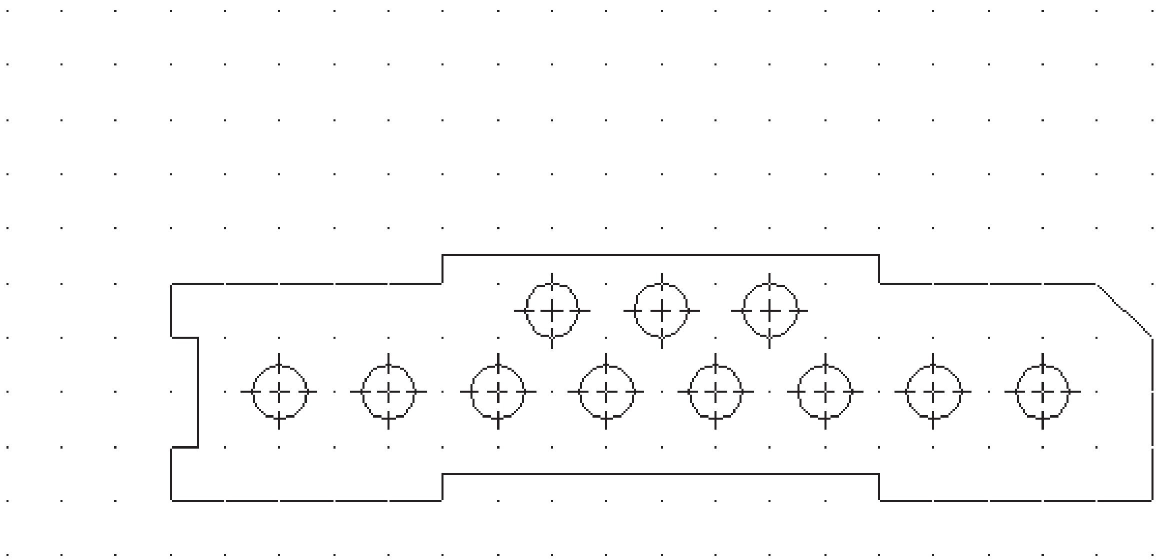
THICKNESS:
8mm
.25in.

Figure P7-15



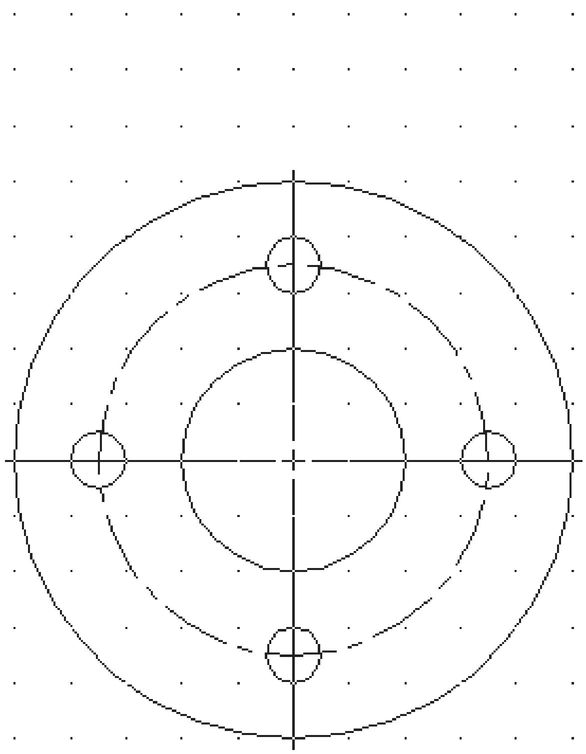
THICKNESS:
20mm
.75in.

Figure P7-16



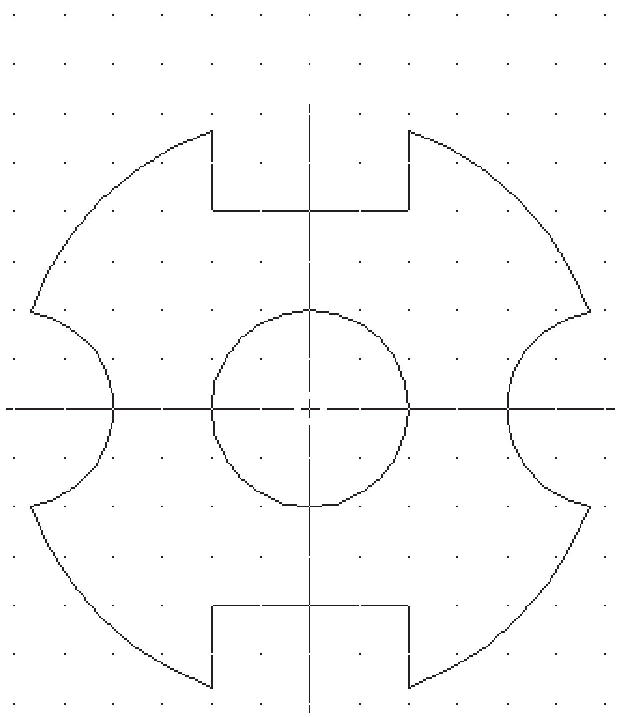
THICKNESS:
20mm
.75in.

Figure P7-17



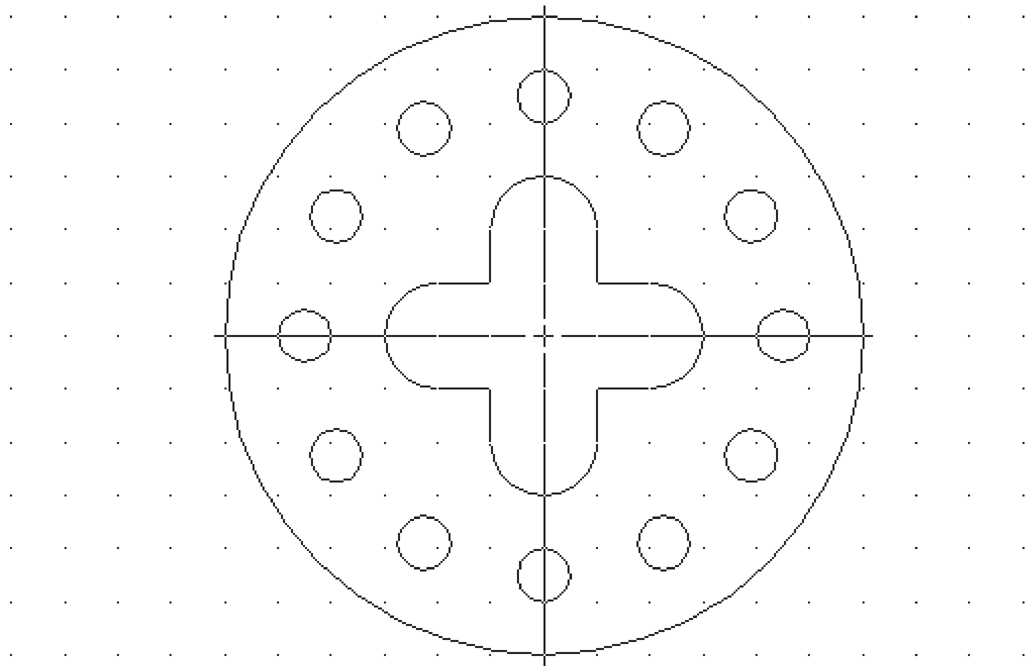
THICKNESS:
20mm
.75in.

Figure P7-18



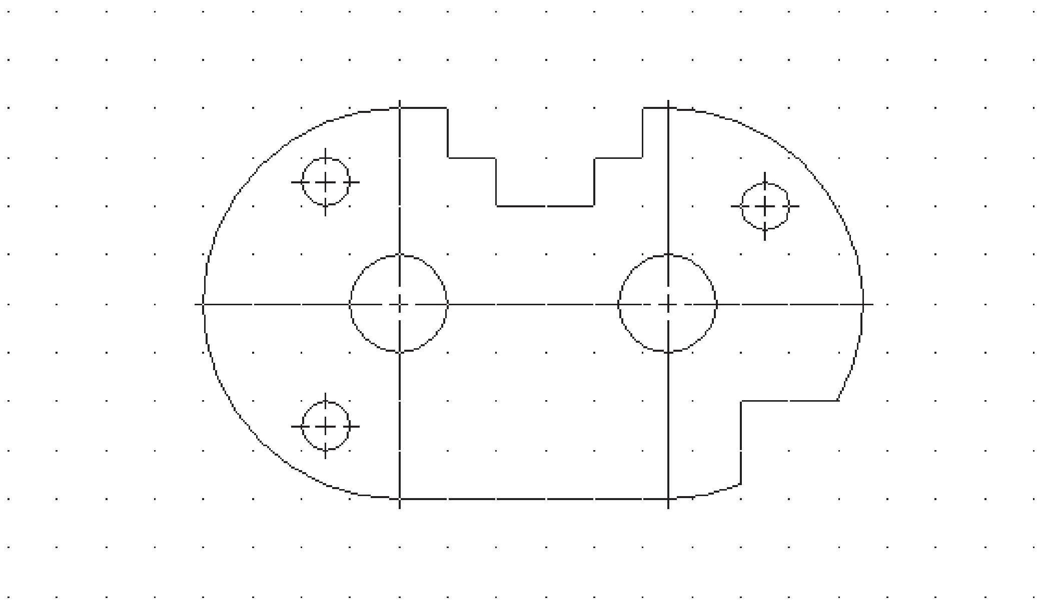
THICKNESS:
30mm
1.375in.

Figure P7-19



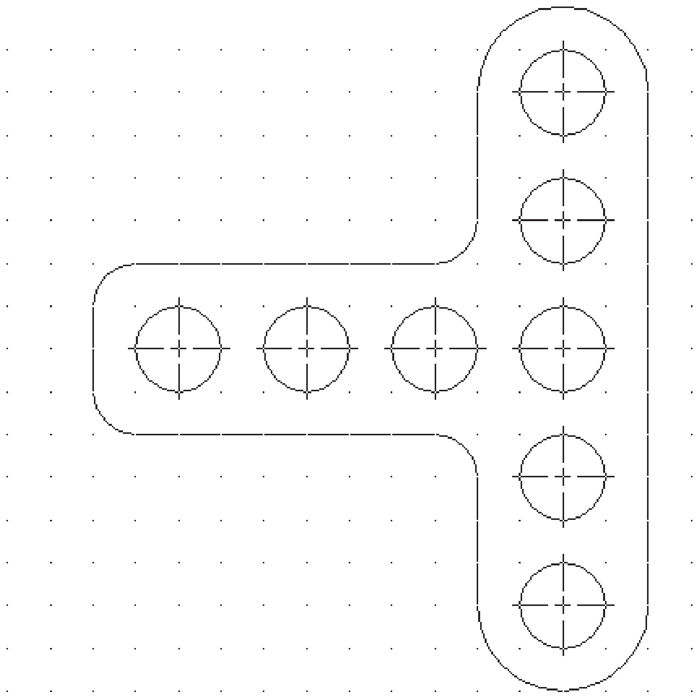
THICKNESS:
12mm
.30in.

Figure P7-20



THICKNESS:
5mm
.125in.

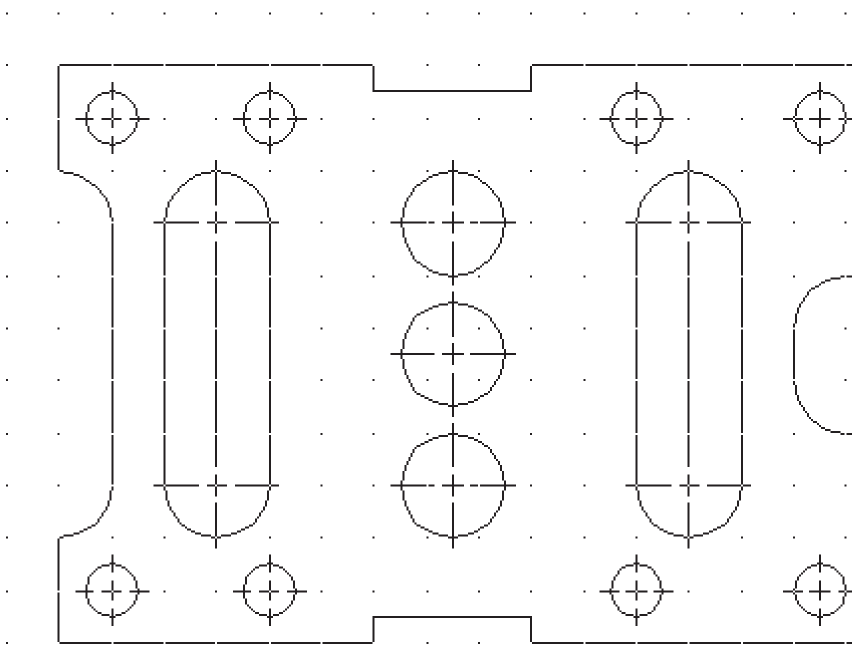
Figure P7-21



THICKNESS:
10mm
.25in.

Dimension using baseline dimensions

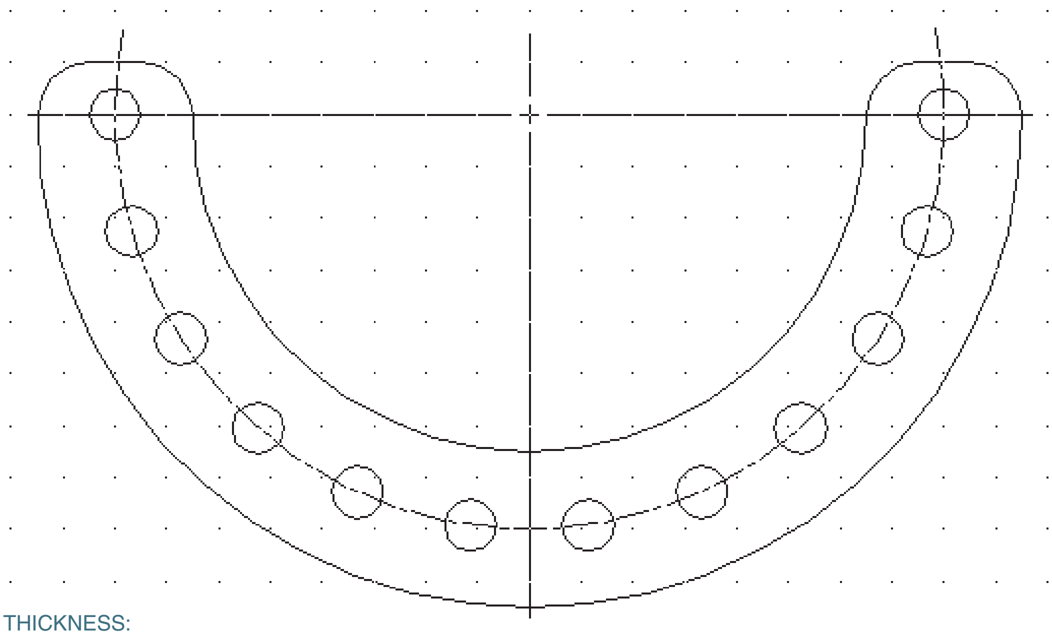
Figure P7-22



THICKNESS:
15mm
.50in.

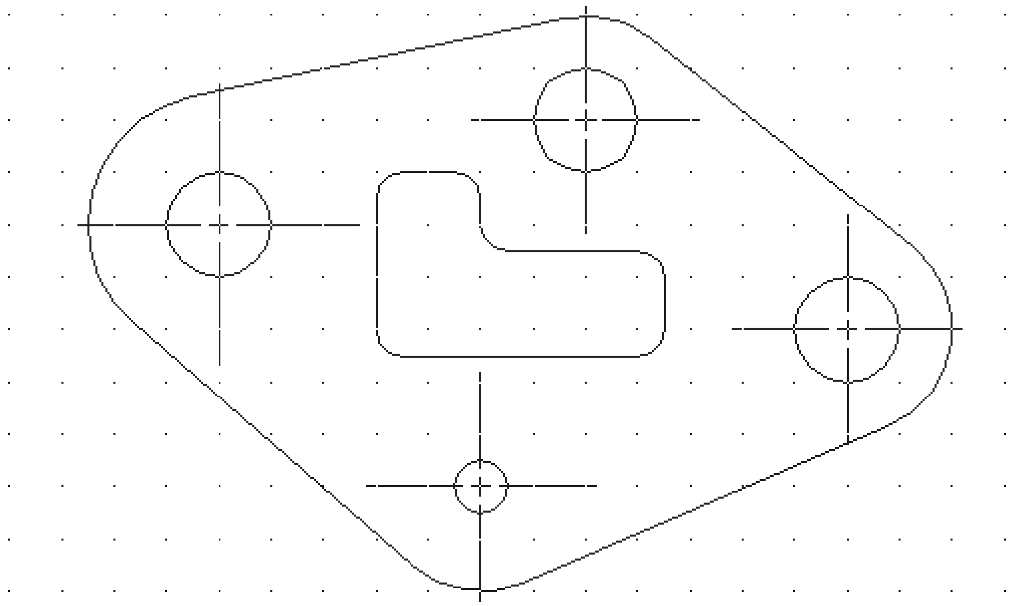
Dimension using
A. Baseline dimensions
B. Ordinate dimensions
C. Chain dimensions
D. Hole table

Figure P7-23



THICKNESS:
5mm
.19in.

Figure P7-24



THICKNESS:
15mm
.625in.

Project 7-2:

Use the **Part** template to draw models of the objects shown in Figures P7-25 through P7-42.

1. Create orthographic views of the objects. Dimension the orthographic views.
2. Create 3D models of the objects. Dimension the 3D models.

Figure P7-25
MILLIMETERS

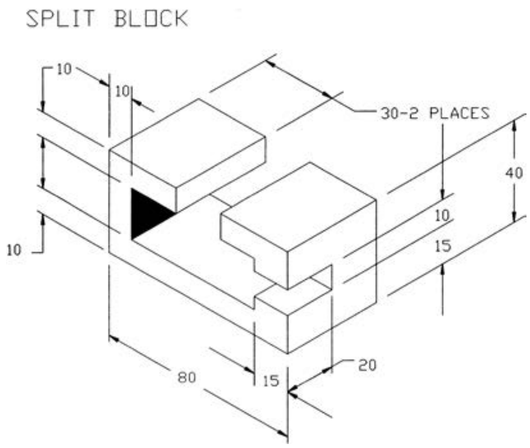


Figure P7-26
MILLIMETERS

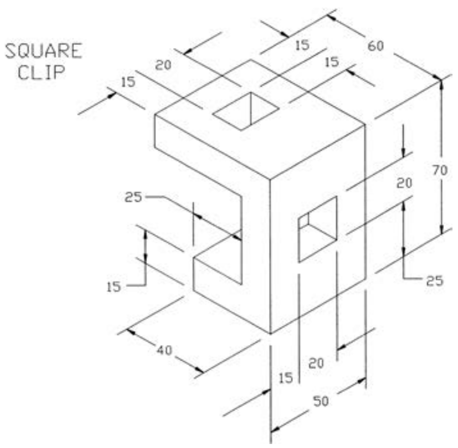


Figure P7-27
INCHES

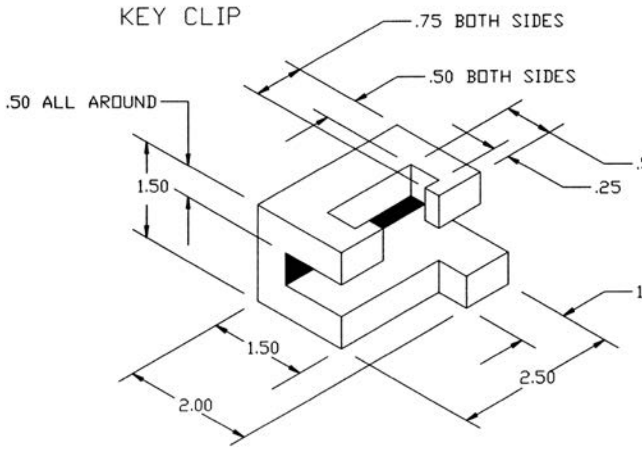


Figure P7-28
MILLIMETERS

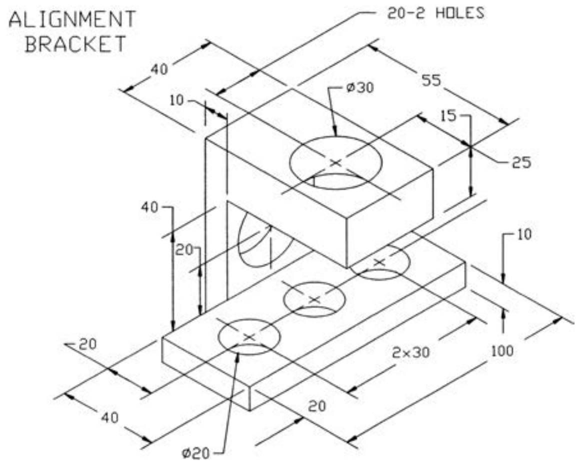


Figure P7-29
MILLIMETERS

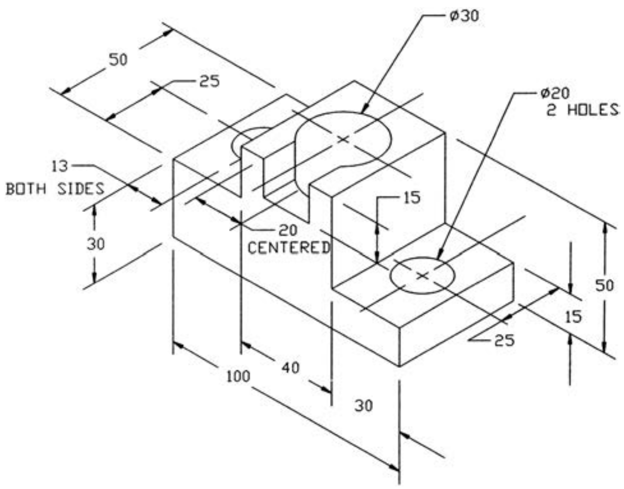


Figure P7-30
INCHES

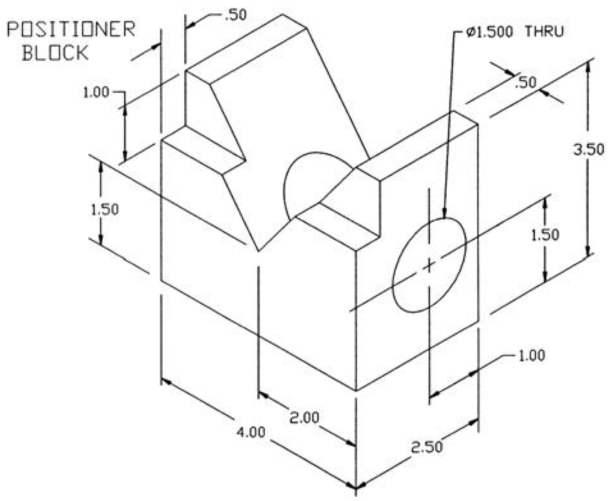


Figure P7-31
MILLIMETERS

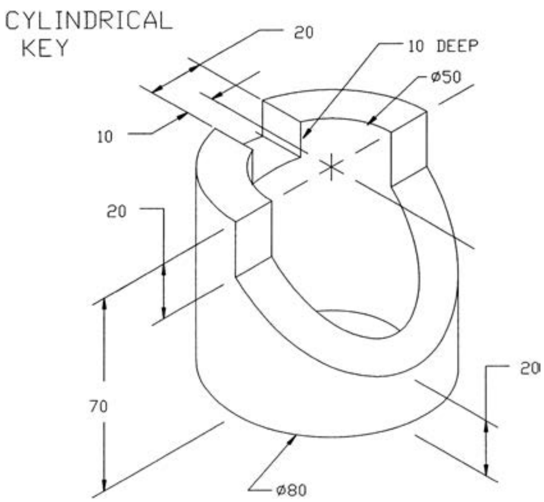


Figure P7-32
INCHES

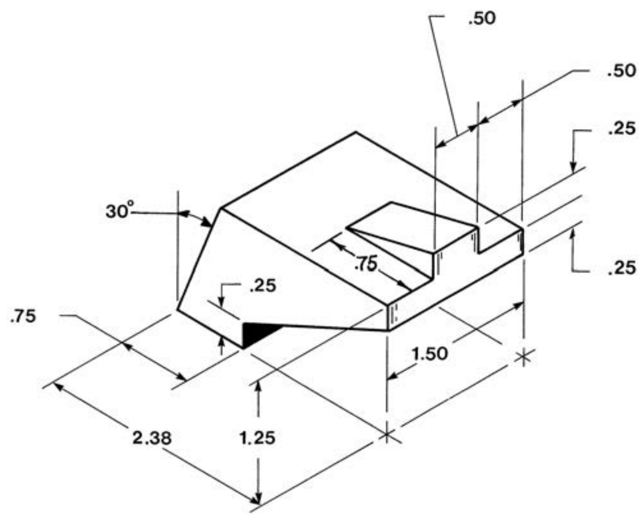


Figure P7-33
MILLIMETERS

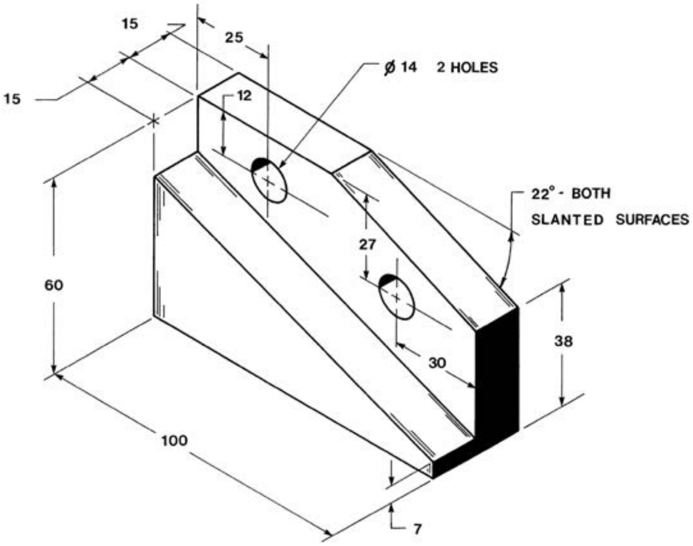


Figure P7-34
INCHES

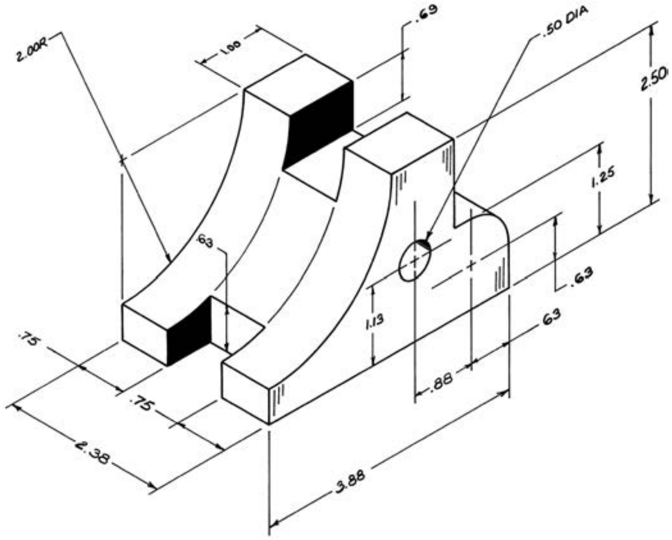


Figure P7-35
MILLIMETERS

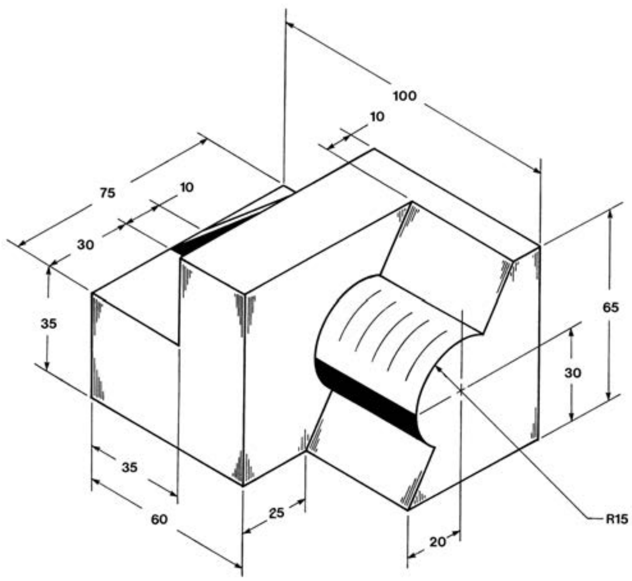


Figure P7-36
MILLIMETERS

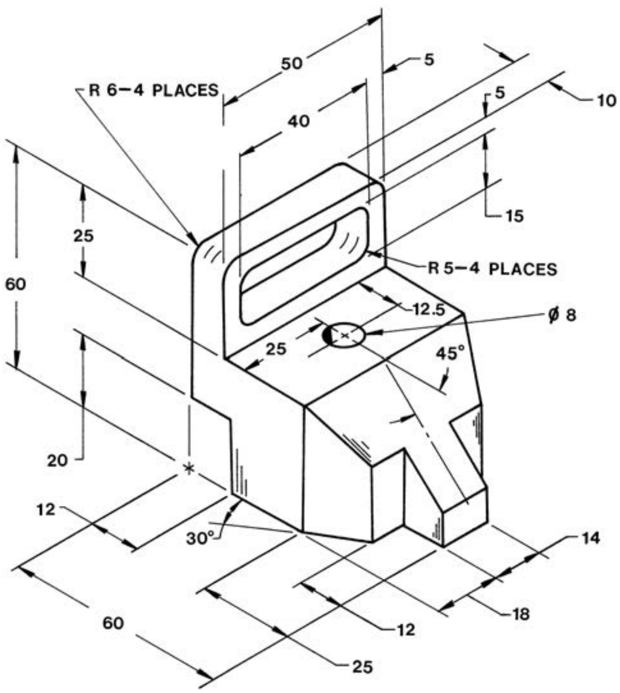


Figure P7-37
INCHES

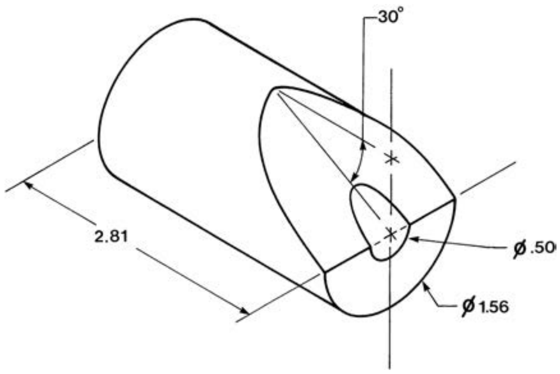


Figure P7-38
MILLIMETERS

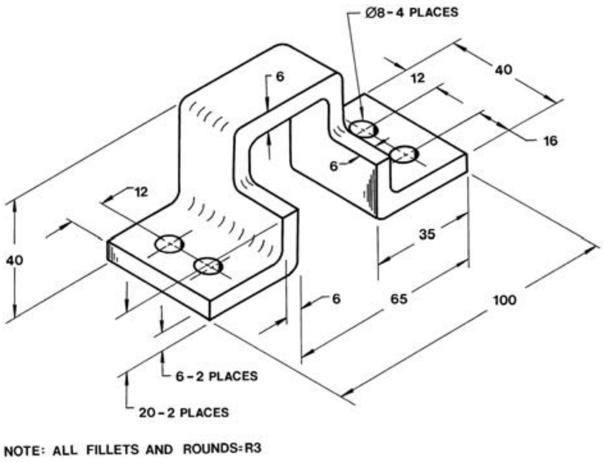


Figure P7-39
MILLIMETERS

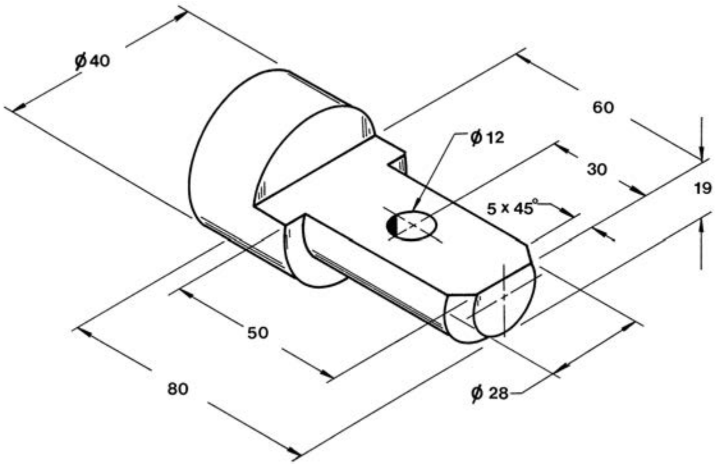


Figure P7-40
MILLIMETERS

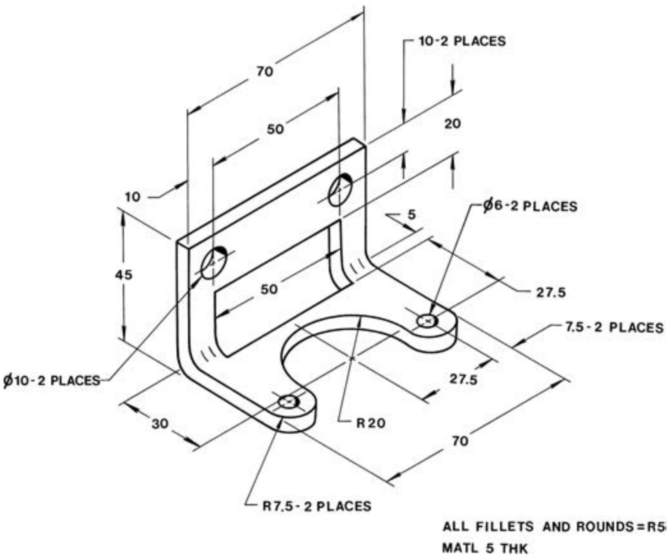


Figure P7-41
MILLIMETERS

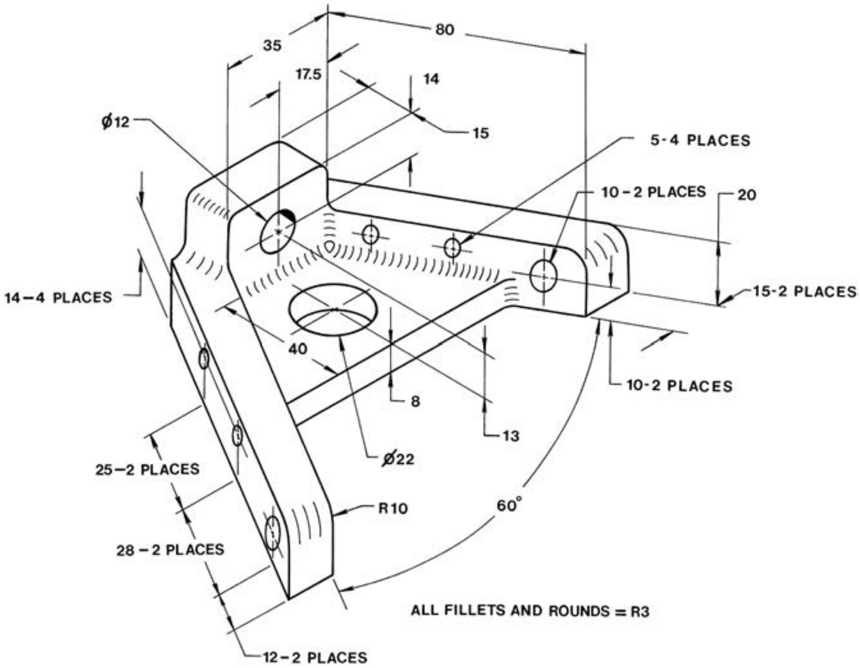
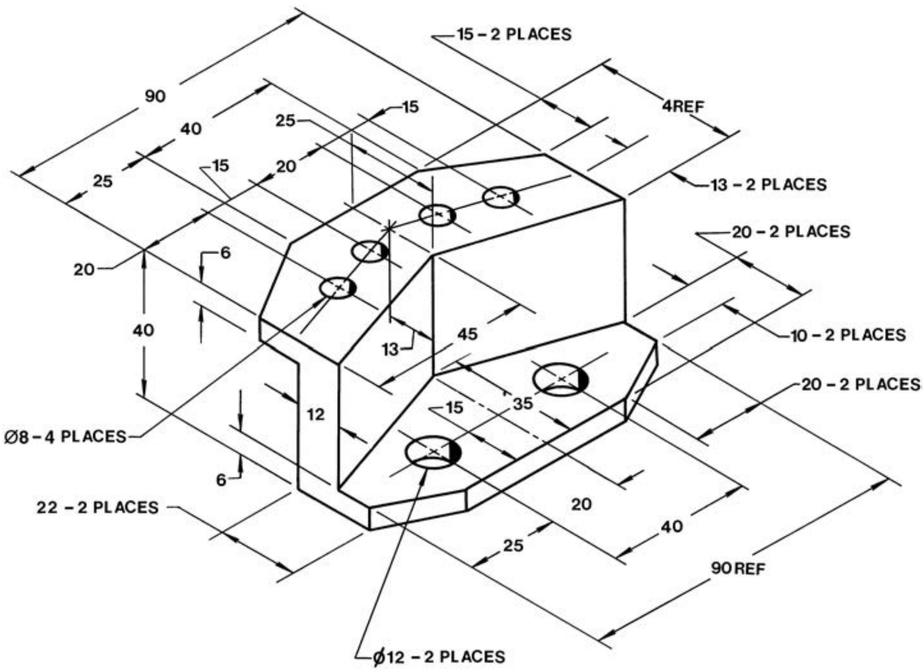


Figure P7-42
MILLIMETERS



Project 7-3:

1. Draw a 3D model from the given top orthographic and section views in Figure P7-43.
2. Draw a top orthographic view and a section view of the object and add dimensions.

Figure P7-43
MILLIMETERS

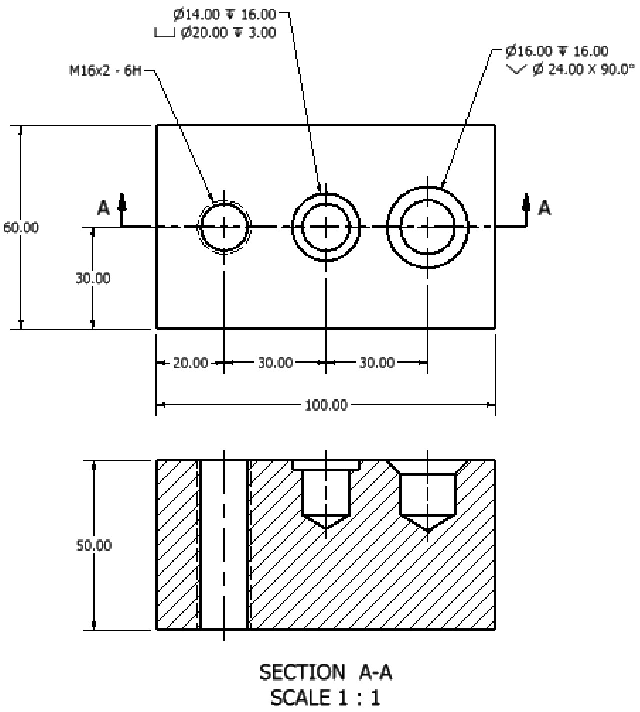
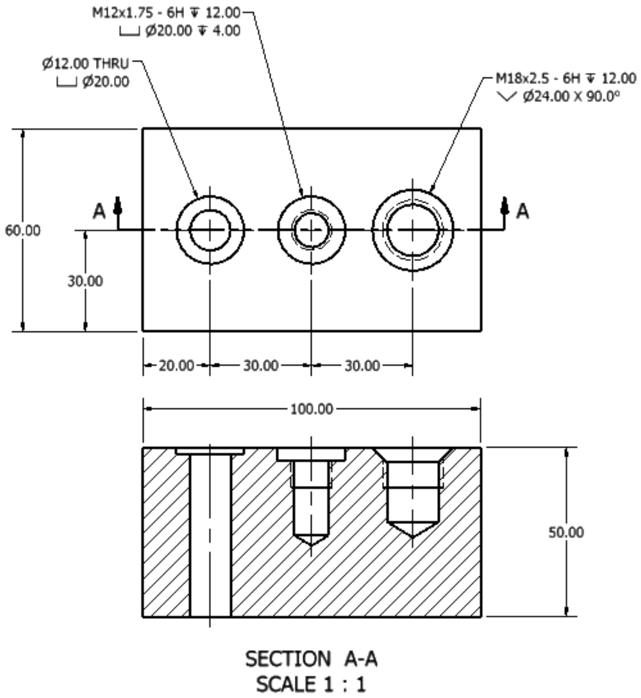


Figure P7-44
MILLIMETERS



Project 7-4:

1. Draw a 3D model from the given top orthographic and section views in Figure P7-44.
2. Draw a top orthographic view and a section view of the object and add dimensions.

Project 7-5:

1. Draw a 3D model from the given top orthographic and section views in Figure P7-45.
2. Draw a top orthographic view and a section view of the object and add dimensions.

Project 7-6:

1. Draw a 3D model from the given top orthographic and section views in Figure P7-46.
2. Draw a top orthographic view and a section view of the object and add dimensions.

Figure P7-45
INCHES

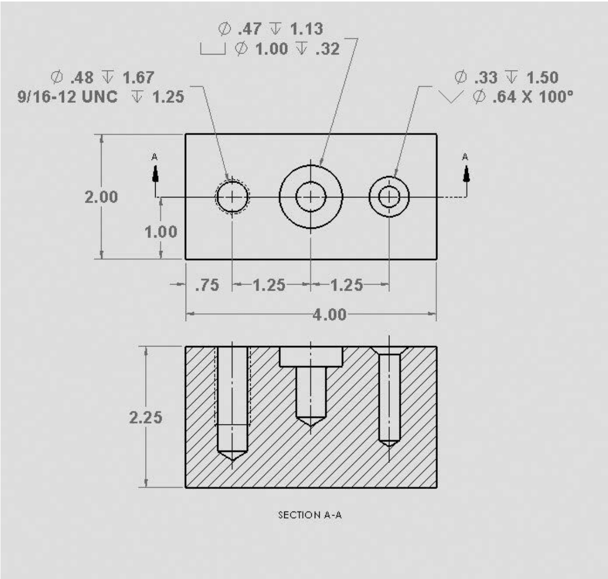
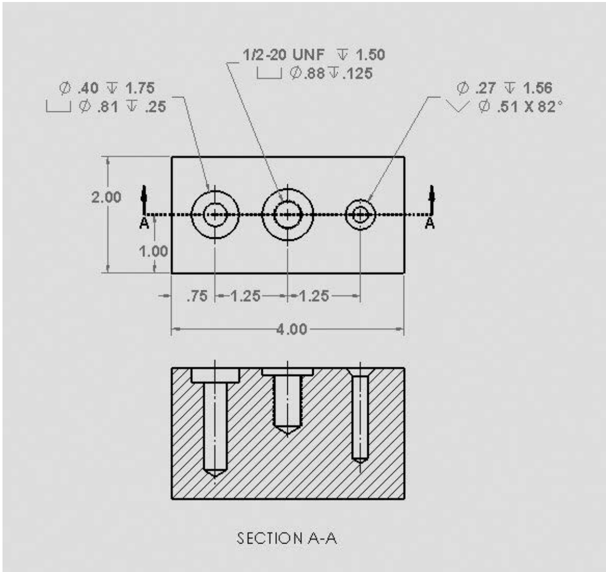


Figure P7-46
INCHES

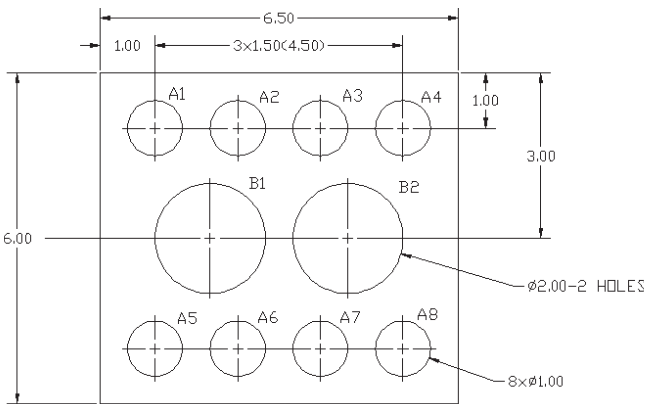


Project 7-7:

Redraw the given shapes in Figures P7-47 through P7-49 and dimension them using the following dimension styles.

1. Baseline
2. Ordinate
3. Hole Table

Figure P7-47
MILLIMETERS



MATL = 0.50

Figure P7-48
MILLIMETERS

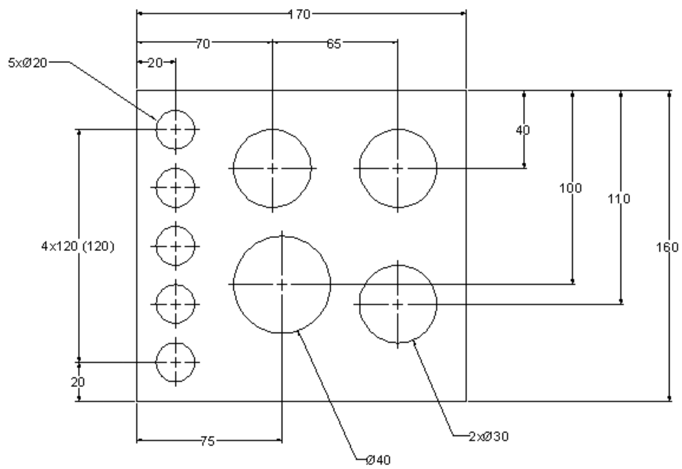
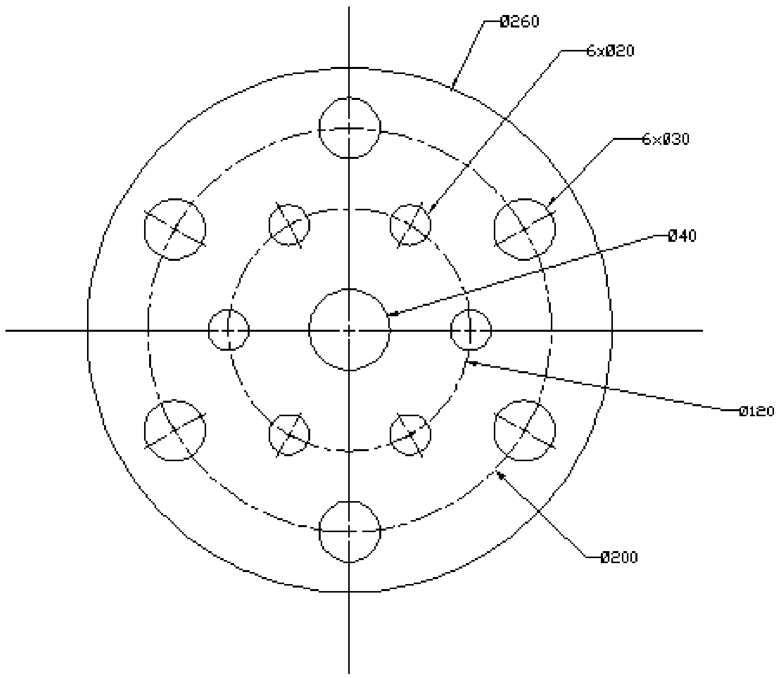


Figure P7-49
INCHES



Index

Q-R

S

U

V

W

X-Y-Z