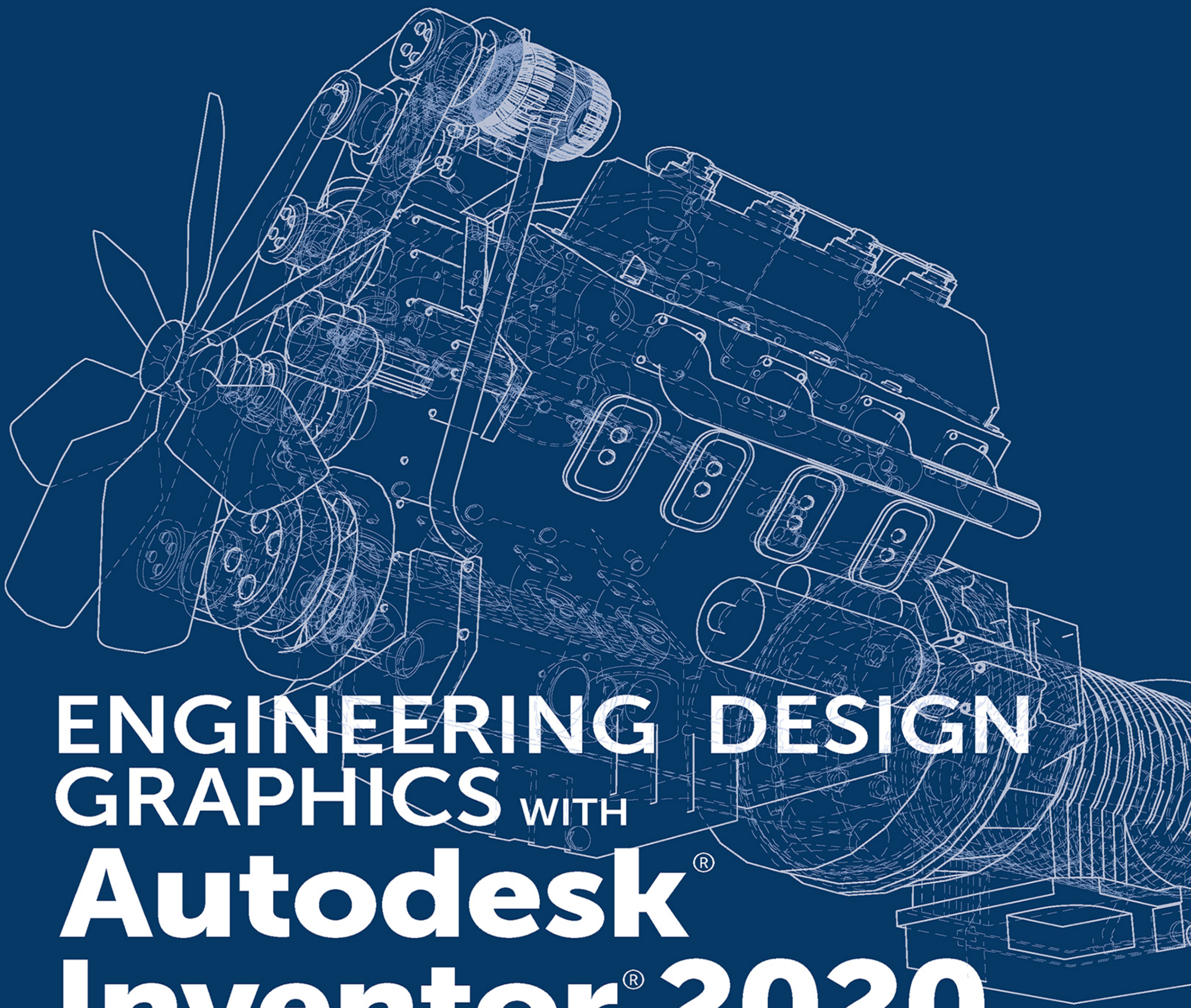

# ENGINEERING DESIGN GRAPHICS WITH Autodesk® Inventor® 2020

JAMES D. BETHUNE

# Engineering Design Graphics with Autodesk® Inventor® 2020

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# Engineering Design Graphics with Autodesk® Inventor® 2020

James D. Bethune

Pearson

**Engineering Design Graphics with Autodesk® Inventor® 2020**



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# Preface

This book introduces Autodesk® Inventor® 2020 and shows how to use Autodesk Inventor to create and document drawings and designs. The book puts heavy emphasis on engineering drawings and on drawing components used in engineering drawings such as springs, bearings, cams, and gears. It shows how to create drawings using many different formats such as .ipt, .iam, ipn, and .idw for both English and metric units. It explains how to create drawings using the tools located under the **Design** tab and how to extract parts from the **Content Center**.

All topics are presented using a step-by-step format so that the reader can work directly from the text to the screen. There are many easy-to-understand labeled illustrations. The book contains many sample problems that demonstrate the subjects being discussed. Each chapter contains a variety of projects that serve to reinforce the material just presented and allow the reader to practice the techniques described.

**Chapters 1 and 2** present 2D sketching tools and the **Extrude** tool. These chapters serve as an introduction to the program. There are 38 Chapter Projects to help students apply the material presented.

**Chapter 3** demonstrates the tools needed to create 3D models, including **Shell, Hole, Rib, Split, Loft, Sweep,** and **Coil.** Work points, work axes, and work planes are explained and demonstrated.

**Chapter 4** shows how to create orthographic views from 3D models. The creation of isometric views, section views, and auxiliary views is covered. In addition, a comparison between first- and third-angle projection is presented using both ANSI and ISO conventions.

**Chapter 5** shows how to create assembly drawings using both the bottom-up and the top-down processes. The chapter includes presentation drawings and exploded isometric drawings with title blocks, parts lists, revision blocks, and tolerance blocks. There is an extensive step-by-step example that shows how to create an animated assembly—that is, a drawing that moves on the screen.

**Chapter 6** covers threads and fasteners. Drawing conventions and callouts are defined for both inch and metric threads. The chapter shows how to calculate thread lengths and how to choose the appropriate fastener from Inventor's **Content Center**. The **Content Center** also includes an extensive listing of nuts, setscrews, washers, and rivets.

**Chapter 7** shows how to apply dimensions to drawings. Both ANSI and ISO standards are demonstrated, but the emphasis is on ANSI standards. Different styles of dimensioning, including ordinate and baseline, and using Inventor's **Hole Table** are presented. Applying dimensions to a drawing is considered an important skill, so many examples and sample problems are included.

**Chapter 8** is an extensive discussion of tolerancing, including geometric tolerances. The chapter first shows how to use Inventor to apply tolerances to a drawing. The chapter then shows how to calculate tolerances in various design situations. Positional tolerances for both linear and geometric applications are included. The chapter introduces the **Limits/Fits Calculator** located on the **Power Transmission** panel under the **Design** tab.

**Chapter 9** shows how to draw springs using the **Standard.ipt** format and the **Coil** tool. It also shows how to draw springs using the tools on the

**Spring** panel under the **Design** tab. Compression, extension, torsion, and Belleville springs are included.

**Chapter 10** shows how to draw shafts using the **Shaft** tool under the Design tab. Chamfers, retaining rings, retaining ring grooves, keys and keyways, splines, pins, O-rings, and O-ring grooves are covered. The chapter contains many exercise problems.

**Chapter 11** shows how to match bearings to specific shafts using the **Content Center**. Plain, ball, and thrust bearings are presented. An explanation of tolerances between a shaft and bearing bore and between the bearing's outside diameter and the assembly housing is given. Both ANSI and ISO standards are presented.

**Chapter 12** emphasizes how to draw gears and how to mount them into assembly drawings. Spur, bevel, and worm gears are introduced. The chapter shows how to create gear hubs with setscrews, and keyways with keys, and how to draw assembly drawings that include gears. There are two new extensive assembly exercise problems.

**Chapter 13** shows how to draw basic sheet metal parts, including features such as tabs, reliefs, flanges, cuts, holes, and hole patterns.

**Chapter 14** shows how to create and draw weldments. Only fillet and groove welds are covered.

**Chapter 15** shows how to design and draw cams. Displacement diagrams and different types of followers are discussed.

**Chapter 16** is available online and includes two large project-type problems. They can be used as team projects to help students learn to work together to share and compile files, or they can be used as end-of-the-semester individual projects. This chapter can be found on the web as a supplement to the Instructor's Manual at http://pearsonhighered.com/irc. Instructors may distribute to students.

## Acknowledgments

I would like to thank the following reviewers for their invaluable input: Rebecca Rosenbauer, Lafayette College; Antigone Sharris, Triton College; Nancy E. Study, Virginia State University; and Marsha Walton, Finger Lakes Community College.

Thanks to my family: David, Maria, Randy, Sandra, Hannah, Wil, Madison, Jack, Luke, Sam, and Ben.

A special thanks to Cheryl.

*James D. Bethune*

## Style Conventions in *Engineering Design Graphics with Autodesk® Inventor® 2020*

| Text Element | Example |
|---|---|
| **Key terms—**Bold and italic on first mention in the body of the text. Brief glossary definition in margin following first mention. | Create a ***work axis*** by clicking on the edge of the block. |
| **Inventor tools**—Bold and follow Inventor capitalization convention. | Click on the **Line** tool. |
| **Toolbar names, menu items, and dialog box names**—Bold and follow capitalization convention in Inventor tab, panel, or pull-down menu (generally first letter capitalized). | The **Design** tab<br>The **Modify** panel<br>The **2D Chamfer** dialog box<br>The **File** pull-down menu |
| **Dialog box controls/buttons/input items**—Bold and follow capitalization convention of the name of the item or the name shown in the Inventor tooltip. | Choose the **Metric** tab in the **New File** dialog box.<br>Click on the **Flush** button on the **Place Constraint** dialog box.<br>On the **Assembly** tab, set the **Offset** to **0.000 mm.**<br>**ONG 300/320 sat 76** |

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# Brief Contents

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# Contents

# 4 chapter four
# Orthographic Views

## CHAPTER OBJECTIVES

- Learn how to draw orthographic views
- Learn ANSI standards and conventions
- Learn about third-angle projection
- Learn how to draw section and auxiliary views

## Introduction

**orthographic views:** Two-dimensional views used to define a three-dimensional model. (Usually more than one view is needed to define a 3D model.)

Orthographic views may be created directly from 3D Inventor models. ***Orthographic views*** are two-dimensional views used to define a three-dimensional model. Unless the model is of uniform thickness, more than one orthographic view is necessary to define the model's shape. Standard practice calls for three orthographic views: a front, a top, and a right-side view, although more or fewer views may be used as needed.

Modern machines can work directly from the information generated when a solid 3D model is created, so the need for orthographic views—blueprints—is not as critical as it once was; however, there are still many drawings in existence that are used for production and reference. The ability to create and read orthographic views remains an important engineering skill.

This chapter presents orthographic views using third-angle projection in accordance with American National Standards Institute (ANSI) standards. International Organization for Standardization (ISO) first-angle projections are also presented.

## Fundamentals of Orthographic Views

Figure 4-1 shows an object with its front, top, and right-side orthographic views projected from the object. The views are two-dimensional, so they show no depth. Note that in the projected right plane, there are three rectangles. There is no way to determine which of the three is closest and which is farthest away if only the right-side view is considered. All views must be studied to analyze the shape of the object.

Figure 4-2 shows three orthographic views of a book. After the views are projected they are positioned as shown. The positioning of views relative to one another is critical. The views must be aligned and positioned as shown.

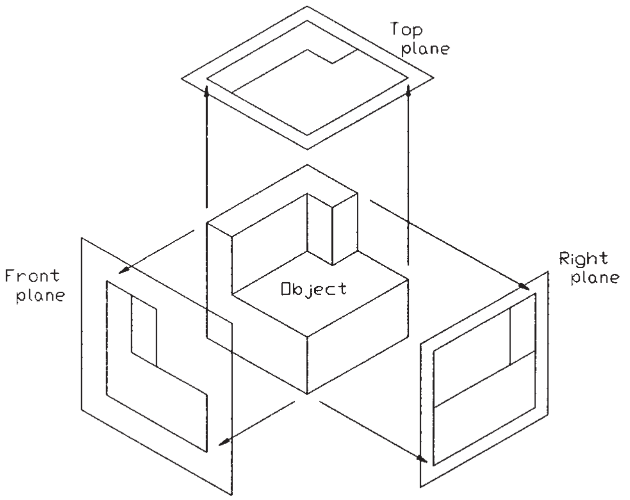



Figure 4-1

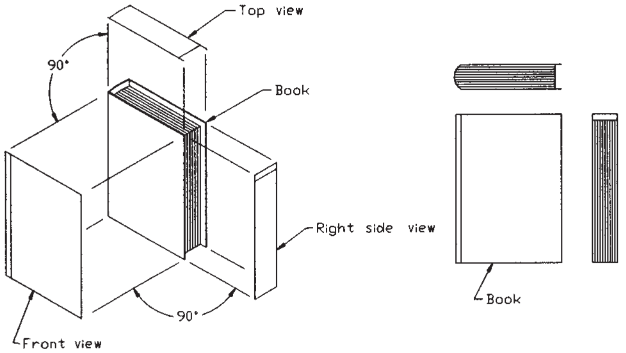



Figure 4-2

## Normal Surfaces

**normal surfaces:** Surfaces that are 90° to each other.

***Normal surfaces*** are surfaces that are at 90° to each other. Figures 4-3, 4-4, and 4-5 show objects that include only normal surfaces and their orthographic views.

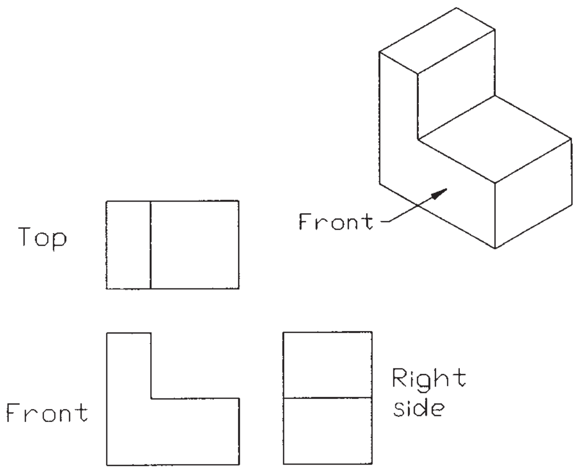



Figure 4-3

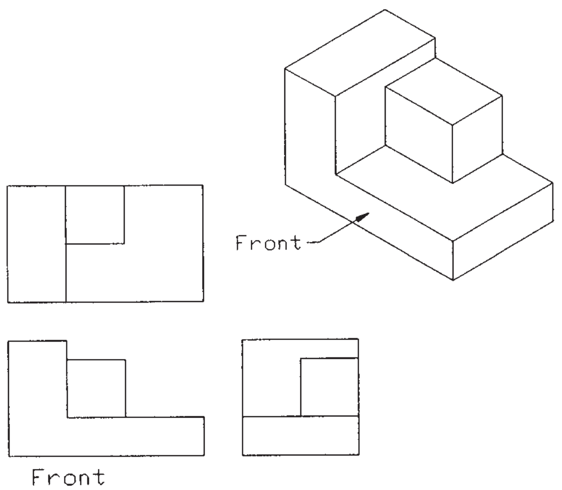
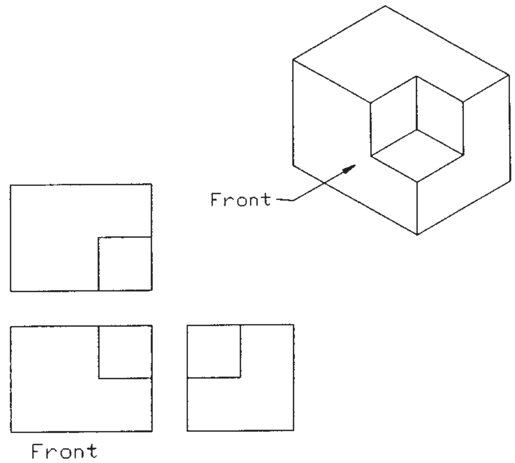



Figure 4-4

**Figure 4-5**




## Hidden Lines

Hidden lines are used to show surfaces that are not directly visible. All surfaces must be shown in all views. If an edge or surface is blocked from view by another feature, it is drawn using a hidden line. Figures 4-6 and 4-7 show objects that require hidden lines in their orthographic views.

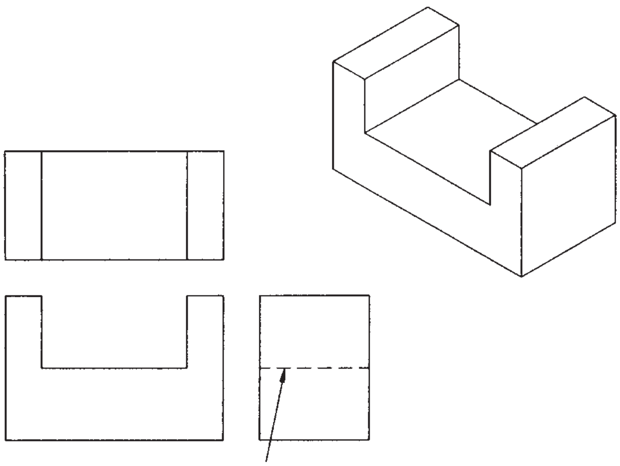


**Figure 4-6**

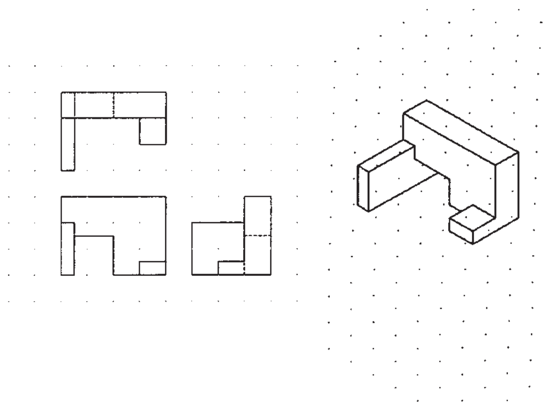
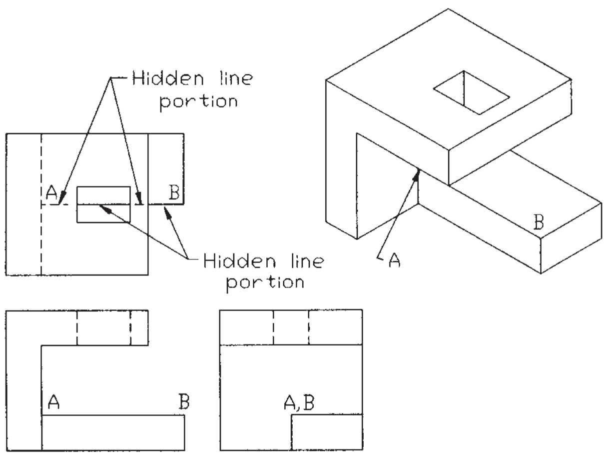


**Figure 4-7**

Figure 4-8 shows an object that contains an edge line, *A-B*. In the top view, line *A-B* is partially hidden and partially visible. The hidden portion of the line is drawn using a hidden-line pattern, and the visible portion of the line is drawn using a solid line.

**Figure 4-8**



Figures 4-9 and 4-10 show objects that require hidden lines in their orthographic views.

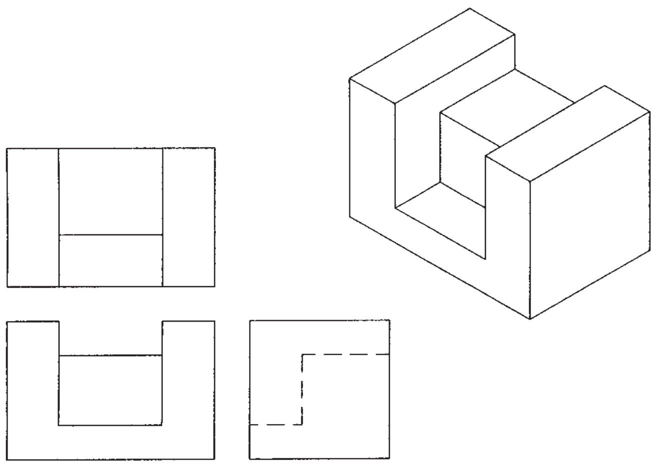


Figure 4-9

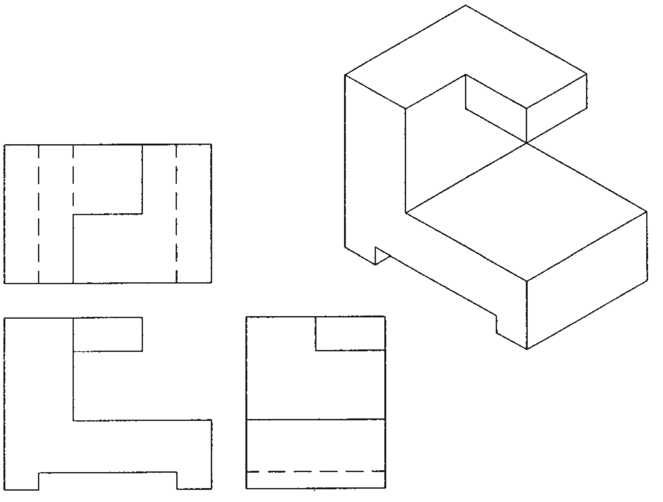


Figure 4-10

## Precedence of Lines

It is not unusual for one type of line to be drawn over another type of line. Figure 4-11 shows two examples of overlap by different types of lines. Lines are shown on the views in a prescribed order of precedence. A solid

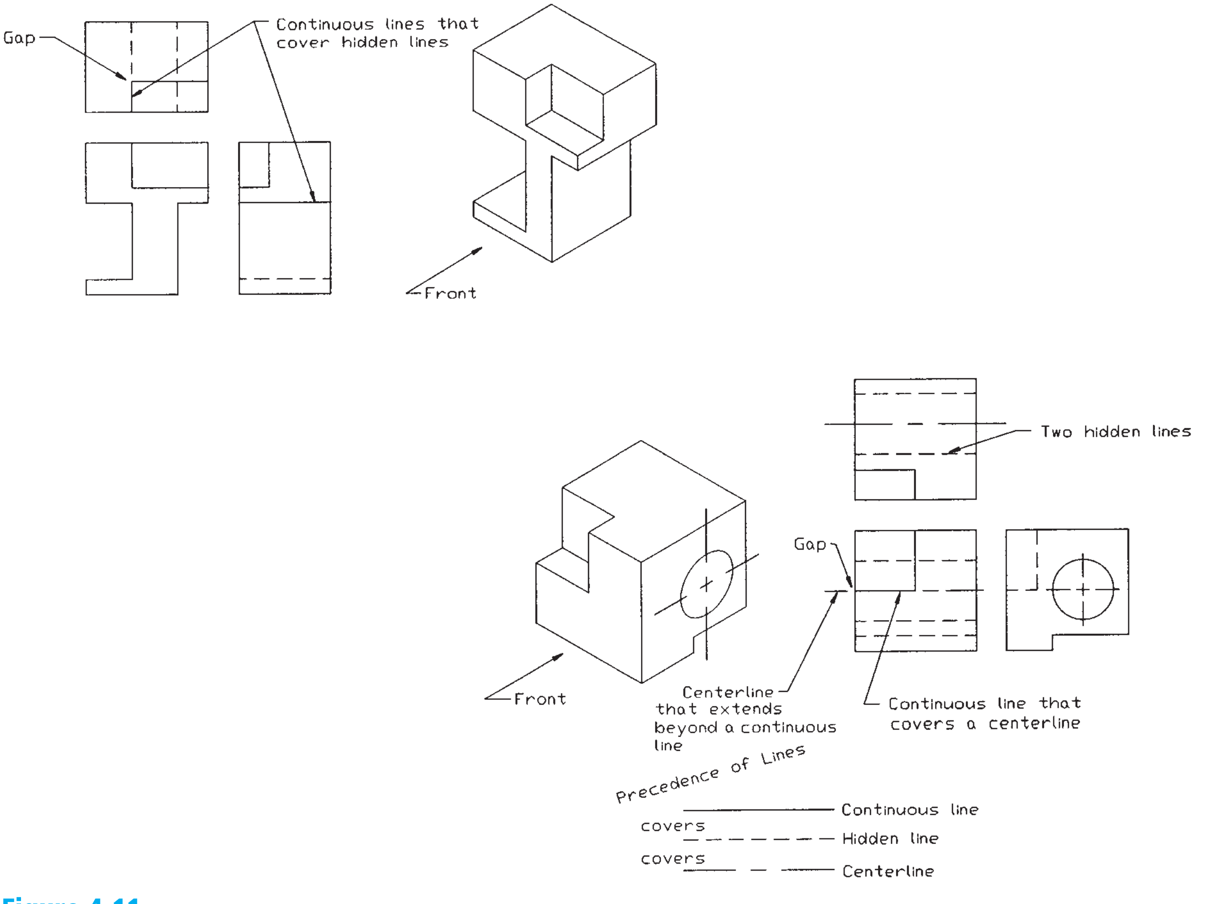



Figure 4-11

line (object or continuous) takes precedence over a hidden line, and a hidden line takes precedence over a centerline.

## Slanted Surfaces

**slanted surfaces:** Surfaces that are at an angle to each other.

***Slanted surfaces*** are surfaces drawn at an angle to each other. Figure 4-12 shows an object that contains two slanted surfaces. Surface *ABCD* appears as a rectangle in both the top and front views. Neither rectangle represents the true shape of the surface. Each is smaller than the actual surface. Also, none of the views shows enough of the object to enable the viewer to accurately define the shape of the object. The views must be used together for a correct understanding of the object's shape.

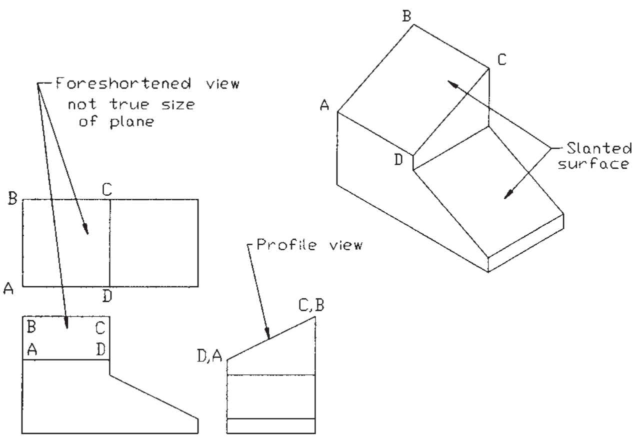



Figure 4-12

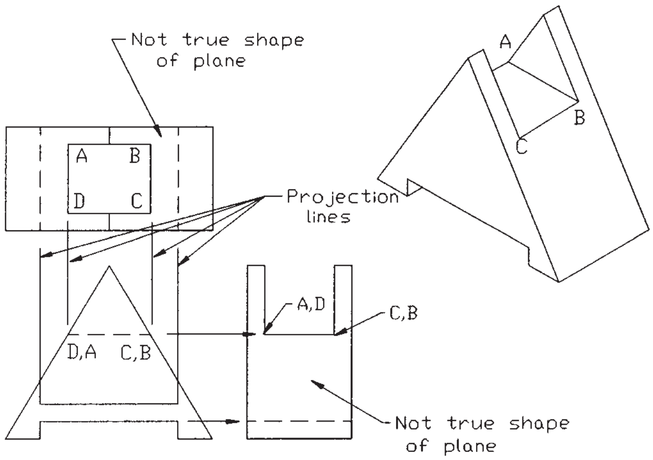
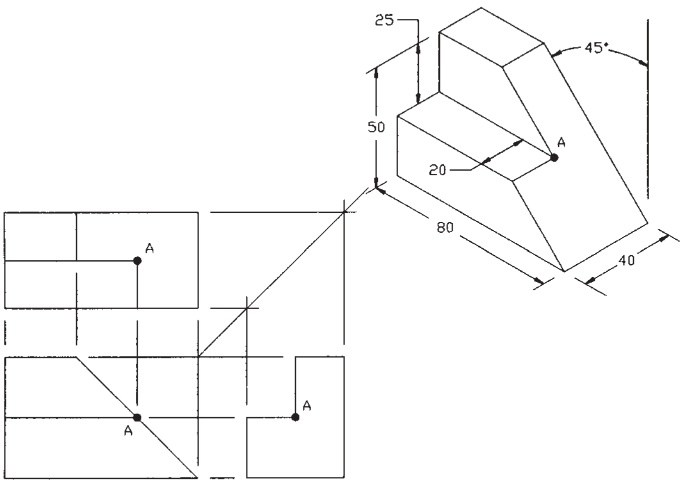



Figure 4-13

Figures 4-13 and 4-14 show objects that include slanted surfaces. Projection lines have been included to emphasize the importance of correct view location. Information is projected between the front and top views using vertical lines and between the front and side views using horizontal lines.

Figure 4-14

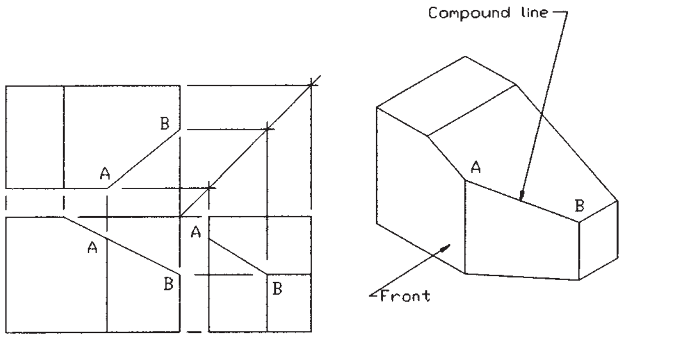



## Compound Lines

**compound line:** A line that is neither perpendicular nor parallel to the *X*, *Y*, or *Z* axis.

A ***compound line*** is formed when two slanted surfaces intersect. Figure 4-15 shows an object that includes a compound line.

Figure 4-15




## Oblique Surfaces

**oblique surface:** A surface that is slanted in two different directions.

An ***oblique surface*** is a surface that is slanted in two different directions. Figures 4-16 and 4-17 show objects that include oblique surfaces.

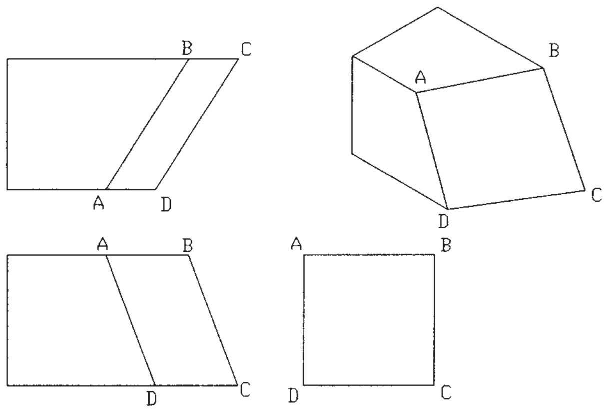



Figure 4-16

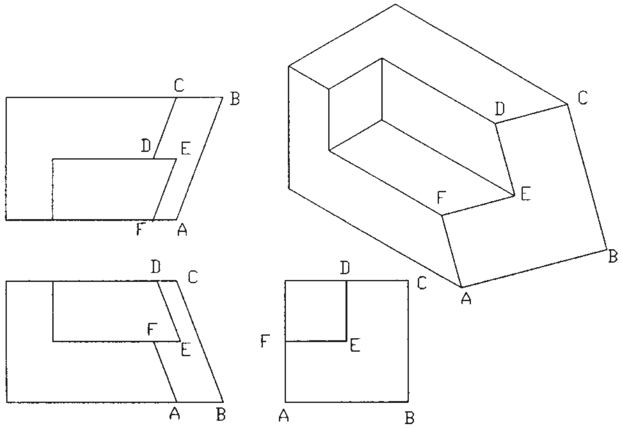
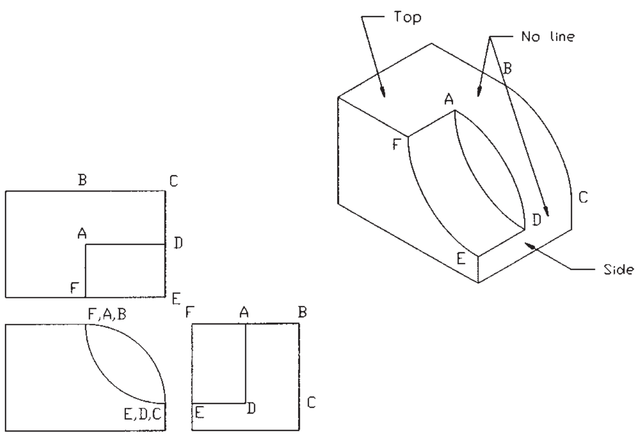



Figure 4-17

## Rounded Surfaces

Figure 4-18 shows an object with two rounded surfaces. Note that as with slanted surfaces, an individual view is insufficient to define the shape of a surface. More than one view is needed to define the surface's shape accurately.

Figure 4-18



Convention calls for a smooth transition between rounded and flat surfaces; that is, no lines are drawn to indicate the tangency. Inventor includes a line to indicate tangencies between surfaces in the isometric drawings created using the multiview options but does not include them in the orthographic views. Tangency lines are also not included when models are rendered.

Figure 4-19 shows the drawing conventions for including lines for rounded surfaces. If a surface includes no vertical portions or no tangency, no line is included.

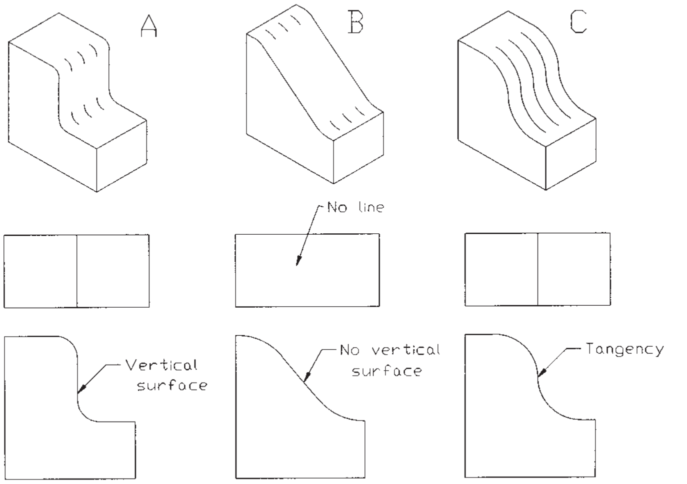



Figure 4-19

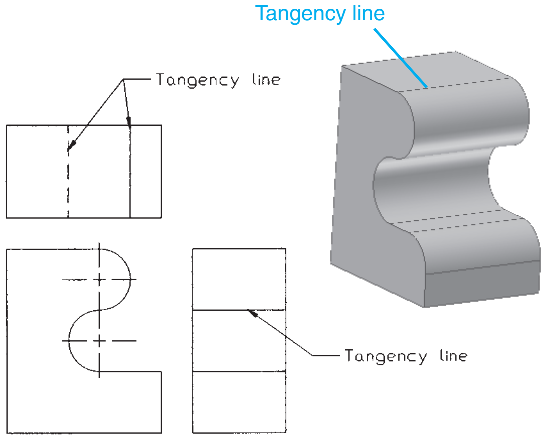



Figure 4-20

Figure 4-20 shows an object that includes two tangencies. Each is represented by a line. Note in Figure 4-20 that Inventor will add tangent lines to the 3D model. These lines will not appear in the orthographic views.

Figure 4-21 shows two objects with similar configurations; however, the boxlike portion of the lower object blends into the rounded portion exactly on its widest point, so no line is required.

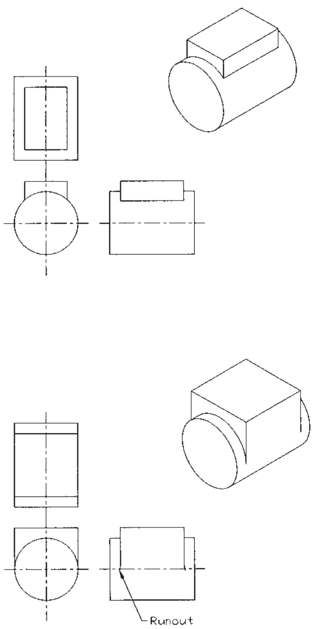
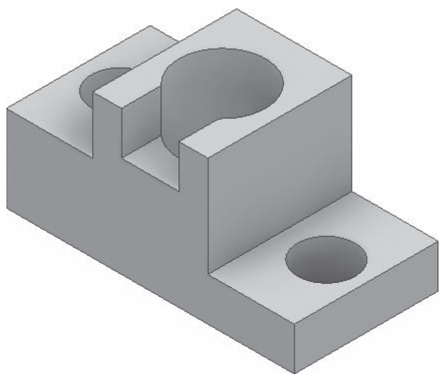



Figure 4-21

## Orthographic Views with Inventor

Inventor will create orthographic views directly from models. Figure 4-22 shows a completed three-dimensional model. See Figure P4-7 for the model's dimensions. It was created using an existing file, **BLOCK, 3HOLE.** It will be used throughout this chapter to demonstrate orthographic presentation views.

Figure 4-22

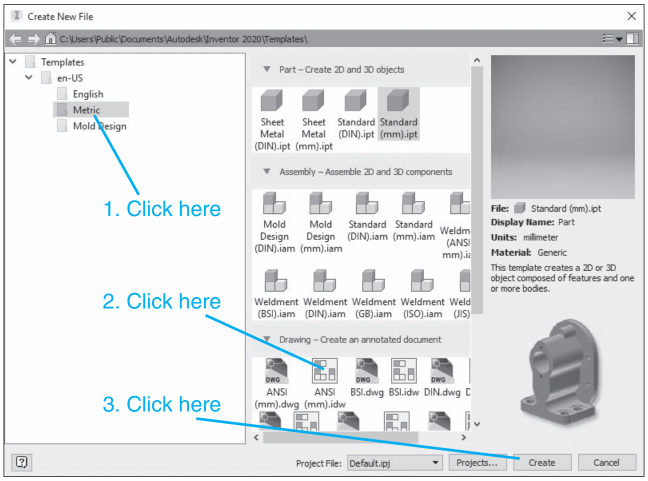


## EXERCISE 4-1 Creating an Orthographic View

**1** Start a new drawing, click the **Metric** tab, and select the **ANSI (mm).idw** option.

See Figure 4-23. ANSI stands for American National Standards Institute.

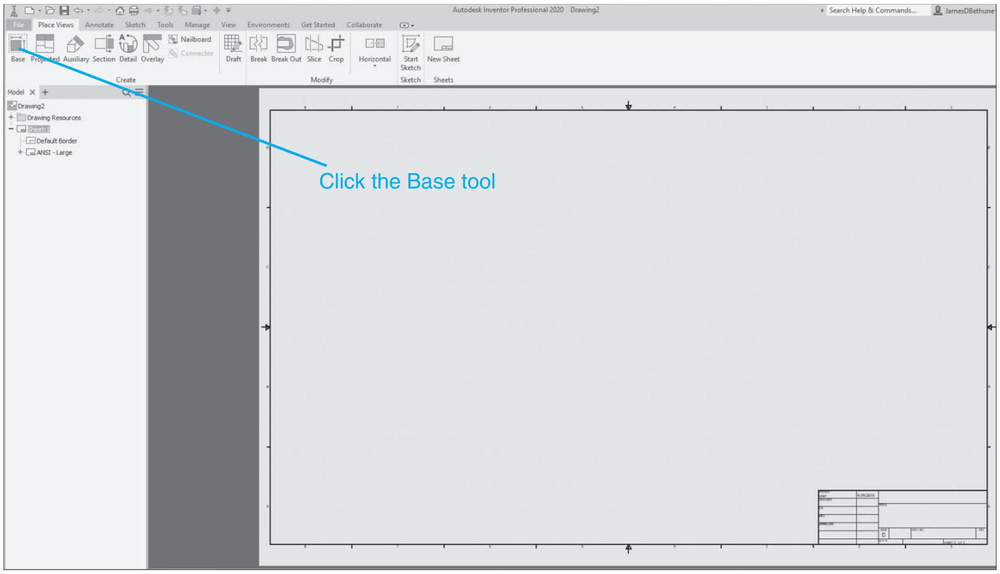
**Figure 4-23**




**2** Click **Create.**

The drawing management screen will appear. See Figure 4-24.

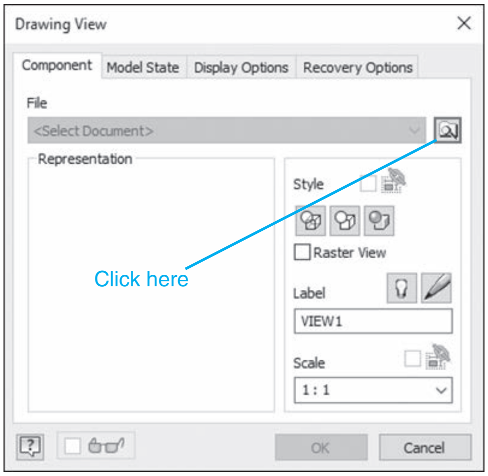
**Figure 4-24**



3 Click the **Base** tool located on the **Create** panel under the **Place Views** tab.

The **Drawing View** dialog box will appear. See Figure 4-25.

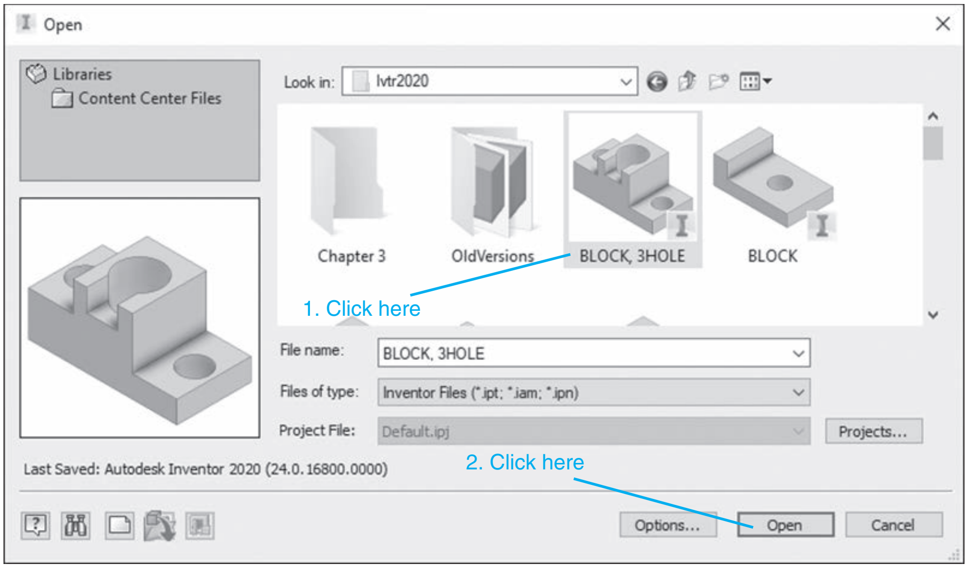
Figure 4-25




4 Click the **Open an existing file** button.

The **Open** dialog box will appear. See Figure 4-26.

Figure 4-26




5 Select the desired model. In this example, the model's file name is **BLOCK, 3HOLE**.

6 Click the **Open** box.

The **Drawing View** dialog box will appear. See Figure 4-27.

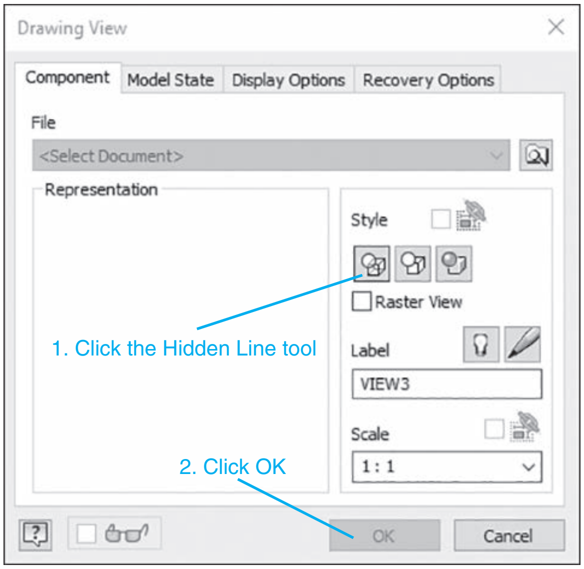



**Figure 4-27**

**Figure 4-28**

**7** Ensure that the **Hidden line** option is active, and click **OK**.

Figure 4-28 shows the resulting orthographic view. The selection of orientation will vary with the model's original orientation.

The screen will include a border and a title block. The lettering in the title block may appear illegible. This is normal. The text will be legible when printed. The section on title blocks will explain how to work with title blocks.

## EXERCISE 4-2 Creating Other Orthographic Views

**1** The view shown in Figure 4-28 will be defined as a top view. Click and drag the border around the view and move it upward on the drawing screen. Click the **Projected View** tool on the **Create** panel under the **Place Views** tab.

**2** Click the view already on the drawing screen.

**3** Move the cursor downward from the view.

A second view will appear.

**4** Select a location, click the left mouse button to place the view, then click the right mouse button and select the **Create** option.

Figure 4-29 shows the resulting two orthographic views. The initial view is defined as the **Top** view. This is a relative term based on the way the model was drawn. The initial view can be defined as the **Top** view, and the second view created from that front view.

## EXERCISE 4-3 Adding Centerlines

Convention calls for all holes to be defined using centerlines. The views in Figure 4-29 do not include centerlines.

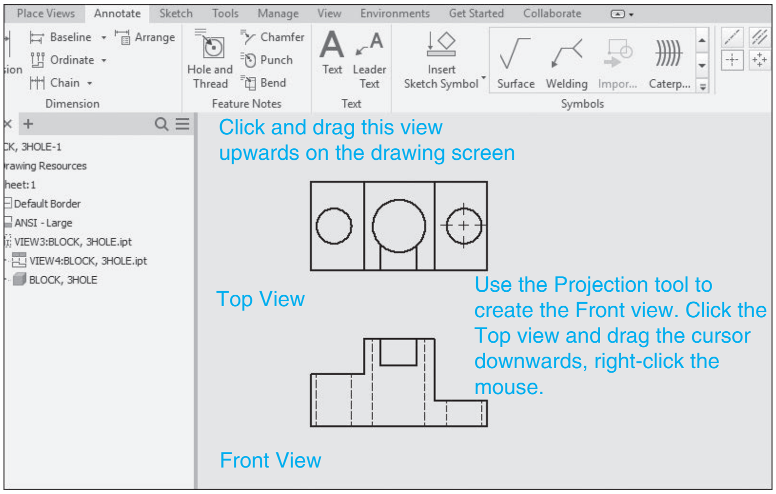



**Figure 4-29**

**1** Click the **Annotate** tab.

See Figure 4-30.

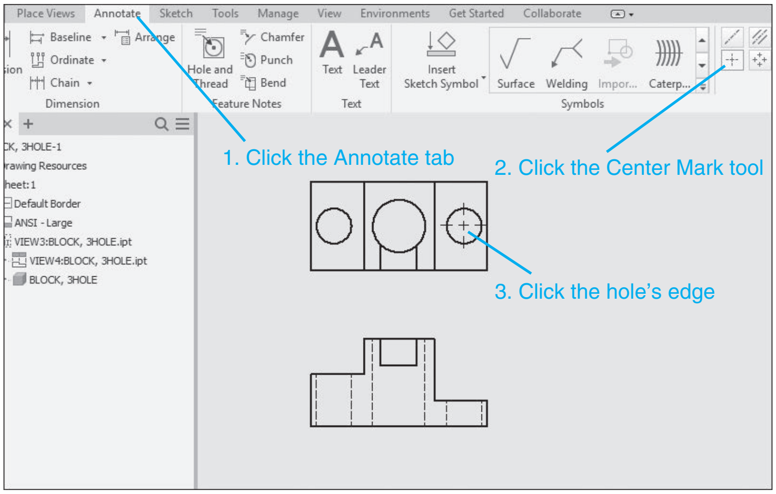



**Figure 4-30**

**2** Click the **Center Mark** tool located on the **Symbols** panel under the **Annotate** tab.

**3** Move the cursor into the drawing screen and click the edges of the holes in the top view. Right-click the mouse and select the **OK** option.

**4** Click and drag the individual center lines for each hole in the **Top view** to create a single center line through all three holes as shown. When the cursor is moved onto a center mark, green-filled circles will appear. Click and drag these circles to form a single center line.

See Figure 4-31.

5 Click the **Centerline Bisector** tool located on the **Symbols** panel under the **Annotate** tab.

6 Click each side of the holes' projections in the front view.

Vertical centerlines will appear. See Figure 4-32.

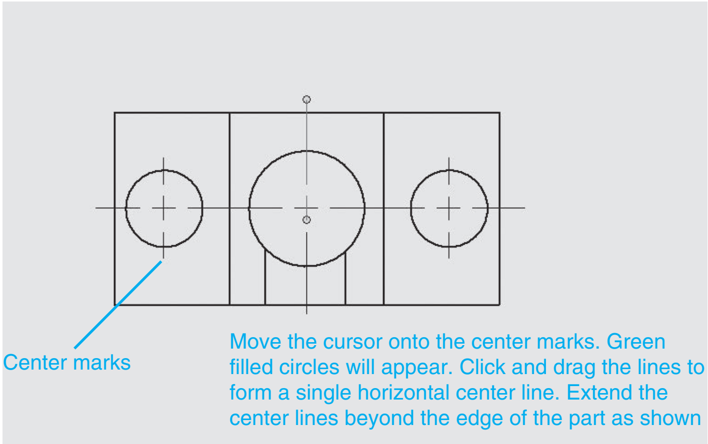



**Figure 4-31**

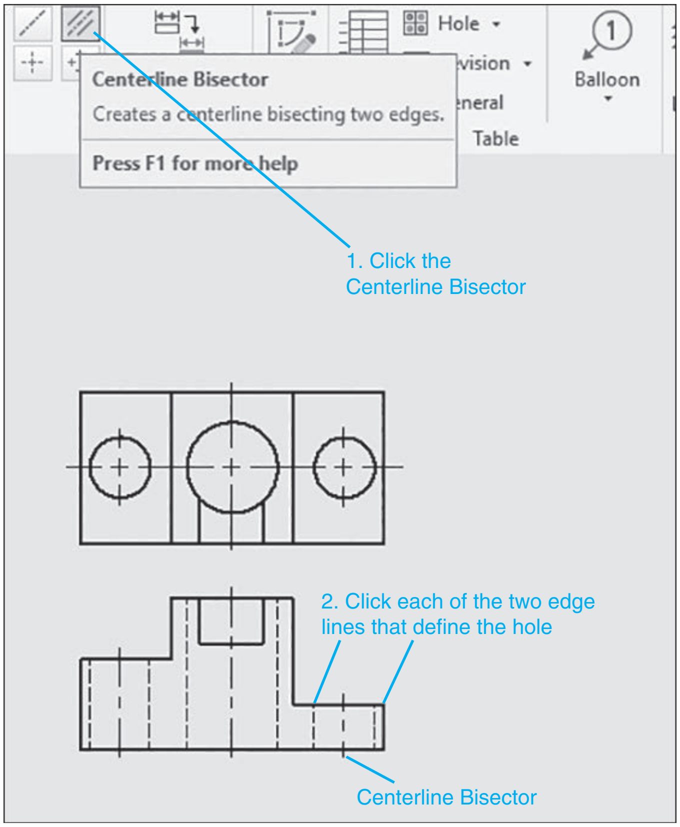



**Figure 4-32**

## EXERCISE 4-4 Editing the Size of a Centerline

If the centerline patterns are too small or too big for the given feature, they may be edited to create a more pleasing visual picture.

**1** Click the **Styles Editor** tool located on the **Styles and Standards** panel under the **Manage** tab.

Click the **+** sign to the left of the **Center Mark** heading, and select the **Center Mark (ANSI)** option. See Figure 4-33.

**2** Change the center mark values as needed.

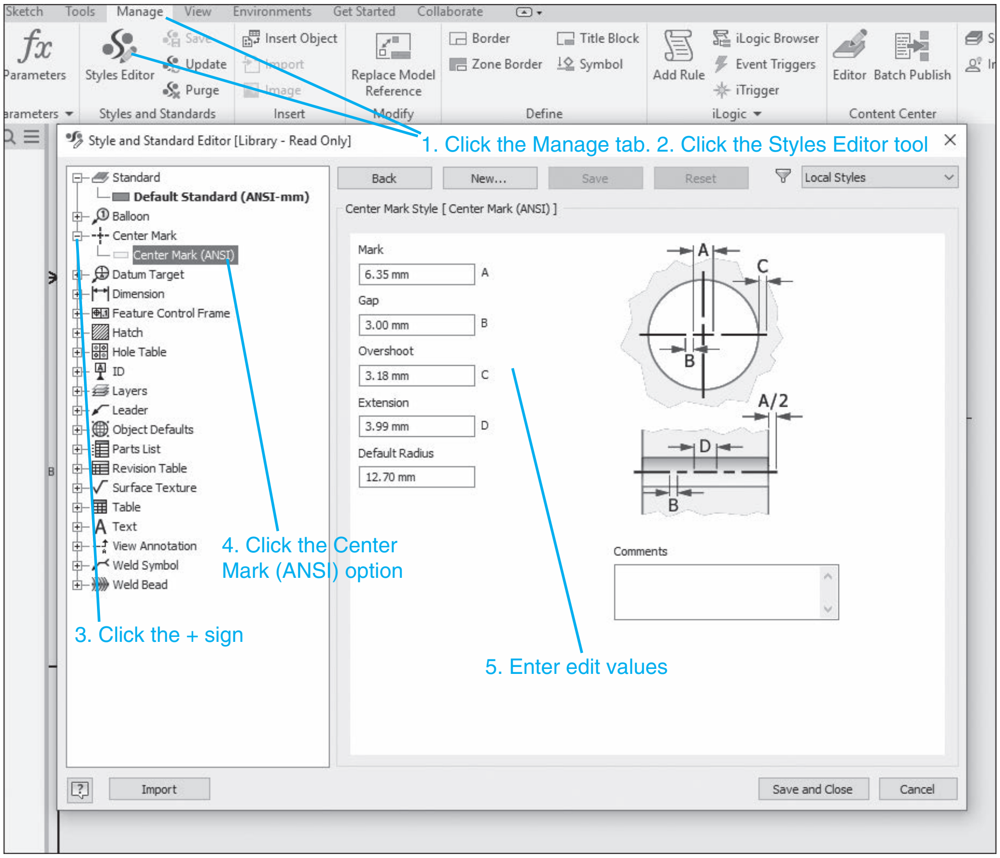



Figure 4-33

## EXERCISE 4-5 Changing the Background Color of the Drawing Screen

**1** Click the **Tools** tab at the top of the screen.

Select the **Application Options** option.

The **Application Options** dialog box will appear. See Figure 4-34.

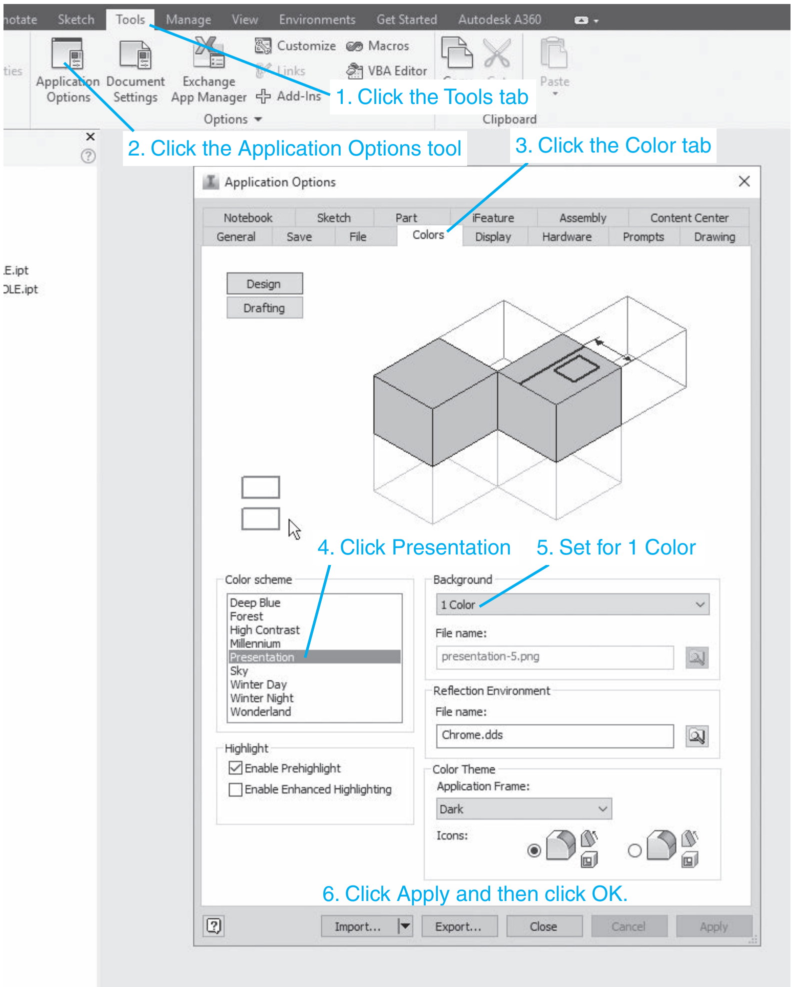
**2** Click the **Colors** tab.

**3** Click the desired color, then **OK**.

The background color will be changed. In this example the **Presentation Color scheme** and **1 Color Background** were selected. This format is used throughout the book for visual clarity.

**4** Click **Apply** and **OK**.

Figure 4-34




## Isometric Views

An isometric view may be created from any view on the screen. The resulting orientation will vary according to the view selected. In this example, the front view is selected.

1 Access the **Create** panel under the **Place Views** tab and click the **Projected View** tool.

2 Click the **Front** orthographic view.

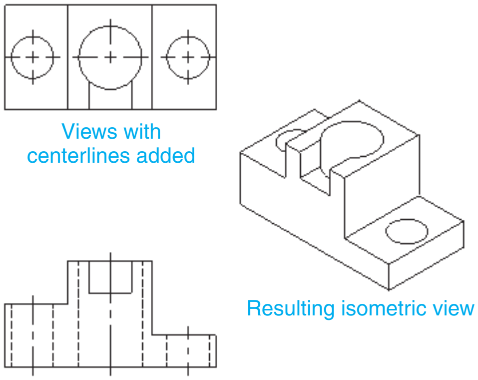
3 Move the cursor to the right of the front view and select a location for the isometric view by clicking the mouse.

4 Move the cursor slightly and click the right mouse button.

5 Select the **Create** option.

Figure 4-35 shows the resulting isometric view. Isometric views help visualize the orthographic views.

**Figure 4-35**




## Section Views

**section view:** A view used to expose an internal surface of a model.

**cutting plane:** A plane used to define the location of a section view.

Some objects have internal surfaces that are not directly visible in normal orthographic views. ***Section views*** are used to expose these surfaces. Section views do not include hidden lines.

Any material cut when a section view is defined is hatched using section lines. There are many different styles of hatching, but the general style is evenly spaced 45° lines. This style is defined as ANSI 31 and will be applied automatically by Inventor.

Figure 4-36 shows a three-dimensional view of an object. The object is cut by a cutting plane. ***Cutting planes*** are used to define the location of the section view. Material to one side of the cutting plane is removed, exposing the section view.

**Figure 4-36**

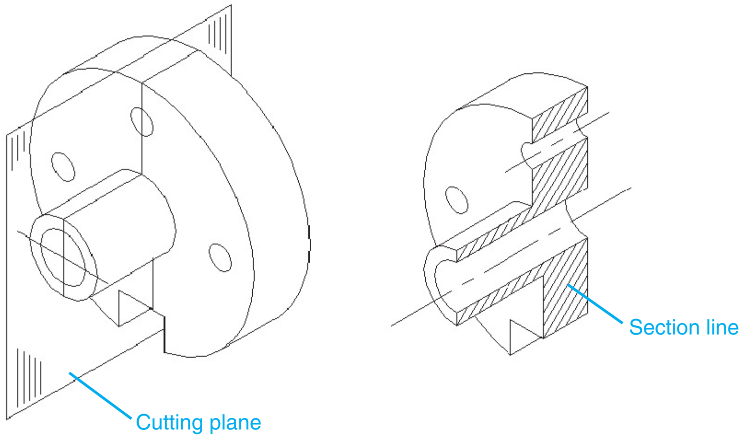



Figure 4-37 shows the same object presented using two dimensions. The cutting plane is represented by a cutting plane line. The cutting plane line is defined as *A-A*, and the section view is defined as view *A-A*.

All surfaces directly visible must be shown in a section view. In Figure 4-38, the back portion of the object is not affected by the section view and is directly visible from the cutting plane. The section view must include these surfaces. Note how the rectangular section blocks out part of the large hole. No hidden lines are used to show the hidden portion of the large hole.

**Figure 4-37**

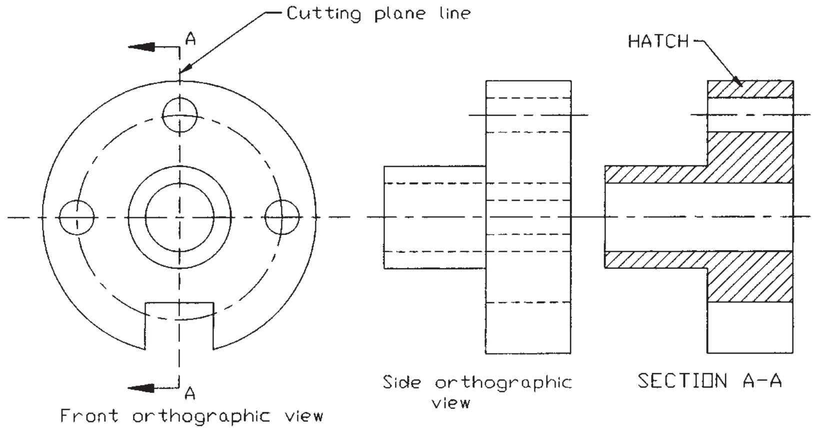
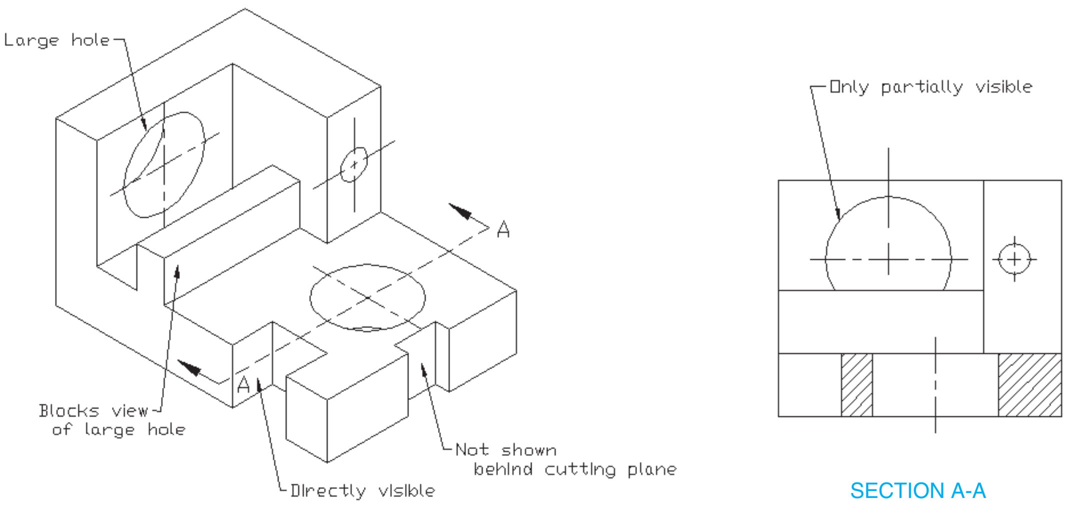



**Figure 4-38**



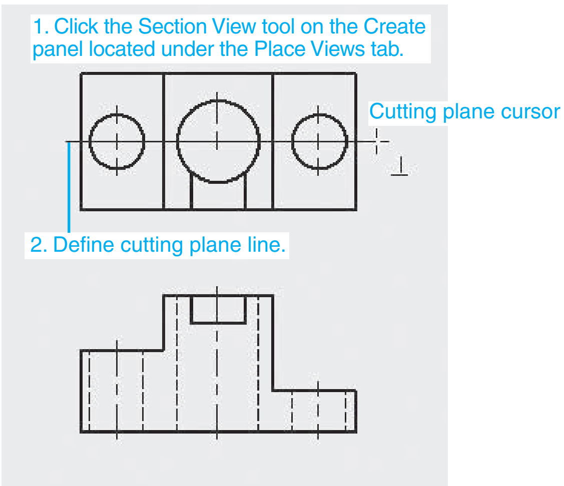

## EXERCISE 4-6 Drawing a Section View Using Inventor

Figure 4-39 shows the front and top views of the object defined in Figure P3-10. A section view will be created by first defining the cutting plane line in the top view, then projecting the section view below the front view.

**1** Click the **Section View** tool on the **Create** panel under the **Place Views** tab, then click the top view.

The cursor will change to a +-like shape.

**Figure 4-39**



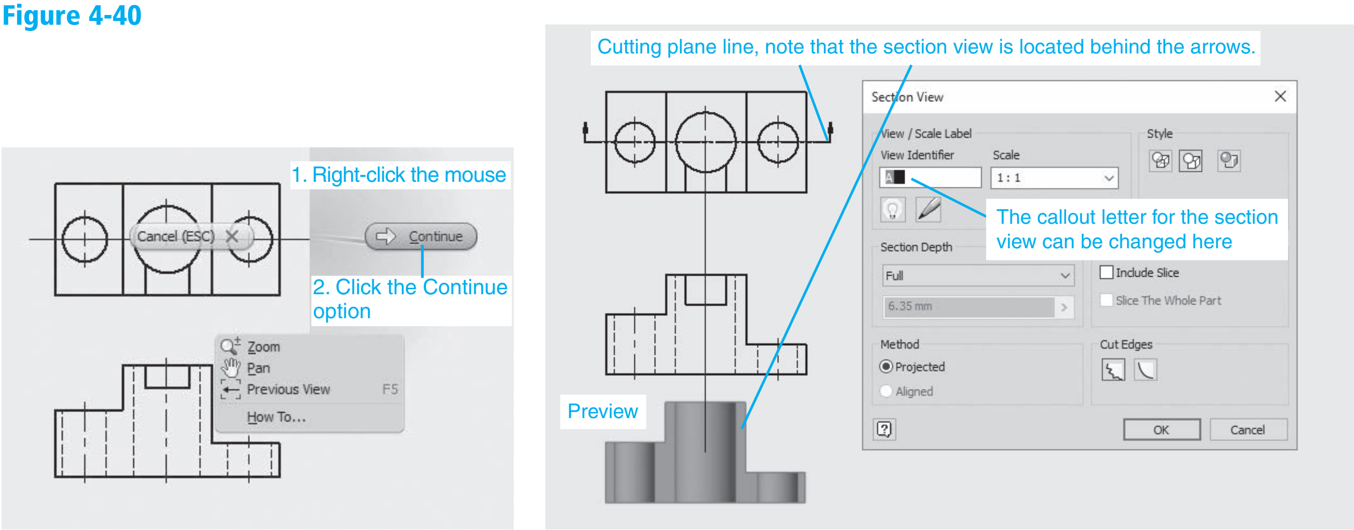
**2** Define the cutting plane by defining two points on the top view.

See Figure 4-39. Note that if you touch the cursor to the endpoint of one of the hole's centerlines, a dotted line will follow the cursor, assuring that the cutting plane line is aligned with the holes' centerlines.

**3** Right-click the mouse and select the **Continue** option.

The **Section View** dialog box will appear. See Figure 4-40.

**Figure 4-40**




**4** Set the **Label** letter for **A,** the **Scale** for **1:1,** and **Style** for **Hidden Lines**.

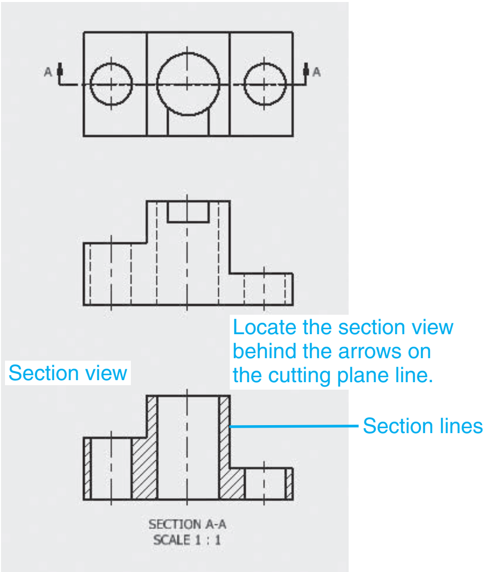
**5** Move the cursor so as to position the section view below the front view.

**6** Click the section view location.

**7** Add the appropriate centerlines using the **Centerline Bisector** tool.

Figure 4-41 shows the resulting section view. Notice that the section view is defined as *A-A*, and the scale is specified. The arrows of the cutting plane line are directed away from the section view. The section view is located behind the arrows.

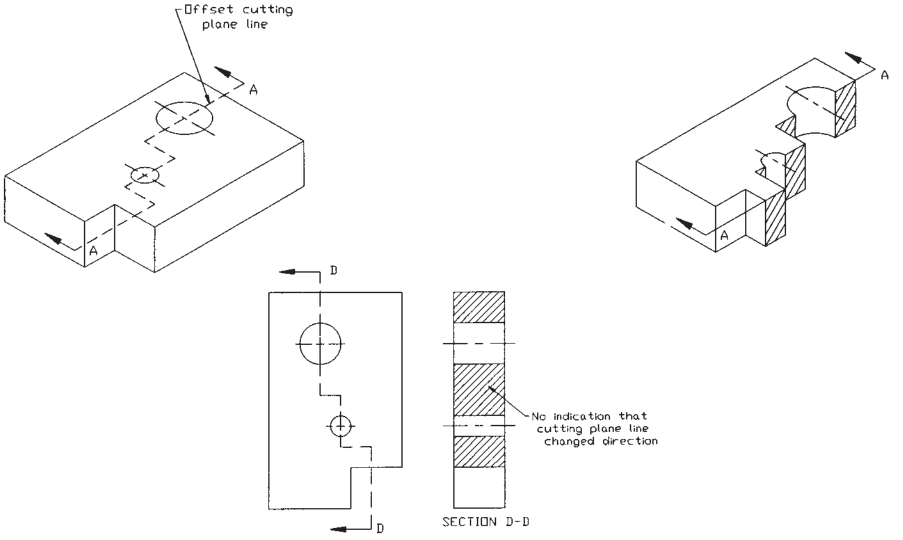
**Figure 4-41**



## Offset Section Views

Cutting plane lines need not pass directly across an object, but may be offset to include several features. Figure 4-42 shows an object that has been cut using an offset cutting plane line.

Figure 4-42




EXERCISE 4-7 **Creating an Offset Cutting Plane**

Figure 4-43 shows the front and top views of an object. The views were created using the **Create View**, **Projected View**, and **Centerline** tools.

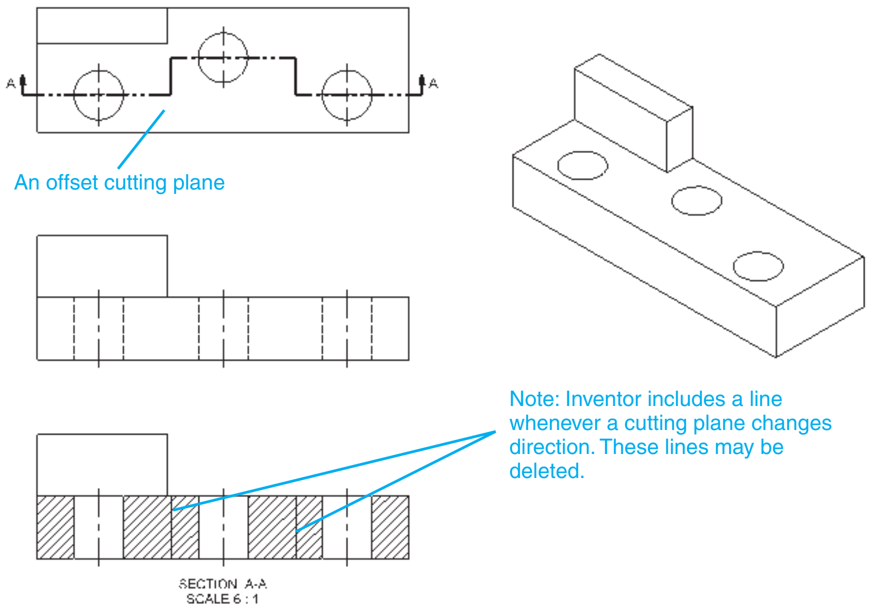
1. Click the **Section View** tool, and click the top view.

2. Draw a cutting plane across the top view through the centers of each of the three holes.

   When drawing an offset cutting plane line, show the line in either horizontal or vertical line segments.

3. Locate the section view below the front view and add the appropriate centerlines.

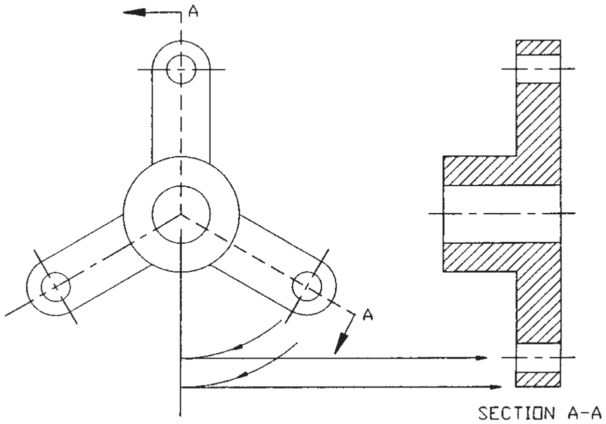
Figure 4-43



## Aligned Section Views

Figure 4-44 shows an example of an aligned section view. Aligned section views are most often used on circular objects and use an angled cutting plane line to include more features in the section view, like an offset cutting plane line.

**Figure 4-44**




An aligned section view is drawn as if the cutting plane line ran straight across the object. The cutting plane line is rotated into a straight position, and the section view is projected.

Figure 4-45 shows an aligned section view created using Inventor.

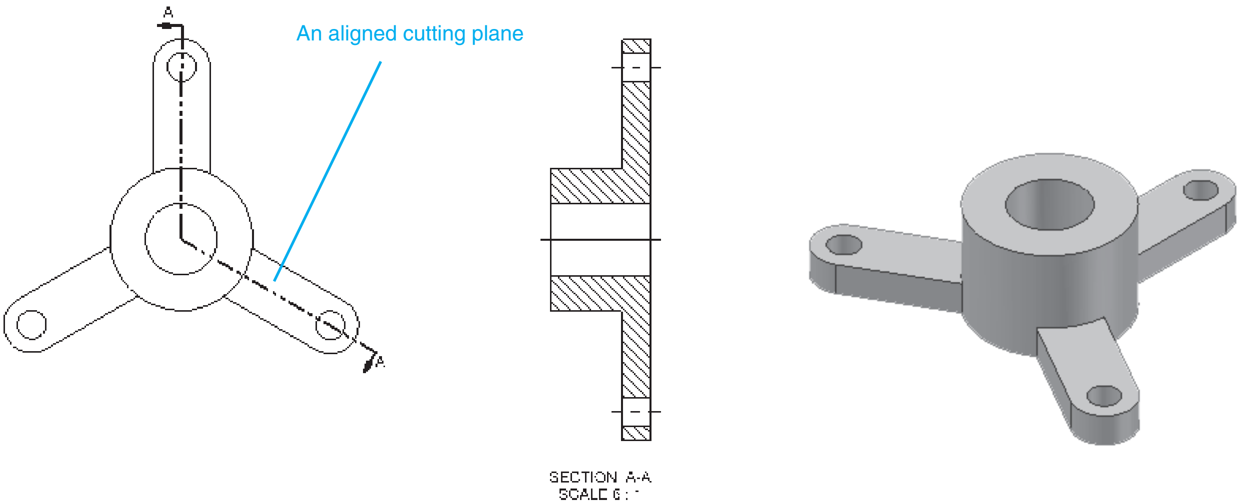



**Figure 4-45**

## Detail Views

**detail view:** An enlarged view of a portion of a model.

***Detail views*** are used to enlarge portions of an existing drawing. The enlargements are usually made of areas that could be confusing because of many crossing or hidden lines.

## EXERCISE 4-8 Creating a Detail View

**1** Click the **Detail View** tool on the **Create** panel under the **Place Views** tab, then click the view to be enlarged.

In this example, the top view was selected.

The **Detail View** dialog box will appear. See Figure 4-46.

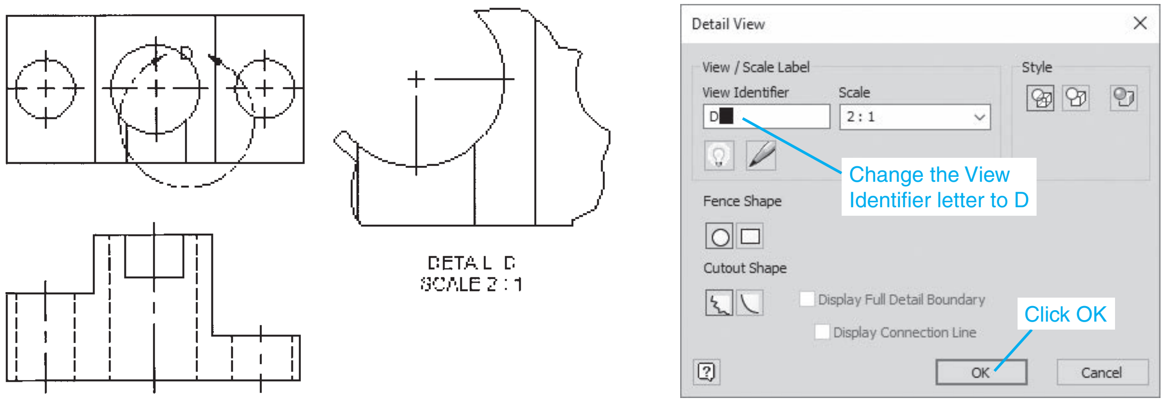



Figure 4-46

**2** Set the **Label** letter to **D** and the **Scale** to **2:1**, then pick a point on the view.

**3** Move the cursor, creating a circle.

The circle will be used to define the area of the detail view.

**4** When the circle is of an appropriate diameter, click the left mouse button and move the cursor away from the view.

**5** Locate the detail view and click the location.

**6** Use the **Center Mark** tool to add a center mark to the circle in the Detail drawing.

# Break Views

It is often convenient to break long, continuous shapes so that they take up less drawing space. Figure 4-47 shows a long L-bracket that has a continuous shape; that is, its shape is constant throughout its length. Figure 4-48 shows an orthographic view of the same L-bracket.

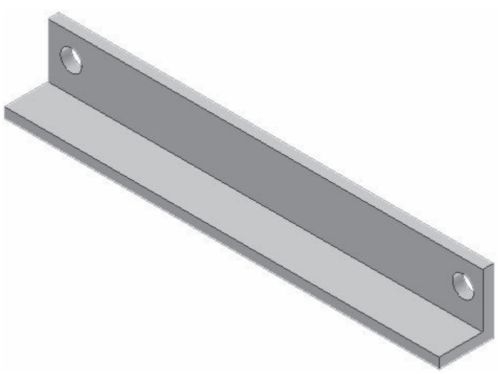
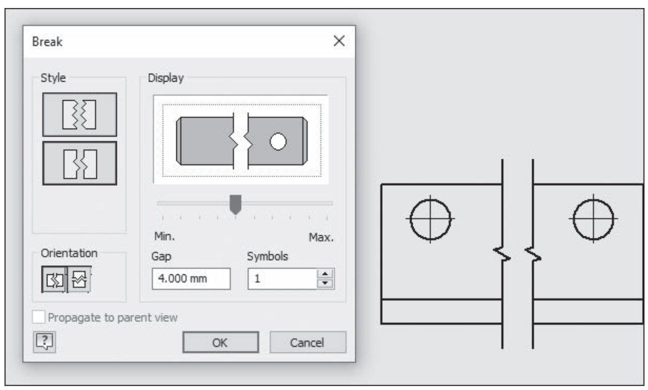


Figure 4-47




The overall dimension remains the same as the original

Figure 4-48

**EXERCISE 4-9** Creating a Broken View

1. Click the **Break** tool located on the **Create** panel under the **Place Views** tab, then click the orthographic view.

   The **Broken View** dialog box will appear.

2. Select the orientation of the break and the gap distance between the two portions of the L-bracket.

   In this example, the gap distance is 1.00. Do not click the **OK** box. Define the break with the **Broken View** dialog box on the drawing screen.

3. Click a point near the left end of the L-bracket, then move the cursor to the right and click a second point near the right end of the L-bracket.

   Figure 4-48 shows the resulting broken view.

## Multiple Section Views

It is acceptable to take more than one section view of the same object to present a more complete picture of the object. Figures 4-49 and 4-50 show objects that use more than one section view.

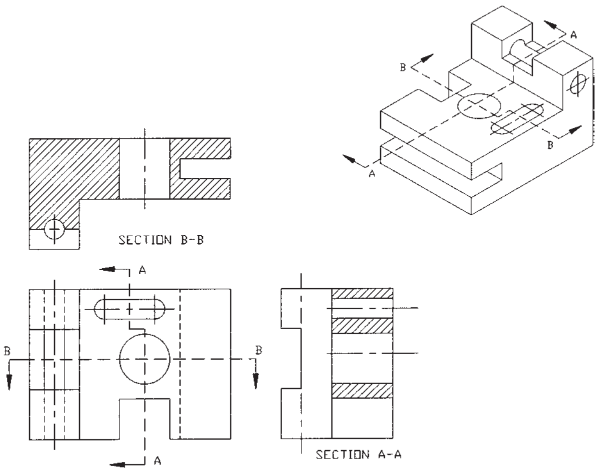



Figure 4-49

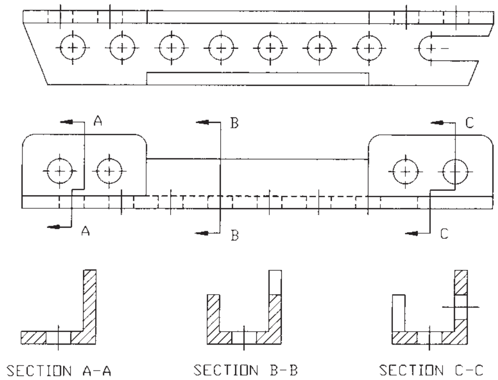



Figure 4-50

## Auxiliary Views

**auxiliary view:** An orthographic view drawn perpendicular to a slanted or oblique surface.

***Auxiliary views*** are orthographic views used to present true-shape views of slanted surfaces. Figure 4-51 shows an object with a slanted surface that includes a hole drilled perpendicular to the slanted surface. Note how the right-side view shows the hole as an ellipse and that the surface *A-B-C-D* is foreshortened; that is, it is not shown at its true size. Surface *A-B-C-D* does appear at its true shape and size in the auxiliary view. The auxiliary view was projected at 90° from the slanted surface so as to generate a true-shape view.

Figure 4-52 shows an object that includes a slanted surface and hole.

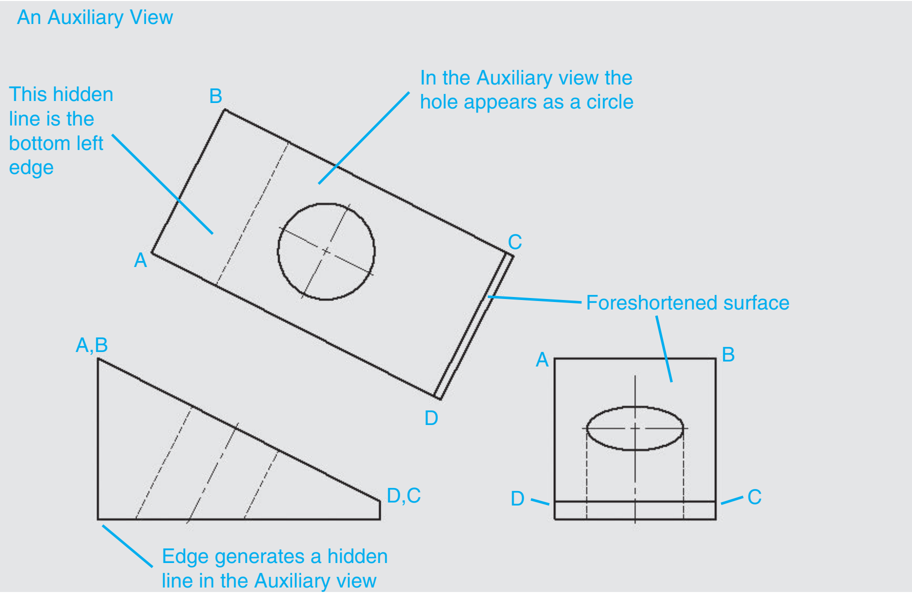



**Figure 4-51**

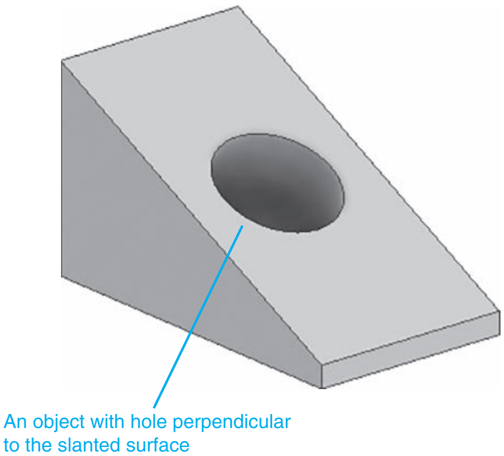



**Figure 4-52**

## EXERCISE 4-10 Drawing an Auxiliary View

1. Create a drawing using the **ANSI (mm).ipt** format. Click the **Base View** and **Projected View** tools on the **Create** panel under the **Place Views** tab, and create a front and a right-side view as shown in Figure 4-53.

   Click the **Auxiliary View** tool, then the front view.

2. The **Auxiliary View** dialog box will appear.

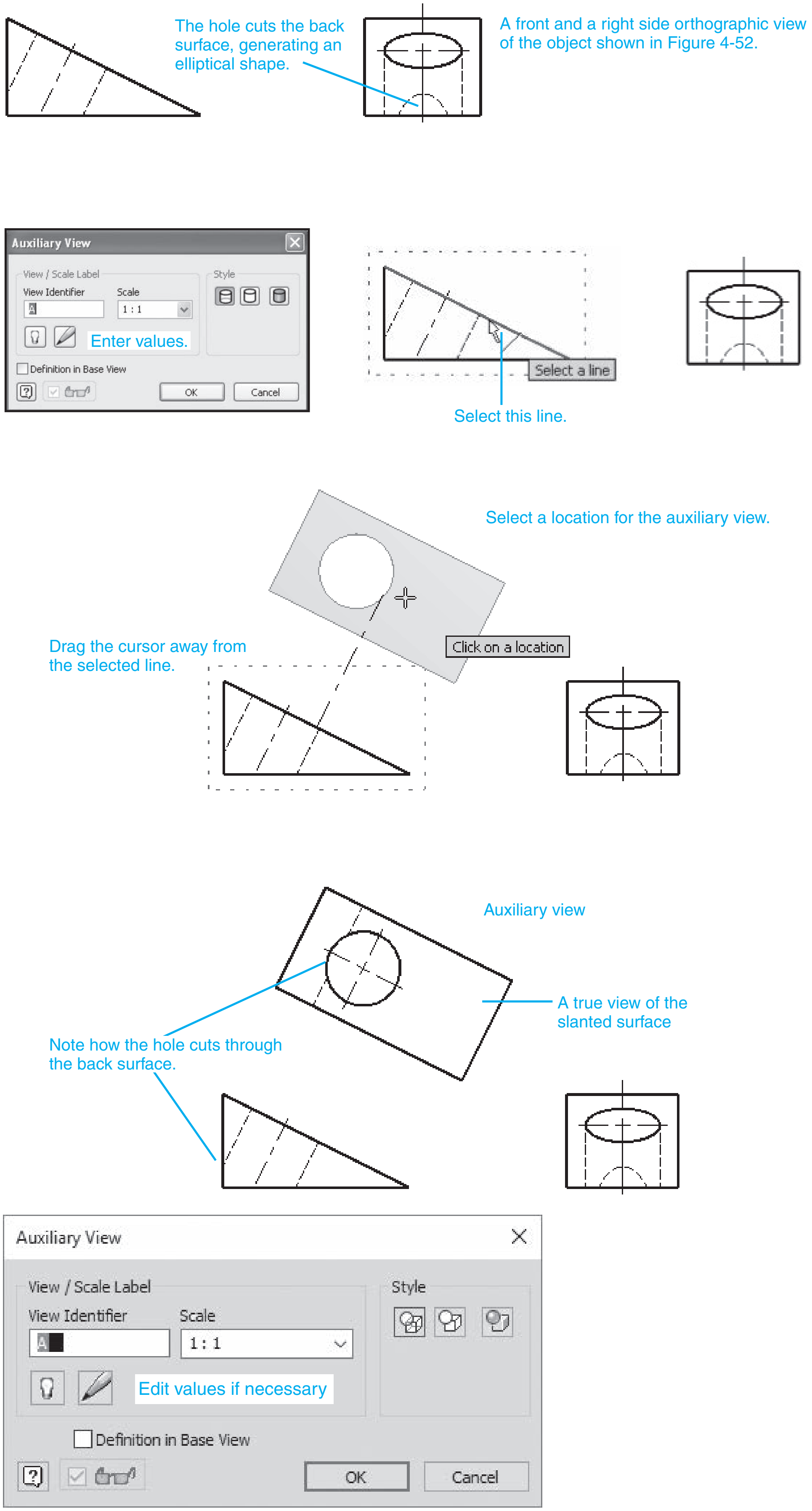
3. Enter the appropriate settings, then click the slanted edge line in the front view.

   In this example, a scale of 1:1 was used.

4. Move the cursor away from the front view and select a location for the auxiliary view.

5. Click the left mouse button and create the auxiliary view.

**Figure 4-53**


The hole cuts the back surface, generating an elliptical shape.
A front and a right side orthographic view of the object shown in Figure 4-52.
Auxiliary View
View / Scale Label
View Identifier
Scale
1 : 1
Style
Enter values.
Definition in Base View
OK
Cancel
Select a line
Select this line.
Select a location for the auxiliary view.
Click on a location
Drag the cursor away from the selected line.
Auxiliary view
A true view of the slanted surface
Note how the hole cuts through the back surface.
Auxiliary View
View / Scale Label
View Identifier
Scale
1 : 1
Style
Edit values if necessary
Definition in Base View
OK
Cancel

## ASME Y14.3-2003

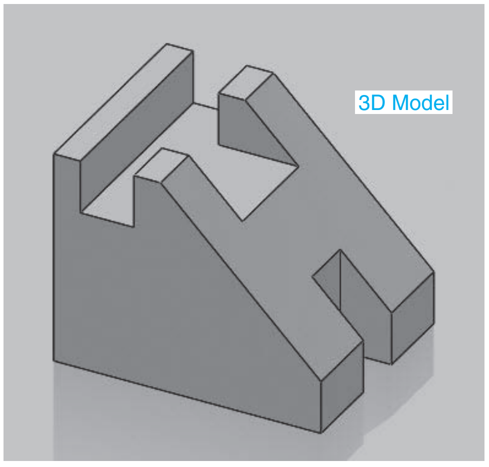
### Drawing Standards

There are two sets of standards used to define the projection and placement of orthographic views: the ANSI and the ISO. The ANSI calls for orthographic views to be created using third-angle projection and is the accepted method for use in the United States. See the American Society of Mechanical Engineers (ASME) publication ASME Y14-3-2003. Some countries other than the United States use first-angle projection. See ISO publication 128-30.

This chapter has presented orthographic views using third-angle projections as defined by ANSI. However, there is so much international commerce happening today that you should be able to work in both conventions, just as you should be able to work in both inches and millimeters.

Figure 4-54 shows a three-dimensional model and three orthographic views created using third-angle projection and three orthographic views created using first-angle projection. Note the differences and similarities. The front view in both projections is the same. The top views are the same, but are in different locations. The third-angle projection presents a right side view, whereas the first-angle projection presents a left side view.

**Figure 4-54**



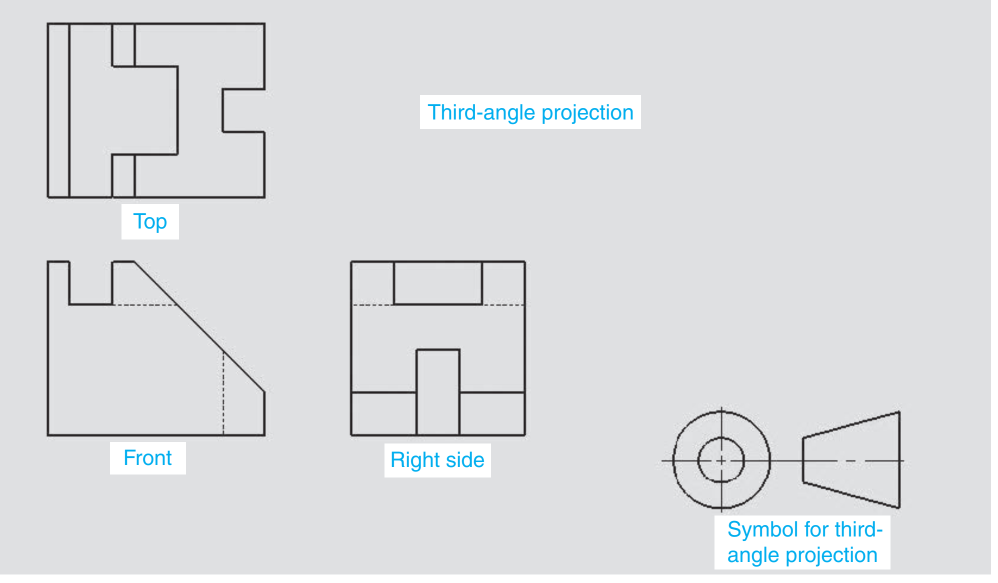
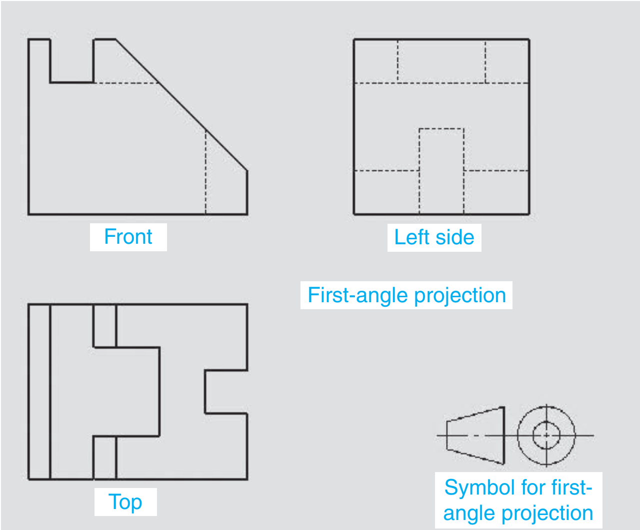



**Figure 4-54**
(*Continued*)



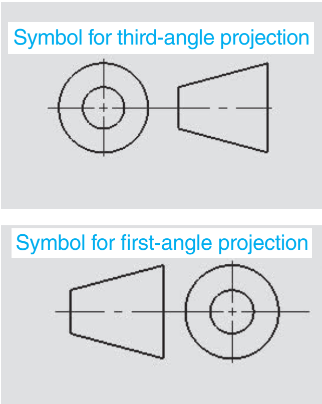




**Figure 4-55**

Figure 4-55 shows the drawing symbols for first- and third-angle projections. These symbols can be added to a drawing to help the reader understand which type of projection is being used. These symbols were included in the projections presented in Figure 4-54.

## Third- and First-Angle Projections

Figure 4-56 shows an object with a front orthographic view and two side orthographic views: one created using third-angle projection, and the other created using first-angle projection. For third-angle projections, the orthographic view is projected on a plane located between the viewer's position and the object. For first-angle projections, the orthographic view

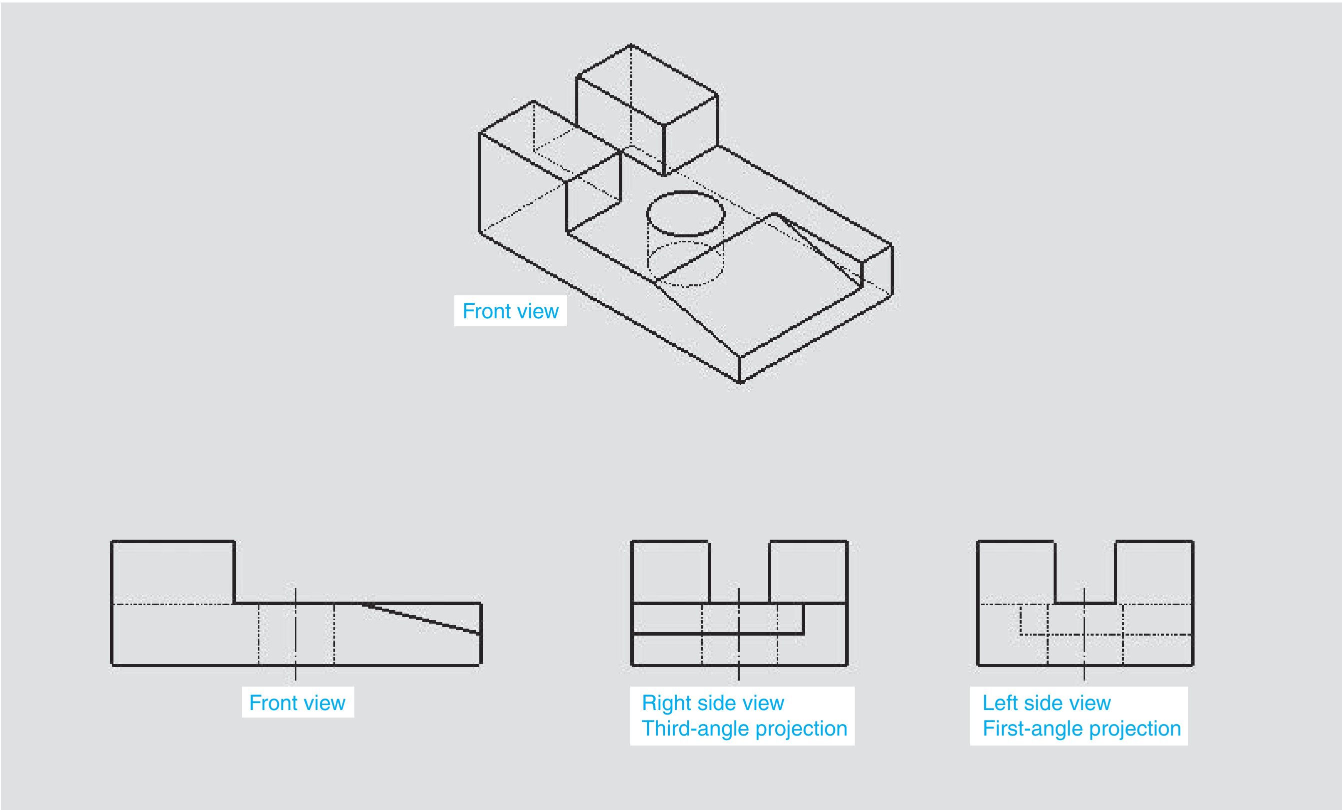



**Figure 4-56**

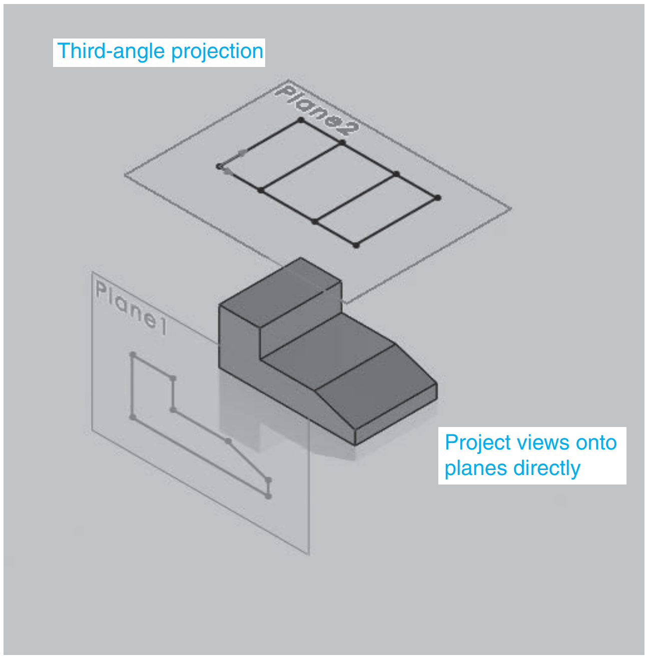



**Figure 4-57**

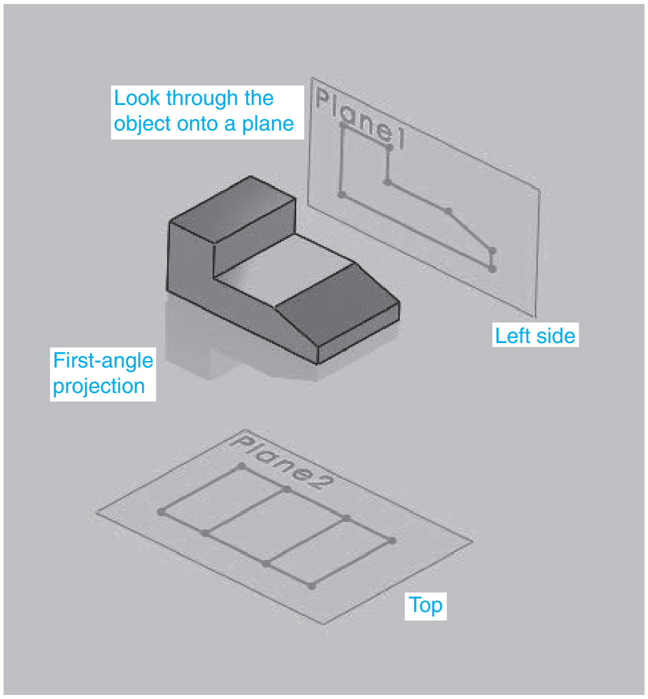



**Figure 4-58**

is projected on a plane located beyond the object. The front and top views for third- and first-angle projections appear the same, but they are located in different positions relative to the front view.

The side orthographic views are different for third- and first-angle projections. Third-angle projection uses a right side view located to the right of the object. First-angle projections use a left side view located to the right of the object. Figures 4-57 and 4-58 show the two different side view projections for the same object. For third-angle projection, the viewer is located on the right side of the object and creates the side orthographic view on a plane located between the view position and the object. The viewer looks directly at the object. For first-angle projection, the viewer is located on the left side of the object and creates the side orthographic view on a plane located beyond the object. The viewer looks through the object.

To help understand the difference between side view orientations for third- and first-angle projections, locate your right hand with the heel facing down and the thumb facing up. Rotate your hand so that the palm is facing up—this is the third-angle projection orientation. Return to the thumb up position. Rotate your hand so that the palm is down—this is the first-angle view orientation.

To create first-angle projections using Inventor:

1 Start a **New** drawing using the **ISO.idw** template.

This template will automatically create first-angle projection drawings.

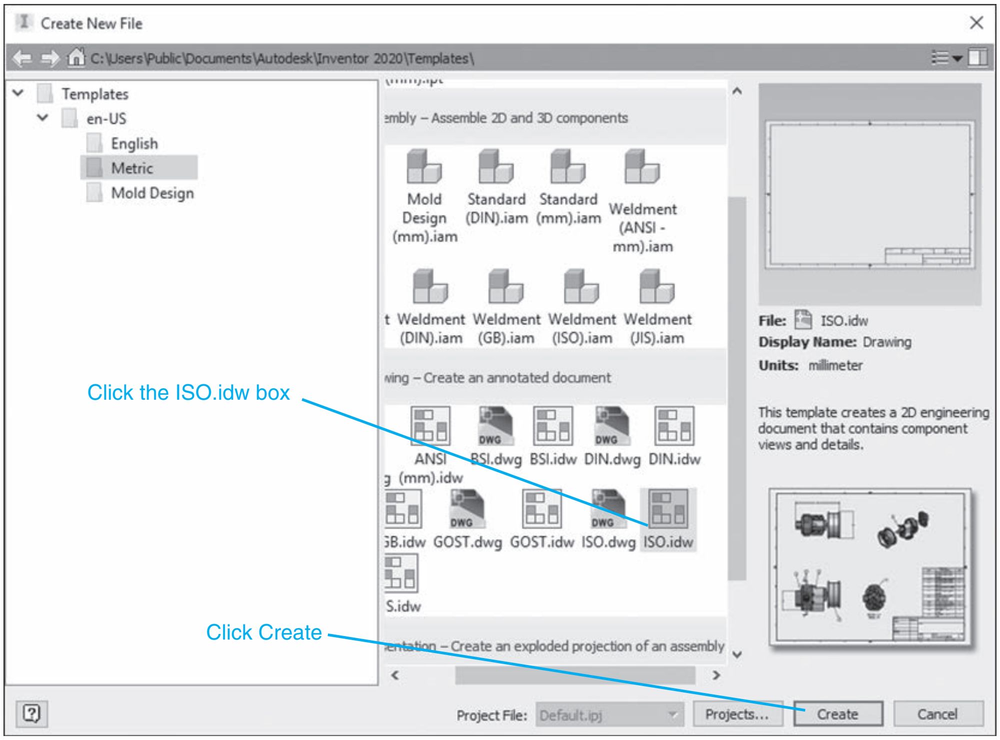
2 Click the **Base** tool.

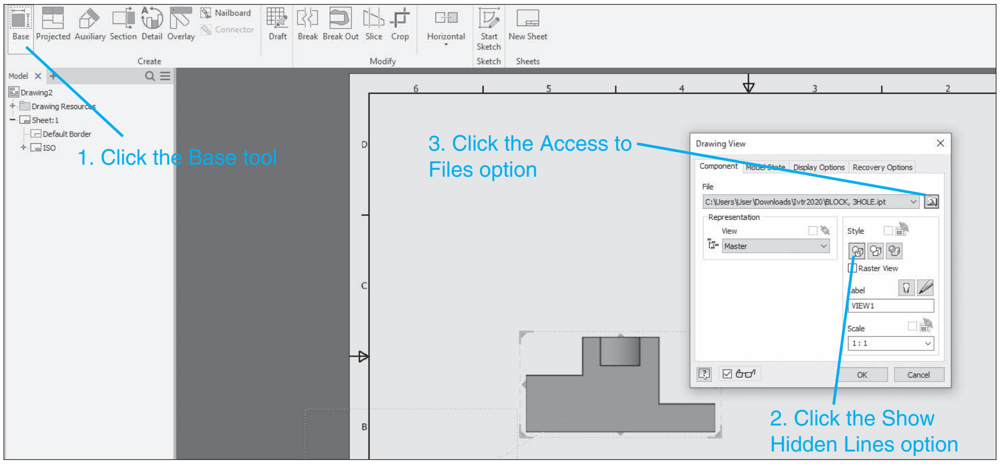
3 Select the appropriate file.

4 Select the orientation.

5 Use the **Projected View** tool to select and position the views (Figure 4-59).

**Figure 4-59**




Chapter 4

# Chapter Summary

This chapter introduced orthographic drawings using third-angle projection in accordance with ANSI standards. Conventions were demonstrated for objects with normal surfaces, hidden lines, slanted surfaces, compound lines, oblique surfaces, and rounded surfaces.

Inventor creates orthographic views directly from models. The tools on the **Create** panel and the **Annotate** panel were introduced for managing orthographic presentation views. Isometric views can also be created from models.

Section views are used to expose internal surfaces that are not directly visible in normal orthographic views. Cutting planes were used to define the location of section views. Offset and aligned section views also were created.

Techniques for creating detail views, broken views, and auxiliary views were demonstrated as well.

# Chapter Test Questions

## Multiple Choice

*Circle the correct answer.*

**1.** Which of the following is not one of the three views generally taken of an object?

a. Front

b. Top

c. Left

d. Right

**2.** In the precedence of lines, a hidden line covers a(n) _____ line.

a. continuous

b. center

c. compound

d. oblique

**3.** Which of the following is used to define a section view?

a. A cutting plane

b. A section line

c. A centerline

**4.** Section lines are used to define which of the following on a section view?

a. The outside edges of the section cut

b. The location of the section view

c. The areas where the section view passes through solid material

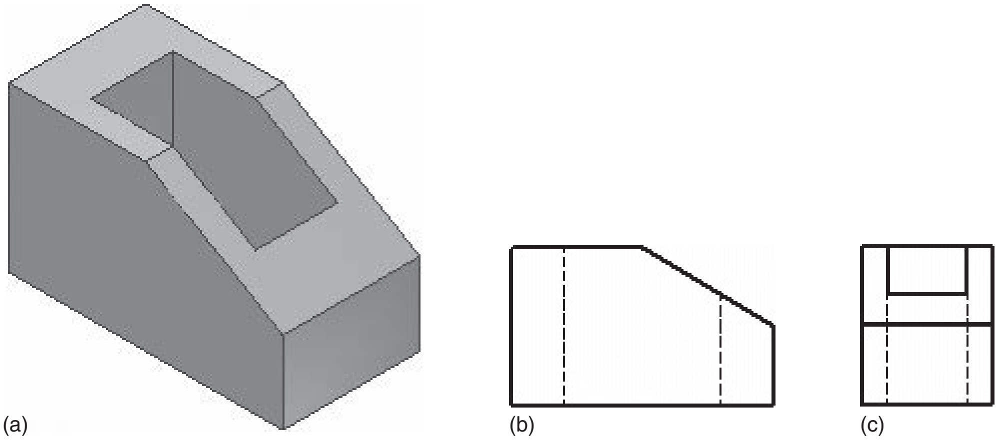


**Figure MC4-1**

5. Given the model shown in Figure MC4-1, which is the correct top view?

   a. b. c.

6. Given the model shown in Figure MC4-1, which is the correct front view?

   a. b. c.

7. Given the model shown in Figure MC4-2, which is the correct top view?

   a. b. c.

8. Given the model shown in Figure MC4-2, which is the correct right-side view?

   a. b. c.

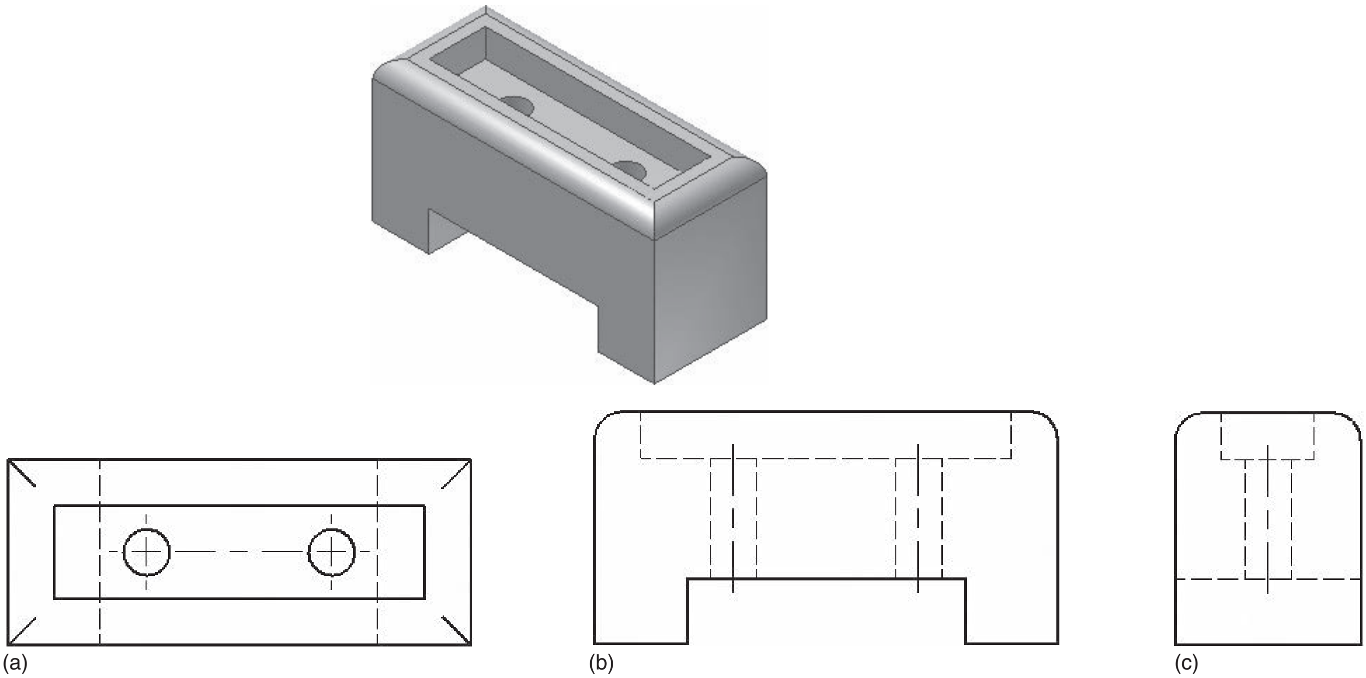


**Figure MC4-2**

9. Given the model shown in Figure MC4-2, which is the correct top view?

   a.   b.   c.

10. Given the model shown in Figure MC4-2, which is the correct front view?

   a.   b.   c.

## Matching

*Given the drawing shown in Figure MC4-3, identify the types of lines used to create the drawing.*

| Column A | Column B |
|---|---|
| a. ____________ | 1. Centerlines |
| b. ____________ | 2. Cutting plane line |
| c. ____________ | 3. Continuous line |
| d. ____________ | 4. Section line |
| e. ____________ | 5. Hidden line |

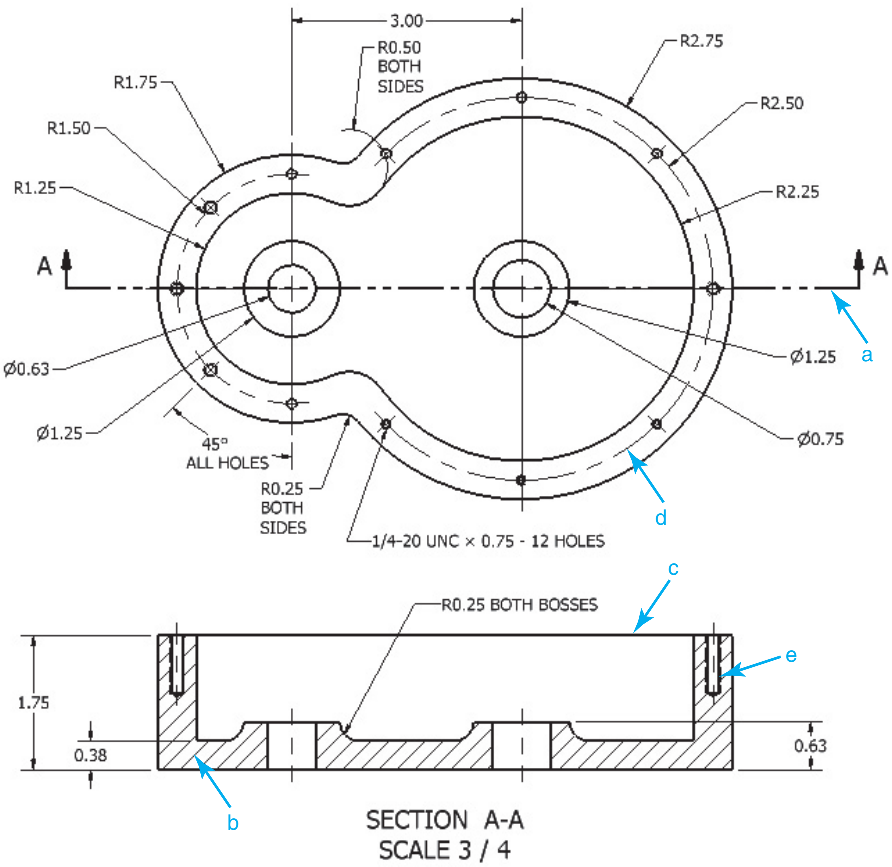



**Figure MC4-3**

## True or False

*Circle the correct answer.*

1. **True or False**: Orthographic views are two-dimensional views used to define three-dimensional models.
2. **True or False**: Normal surfaces are surfaces located 90° to each other.
3. **True or False**: Hidden lines are not used in orthographic views.
4. **True or False**: A compound line is formed when two slanted surfaces intersect.
5. **True or False**: An oblique surface is a surface that is slanted in two different directions.
6. **True or False**: Center points cannot be edited; they can be used only as they appear on the drawing screen.
7. **True or False**: A section view can be taken only across an object's centerline.
8. **True or False**: Aligned section views are most often used on circular objects.
9. **True or False**: A detail view is used to enlarge portions of an existing drawing.
10. **True or False**: Break views are used to shorten long continuous shapes so they can fit within the drawing screen.

chapterfour

# Chapter Project

## Project 4-1

Draw a front, a top, and a right-side orthographic view of each of the objects in Figures P4-1 through P4-24. Make all objects from mild steel.

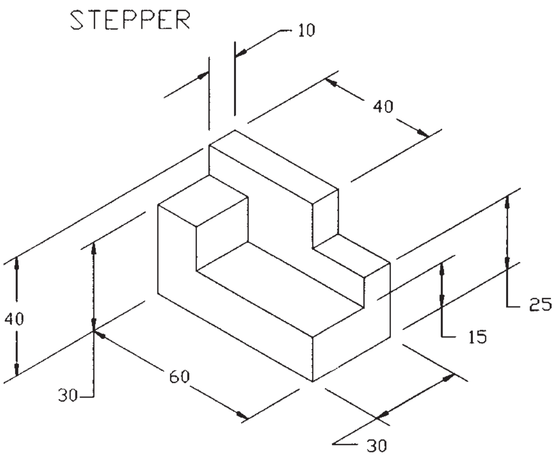



**Figure P4-1**
MILLIMETERS

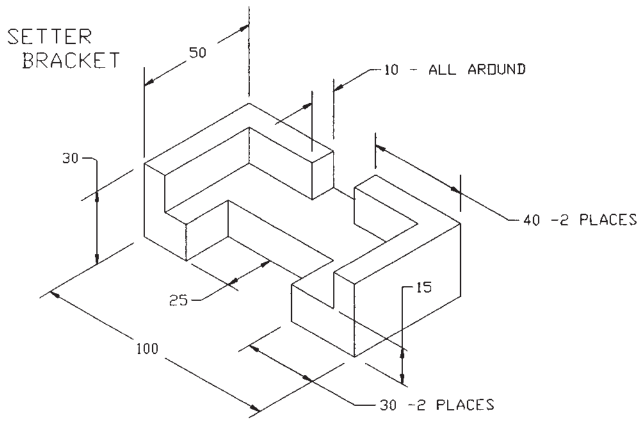



**Figure P4-2**
MILLIMETERS

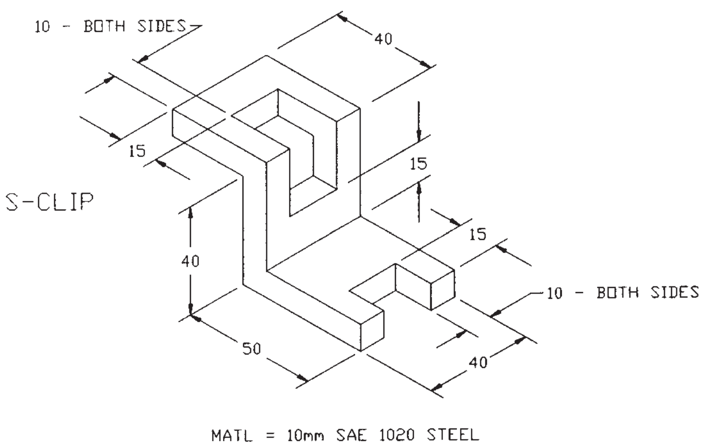



**Figure P4-3**
MILLIMETERS

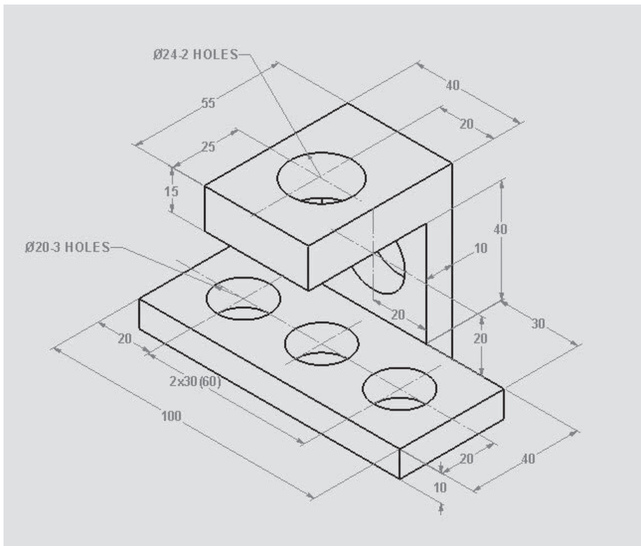



**Figure P4-4**
MILLIMETERS

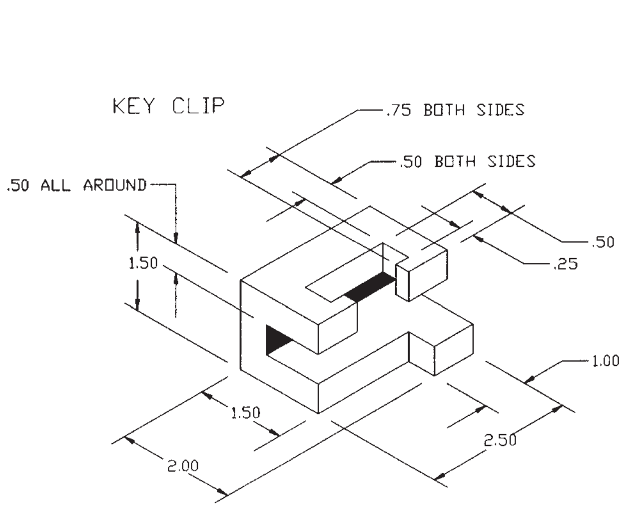



**Figure P4-5**
INCHES

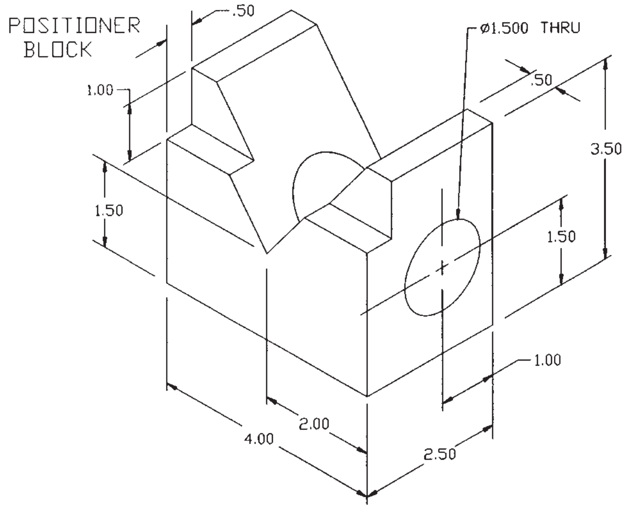



**Figure P4-6**
INCHES

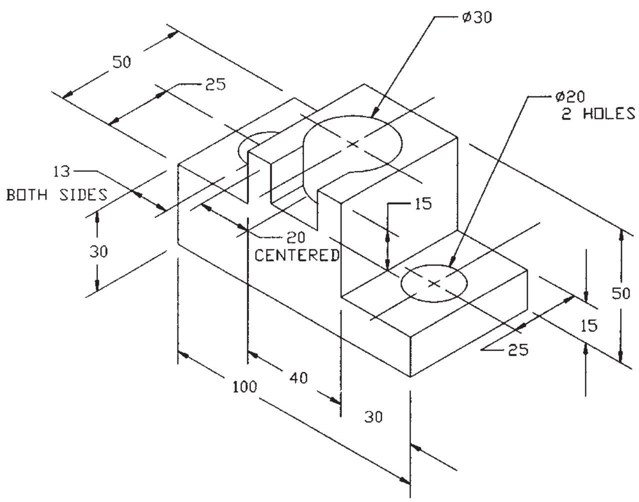



**Figure P4-7**
MILLIMETERS

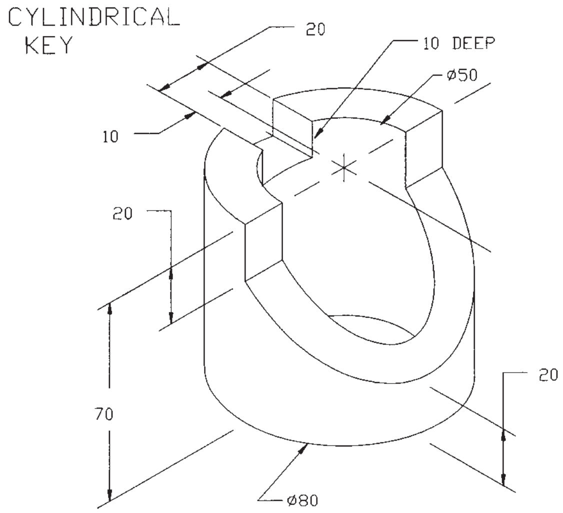



**Figure P4-8**
MILLIMETERS

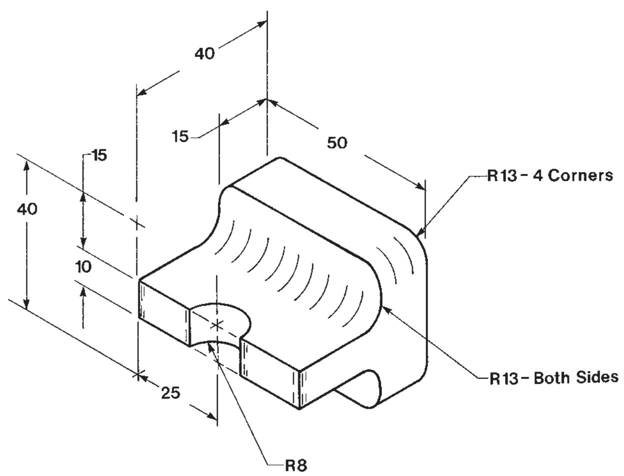



**Figure P4-9**
MILLIMETERS

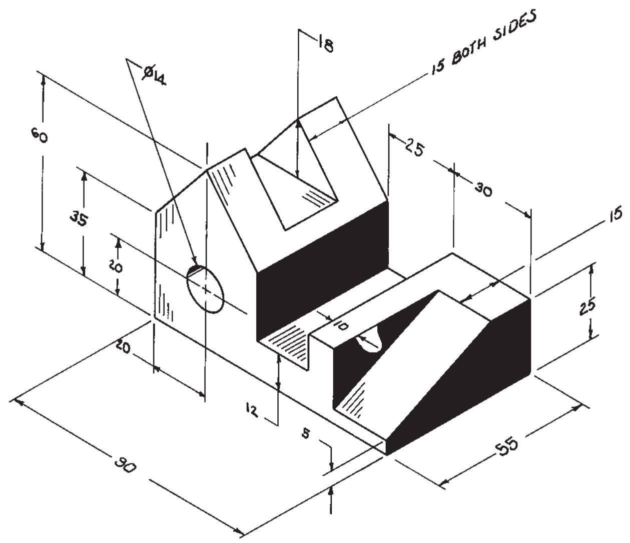



**Figure P4-10**
MILLIMETERS

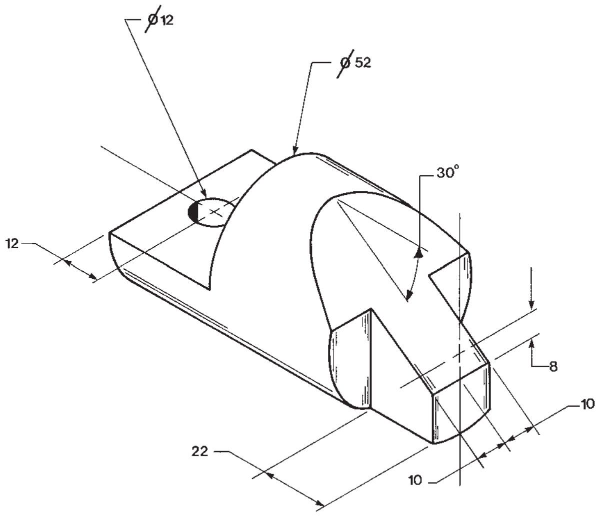



**Figure P4-11**
MILLIMETERS

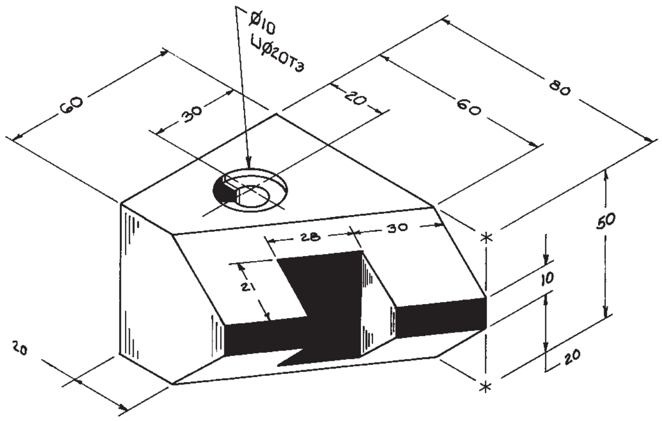
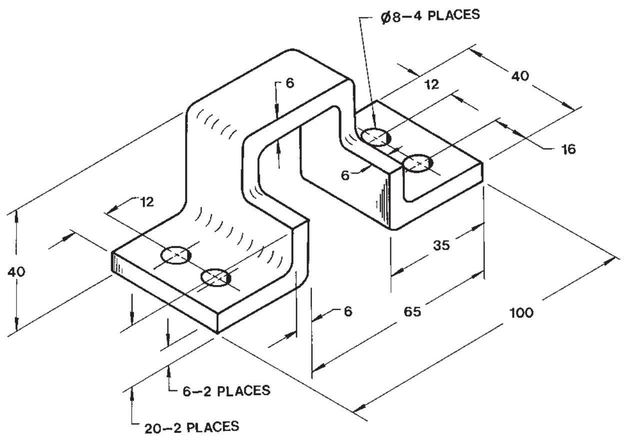
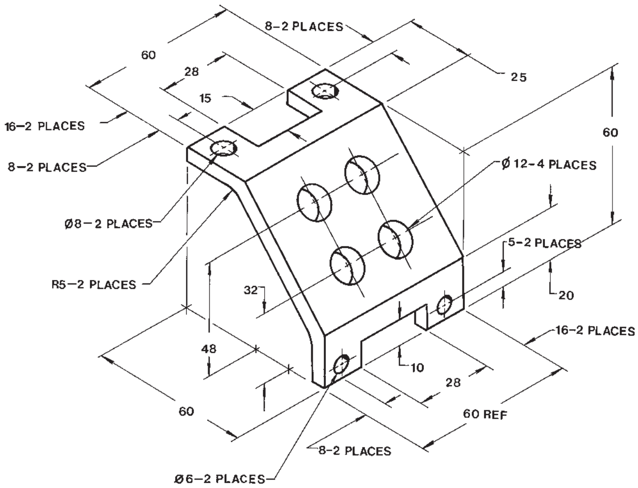



**Figure P4-12**
MILLIMETERS



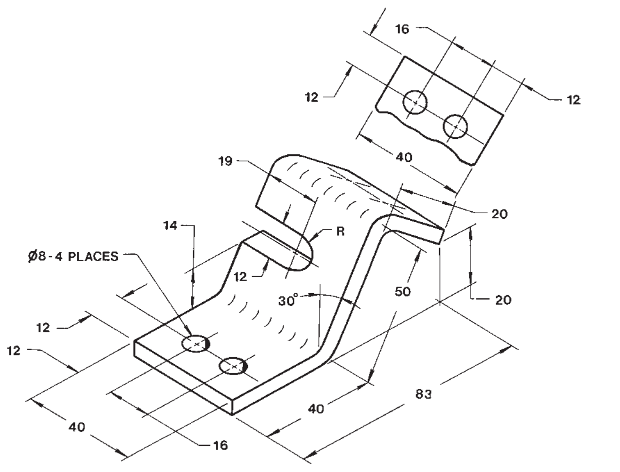

NOTE: ALL FILLETS AND ROUNDS:R3

**Figure P4-13**
MILLIMETERS




MATL 5 THK

**Figure P4-14**
MILLIMETERS




MATL 5 THK

ALL INSIDE BEND RAD 5

**Figure P4-15**
MILLIMETERS

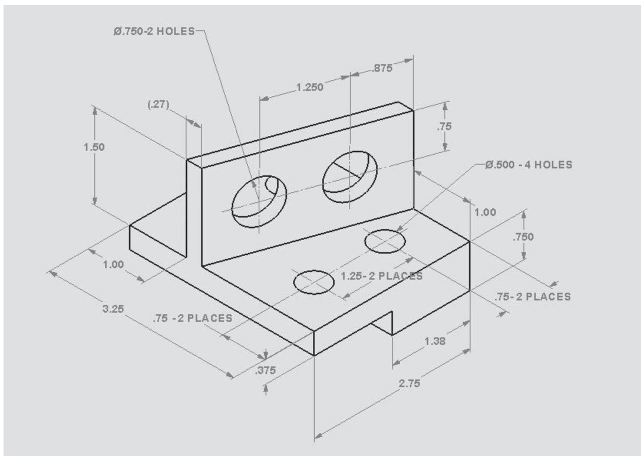



**Figure P4-16**
INCHES

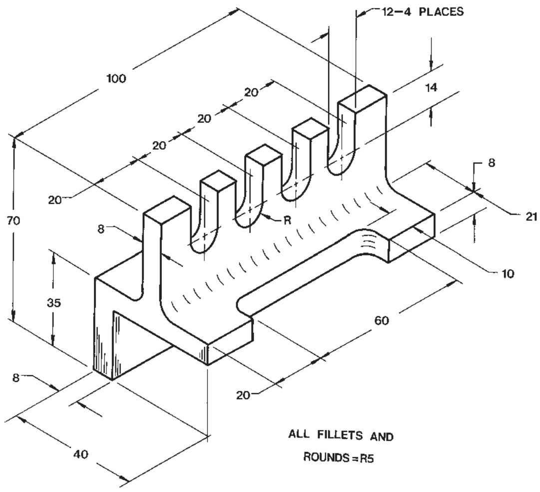



**Figure P4-17**
MILLIMETERS

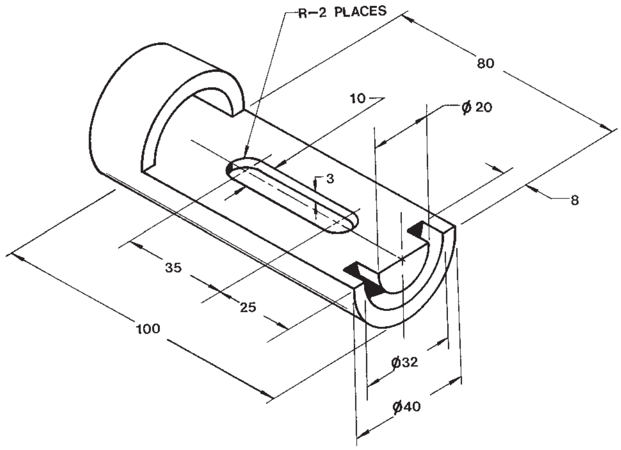



**Figure P4-18**
MILLIMETERS

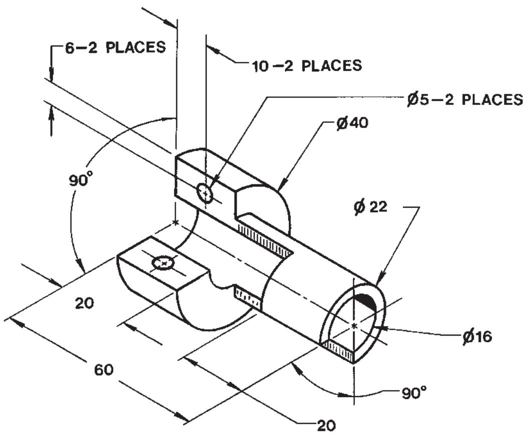



**Figure P4-19**
MILLIMETERS

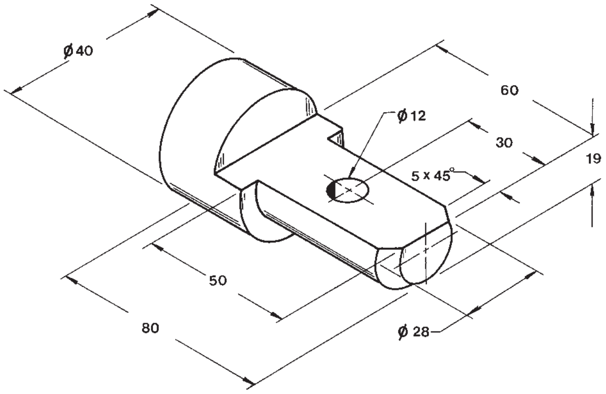



**Figure P4-20**
MILLIMETERS

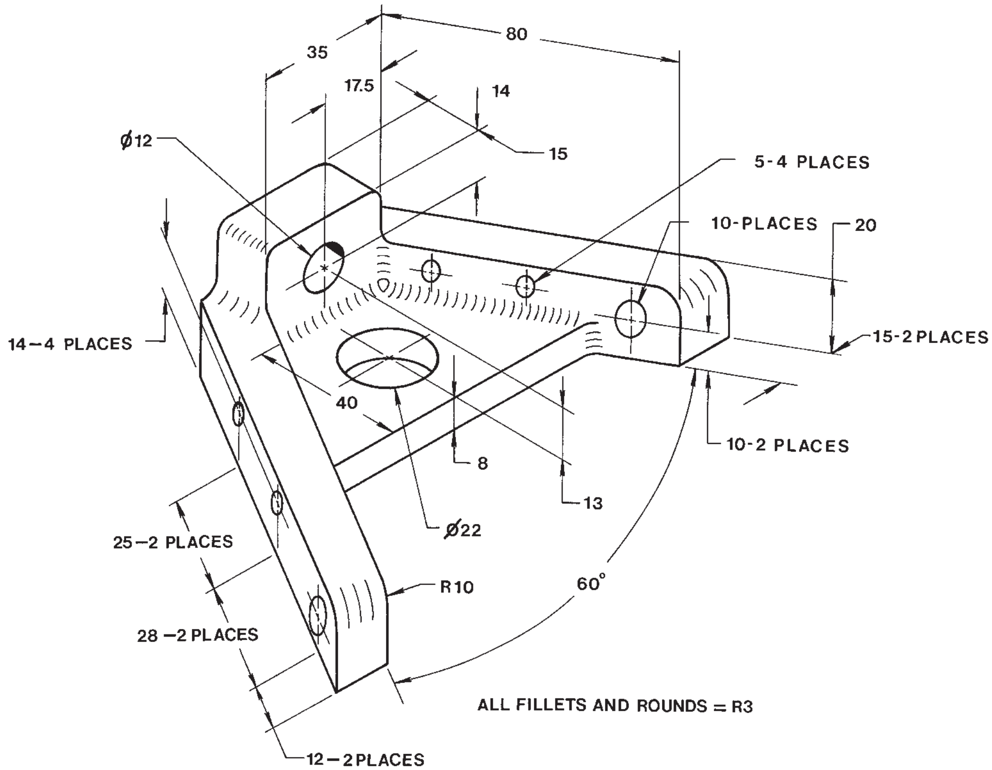



**Figure P4-21**
MILLIMETERS

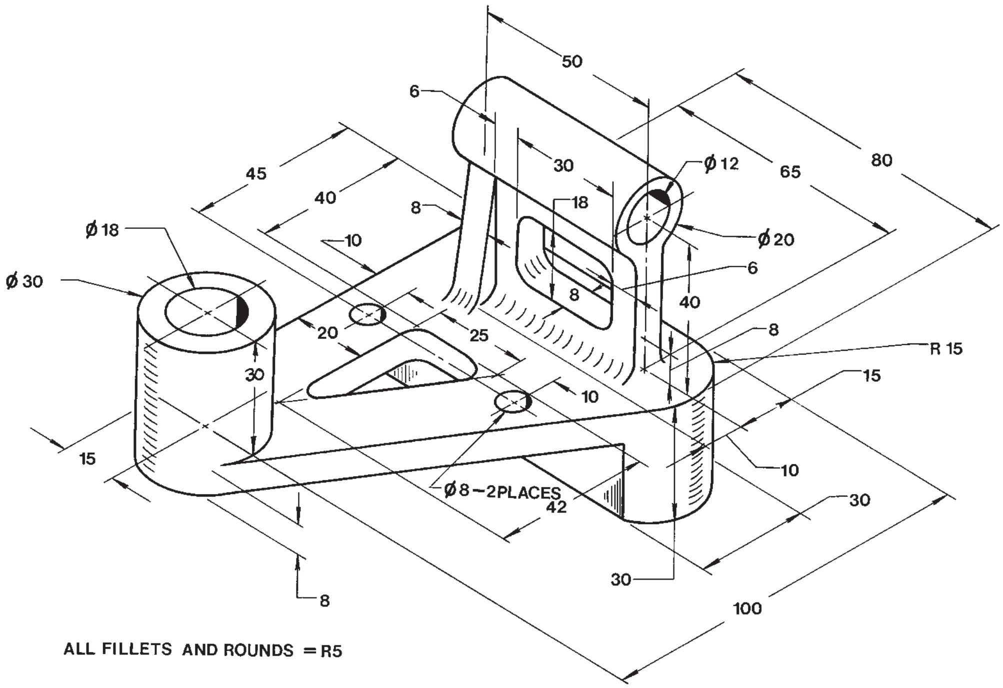



**Figure P4-22**
MILLIMETERS

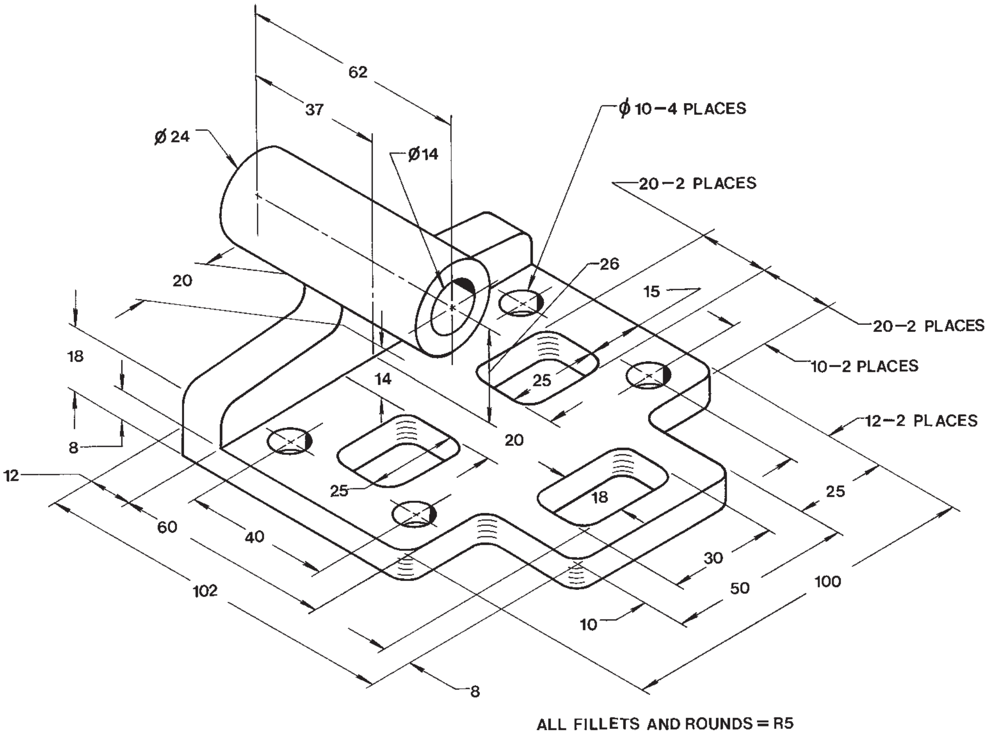



**Figure P4-23**
MILLIMETERS

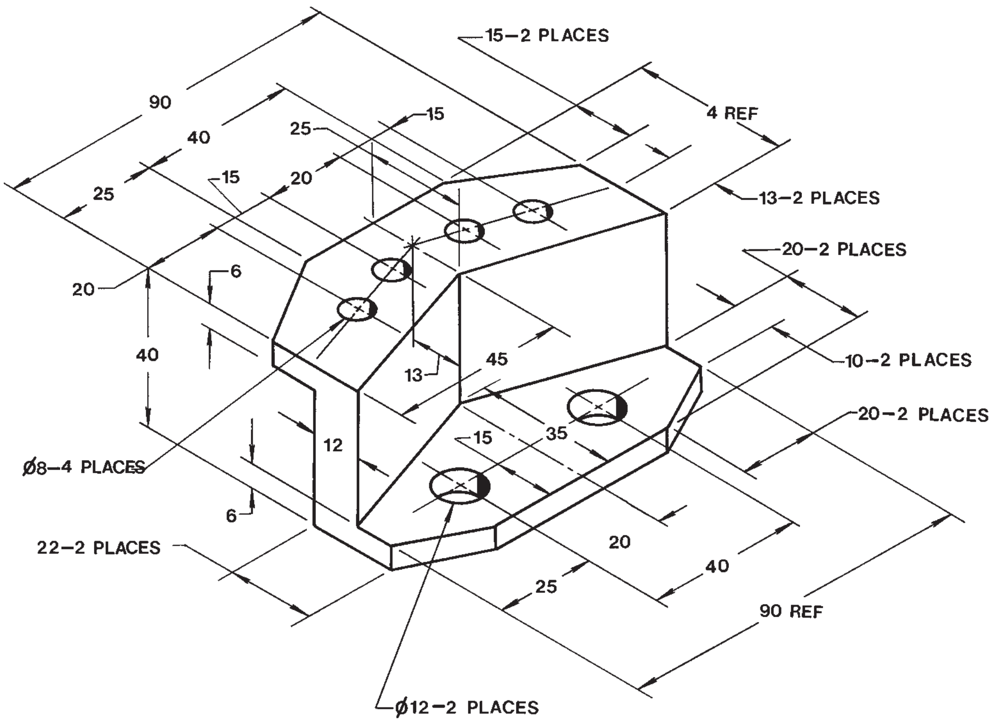



**Figure P4-24**
MILLIMETERS

# Project 4-2

Draw at least two orthographic views and one auxiliary view of each of the objects shown in Figures P4-25 through P4-36.

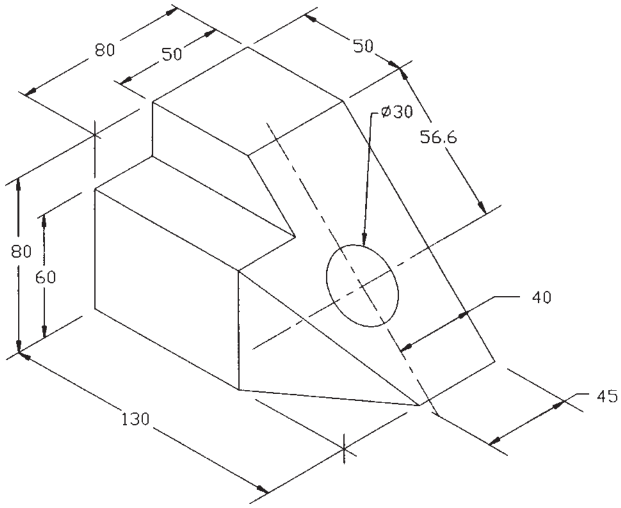



**Figure P4-25**
MILLIMETERS

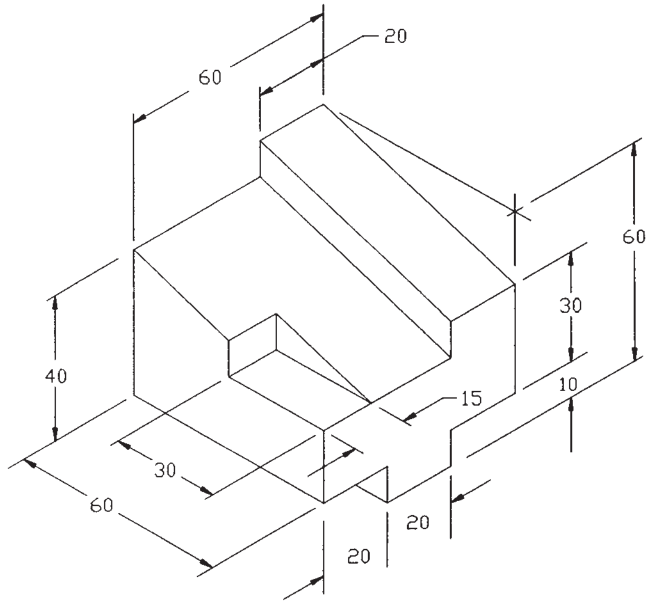



**Figure P4-26**
MILLIMETERS

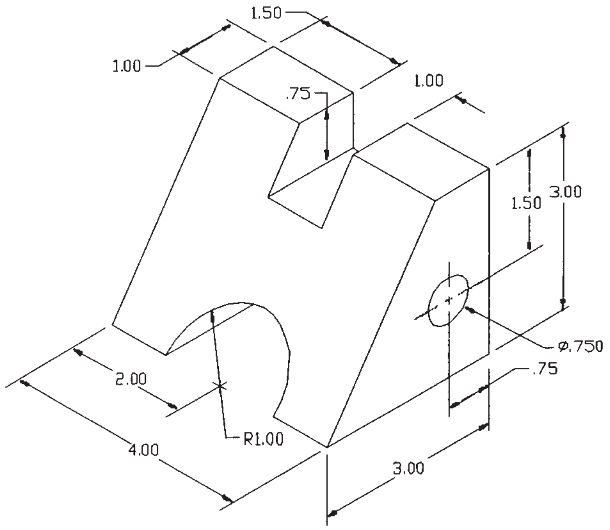



**Figure P4-27**
INCHES

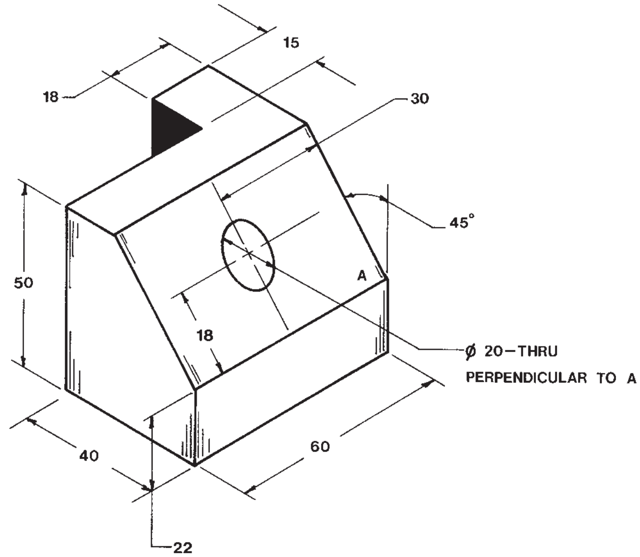



**Figure P4-28**
MILLIMETERS

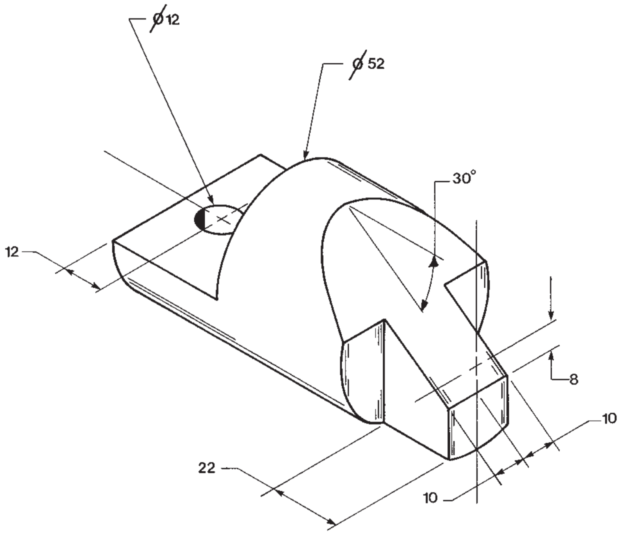



**Figure P4-29**
MILLIMETERS

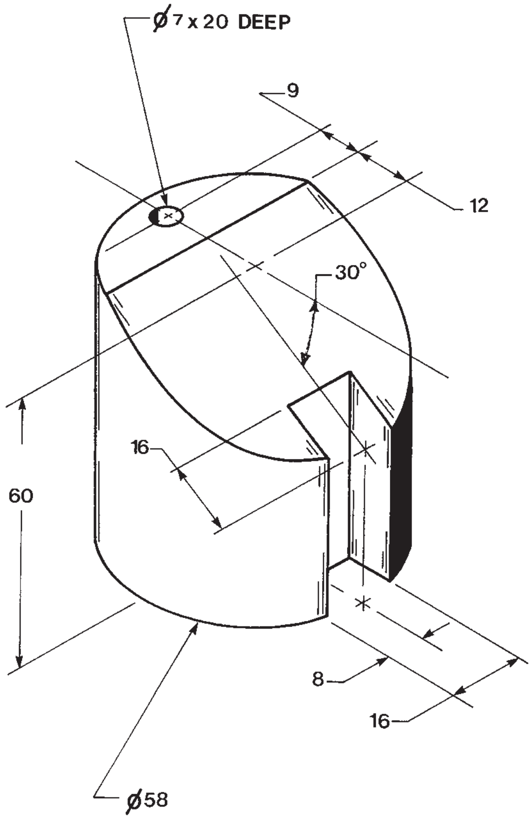



**Figure P4-30**
MILLIMETERS

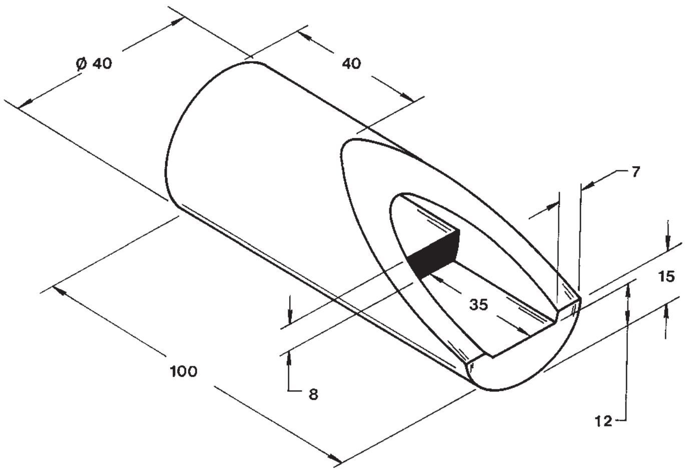



**Figure P4-31**
MILLIMETERS

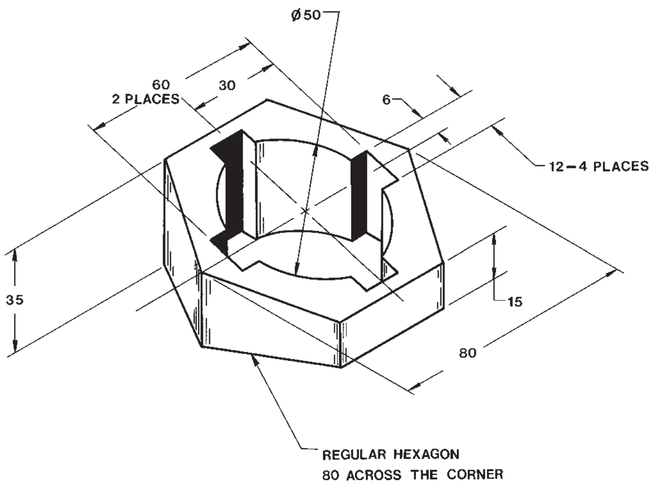



**Figure P4-32**
MILLIMETERS

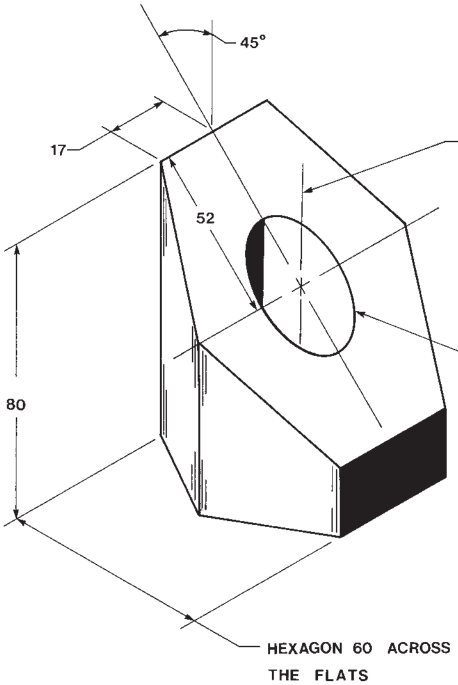



**Figure P4-33**
MILLIMETERS

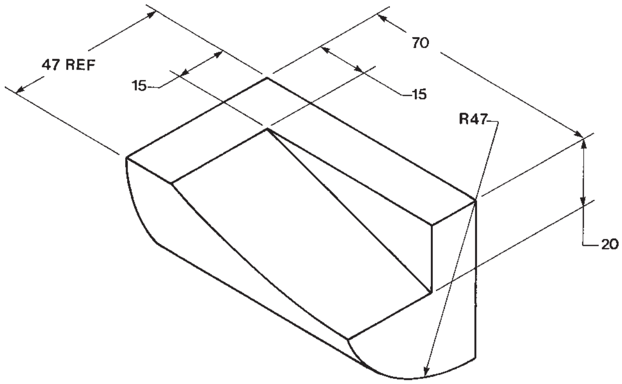



**Figure P4-34**
MILLIMETERS

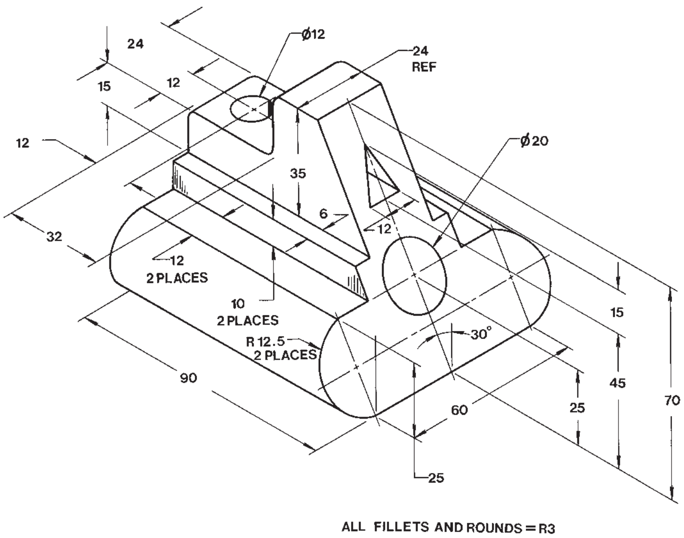



**Figure P4-35**
INCHES

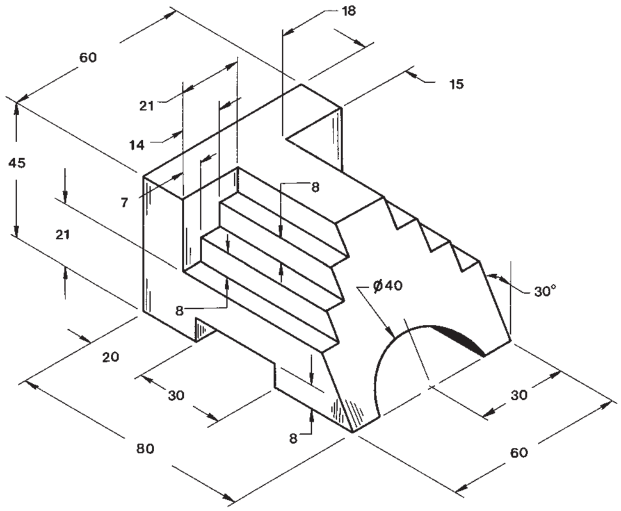



**Figure P4-36**
MILLIMETERS

# Project 4-3

Define the true shape of the oblique surfaces in each of the objects shown in Figures P4-37 through P4-40.

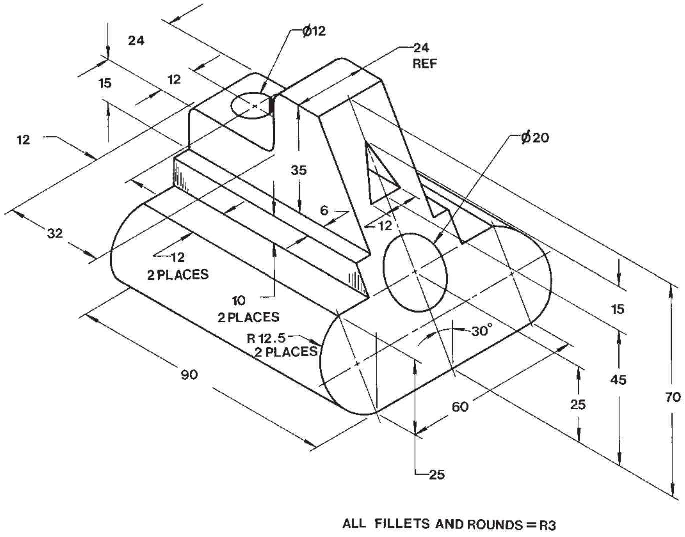



**Figure P4-37**
INCHES

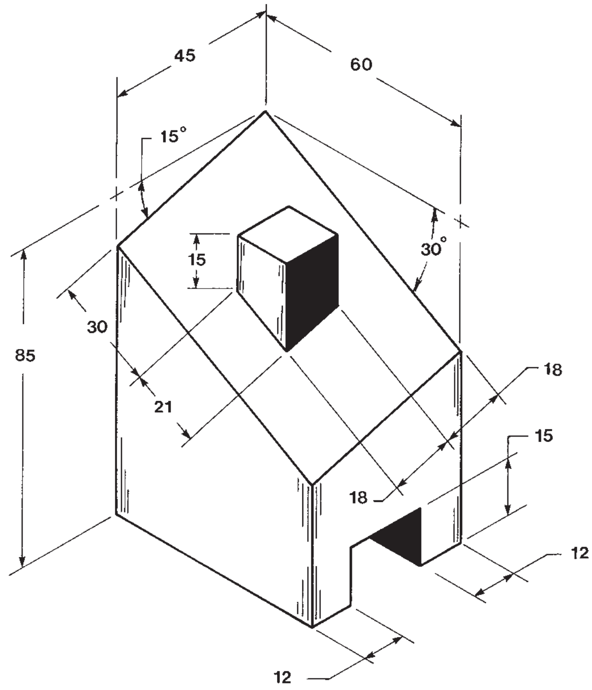



**Figure P4-38**
MILLIMETERS

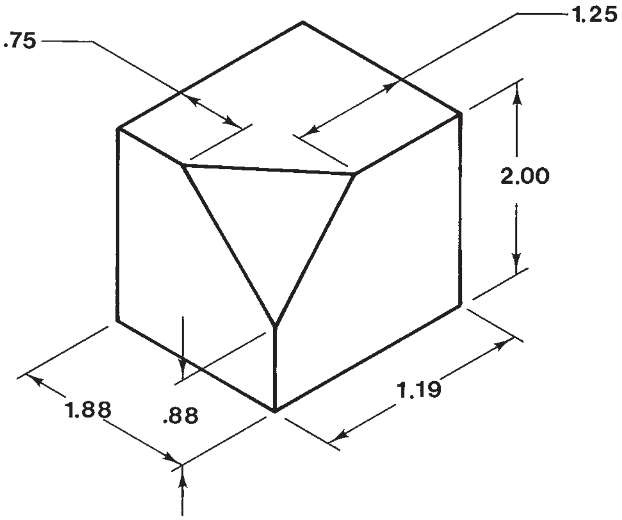



**Figure P4-39**
INCHES

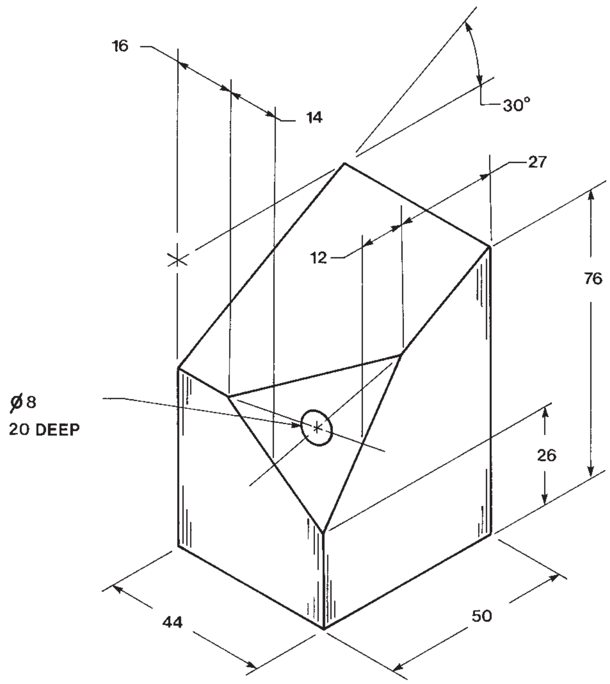



**Figure P4-40**
MILLIMETERS

# Project 4-4

Draw each of the objects shown in Figures P4-41 through P4-44 as a model, then draw a front view and an appropriate section view of each.

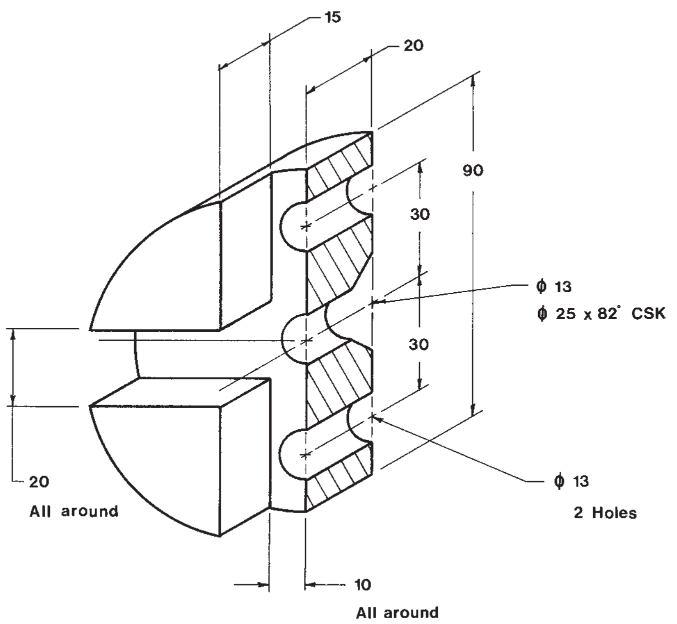



Figure P4-41
MILLIMETERS

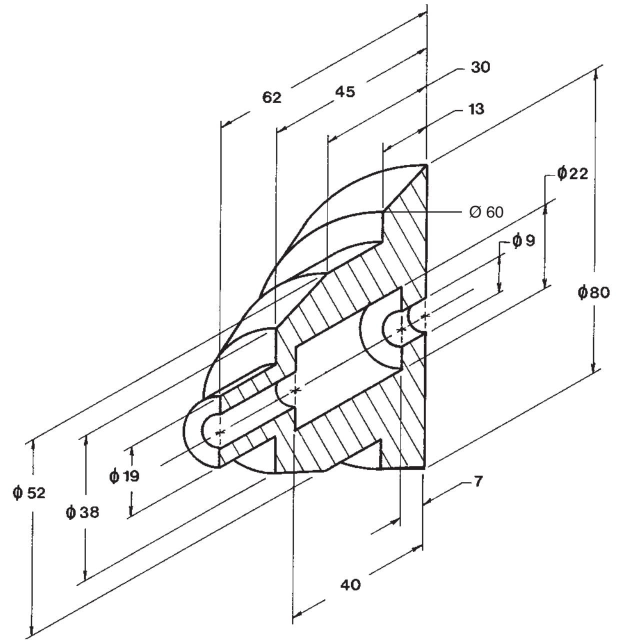



Figure P4-42
MILLIMETERS

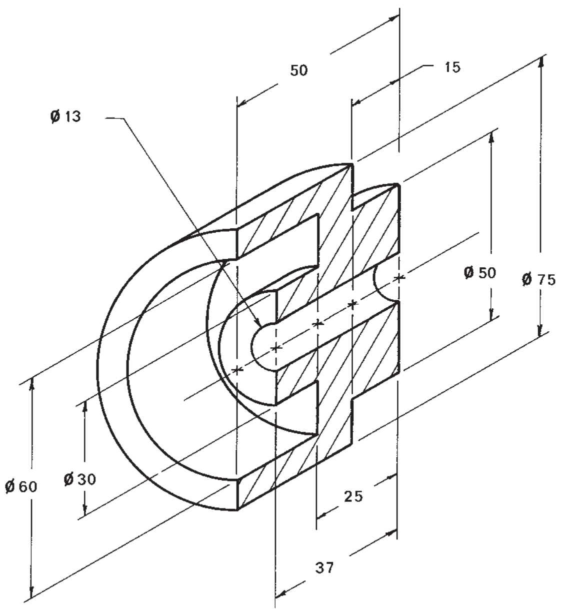



Figure P4-43
MILLIMETERS

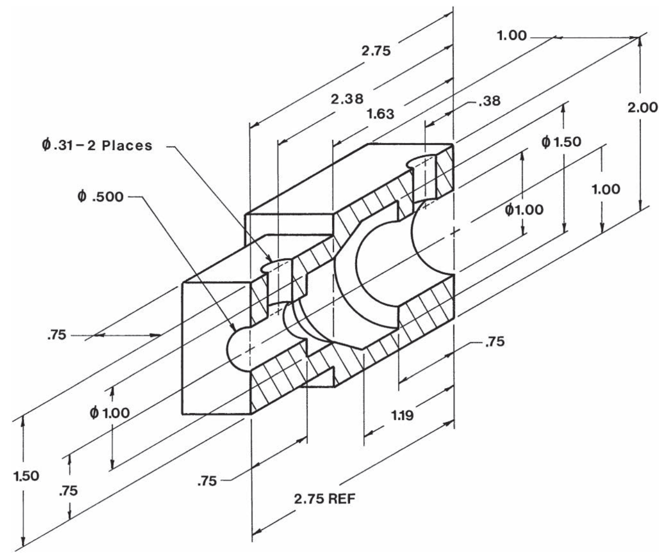



Figure P4-44
INCHES

# Project 4-5

Draw at least one orthographic view and the indicated section view for each object shown in Figures P4-45 through P4-50.

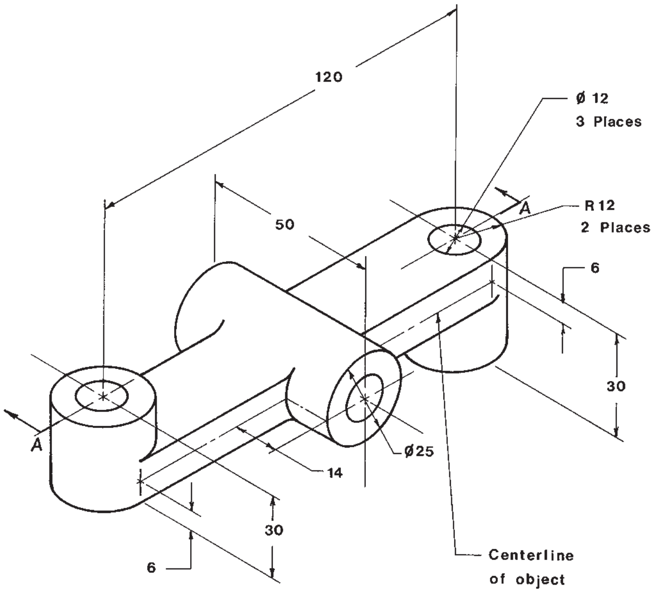



**Figure P4-45**
MILLIMETERS

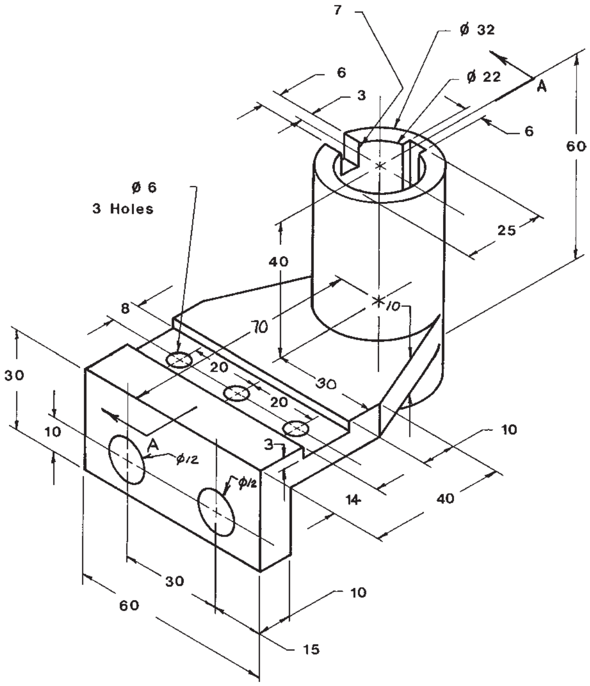



**Figure P4-46**
MILLIMETERS

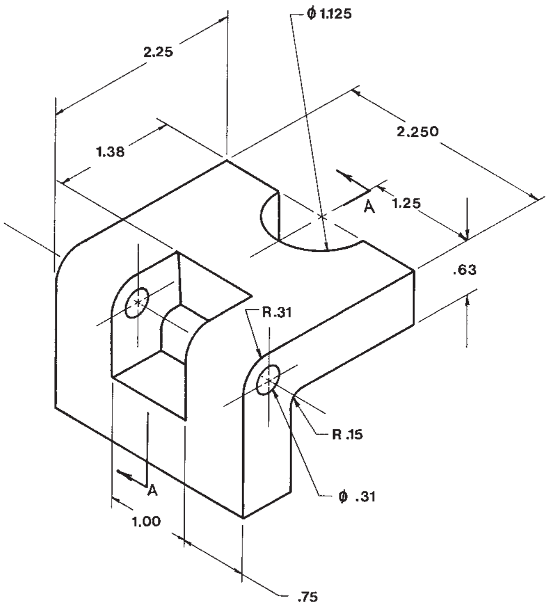



**Figure P4-47**
INCHES

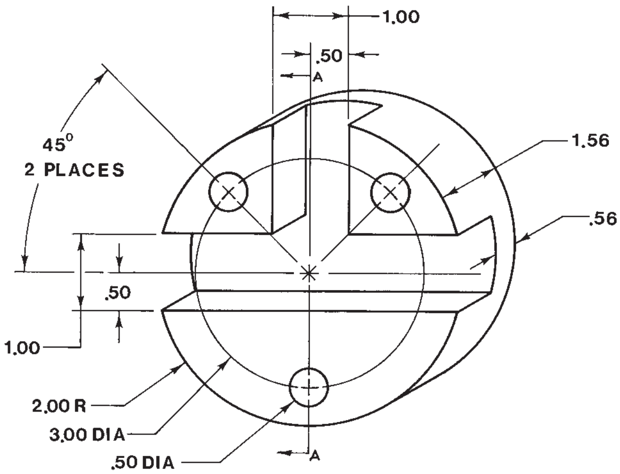



**Figure P4-48**
INCHES

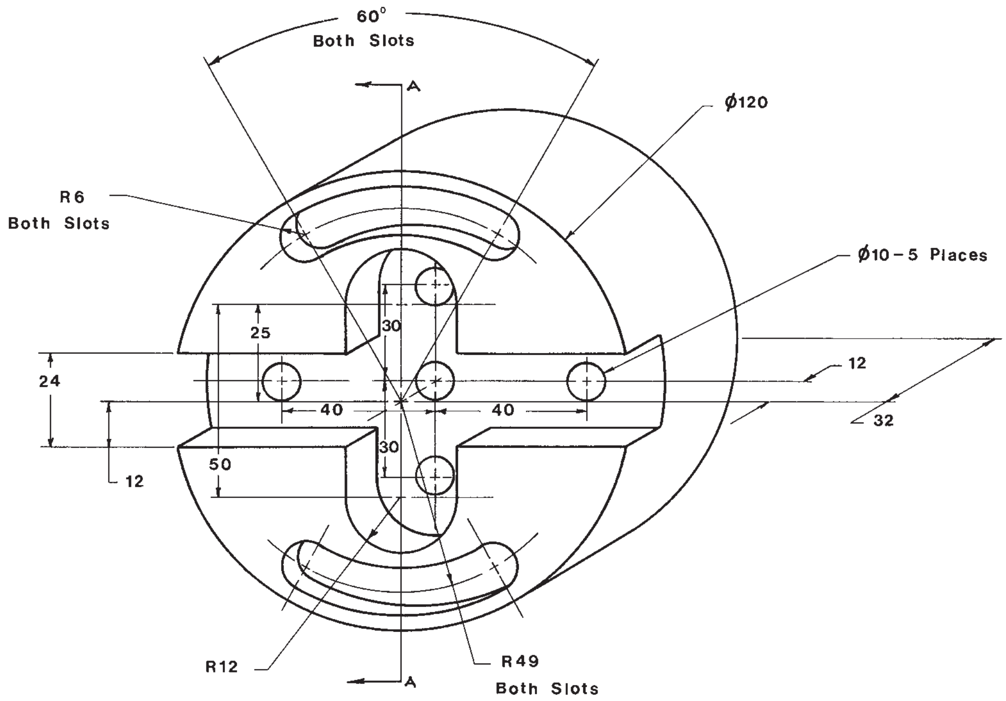



**Figure P4-49**
MILLIMETERS

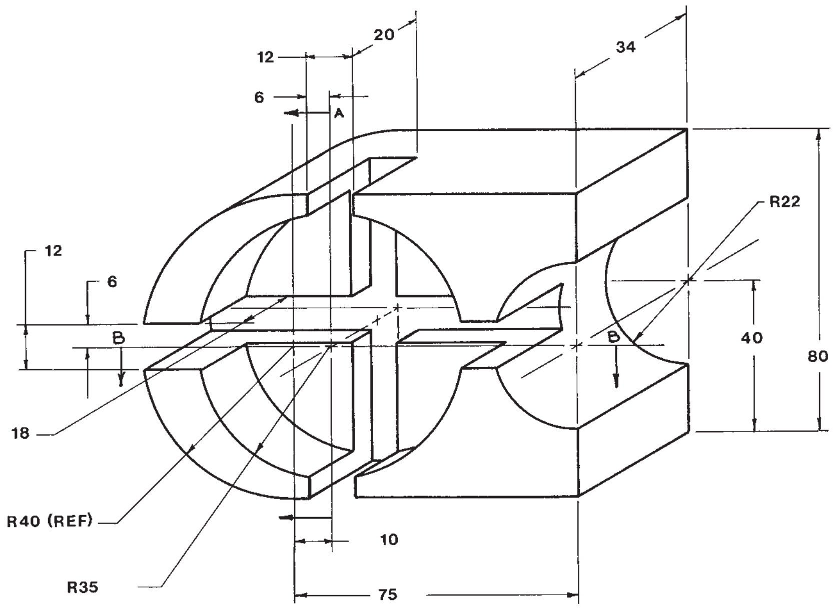



**Figure P4-50**
MILLIMETERS

# Project 4-6

Given the orthographic views in Figures P4-51 and P4-52, draw a model of each, then draw the given orthographic views and the appropriate section views.

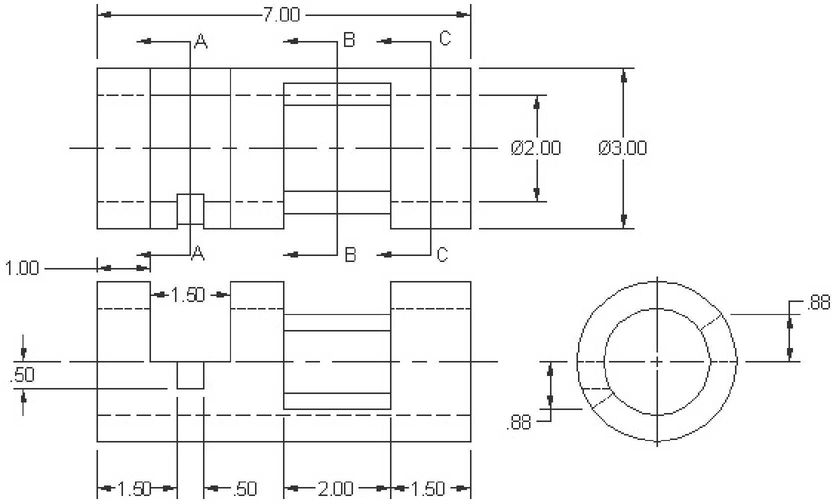



**Figure P4-51**
INCHES

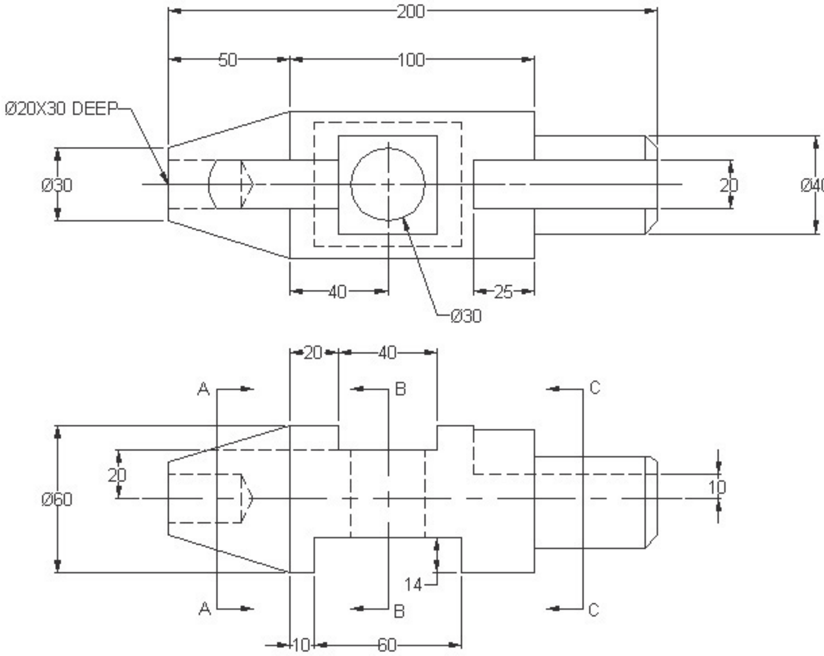



**Figure P4-52**
MILLIMETERS

# Project 4-7

Draw a 3D model and a set of multiviews for each object shown in Figures P4-53 through P4-60.

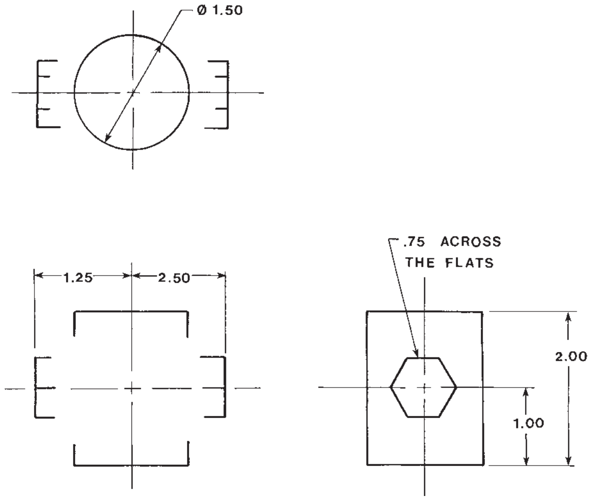



**Figure P4-53**
INCHES

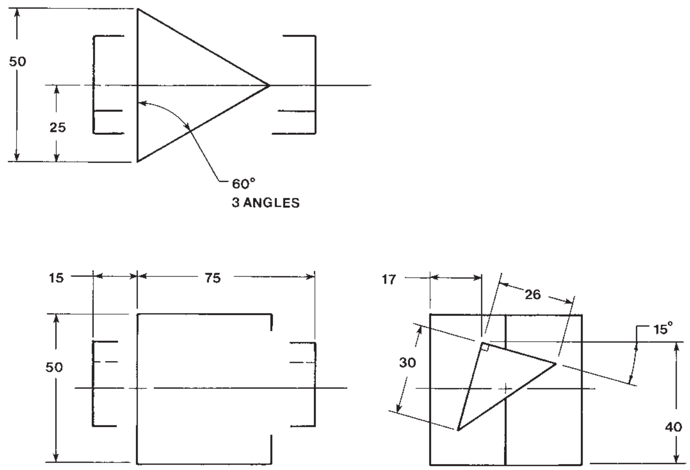



**Figure P4-54**
MILLIMETERS

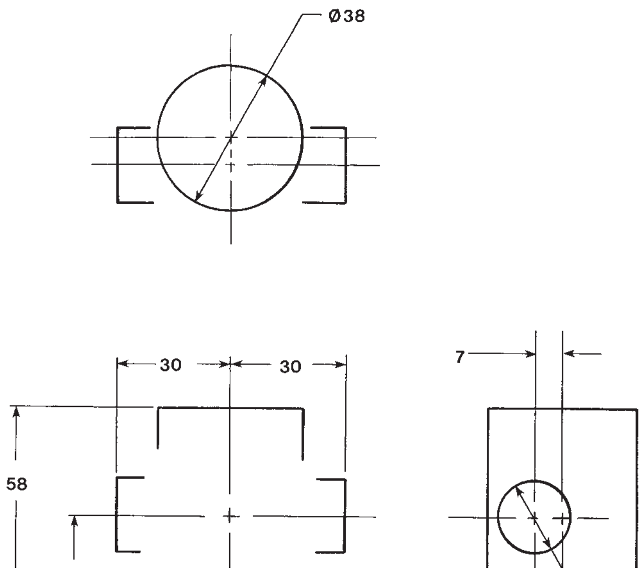



**Figure P4-55**
MILLIMETERS

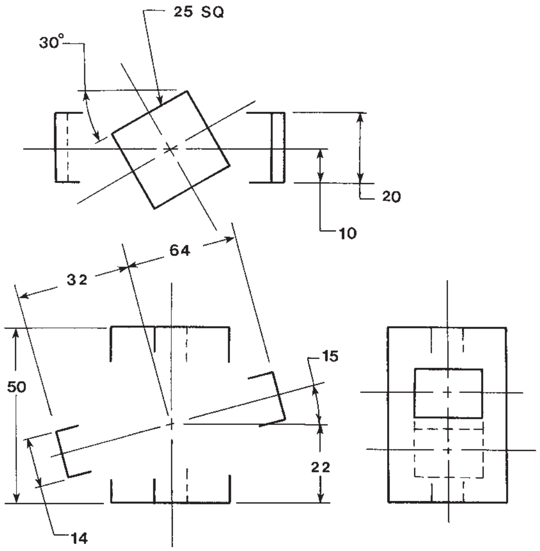



**Figure P4-56**
MILLIMETERS

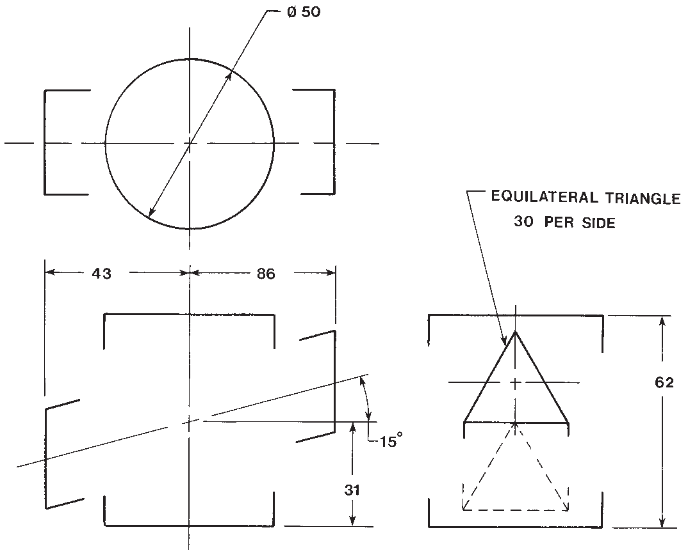



**Figure P4-57**
MILLIMETERS

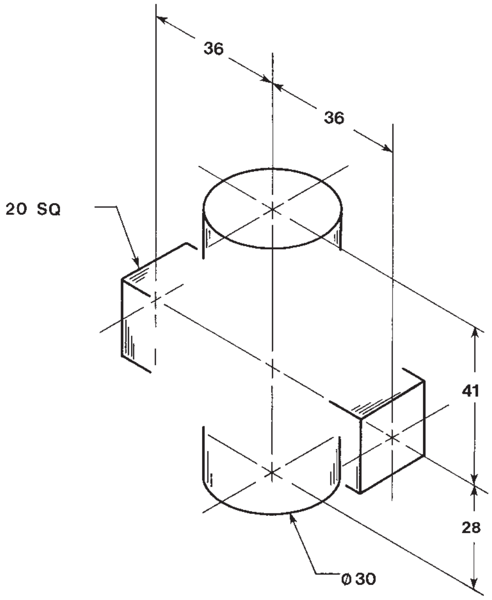



**Figure P4-58**
MILLIMETERS

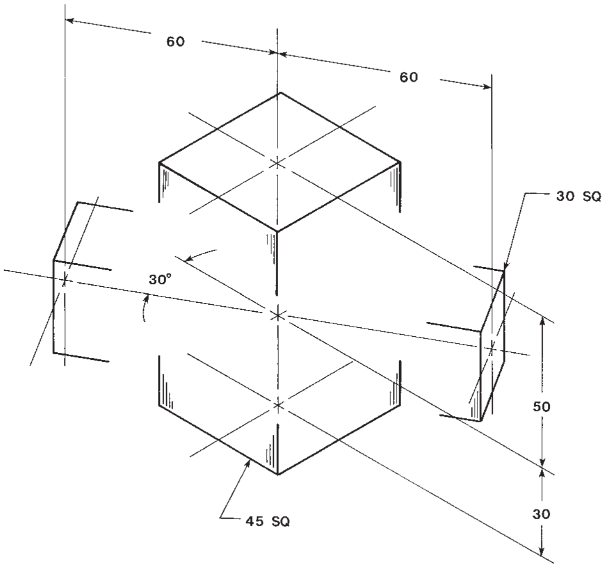



**Figure P4-59**
MILLIMETERS

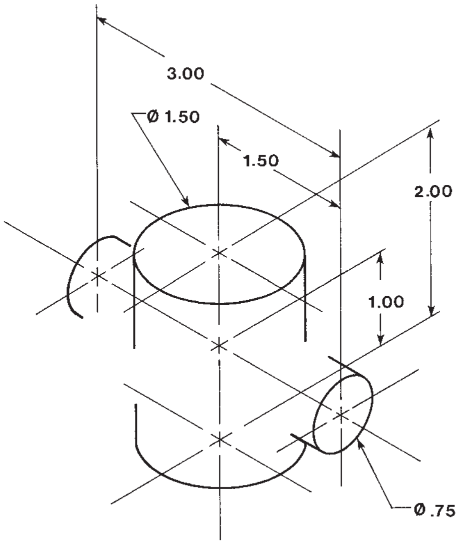



**Figure P4-60**
INCHES

# Project 4-8

Figures P4-61 through P4-66 are orthographic views. Draw 3D models from the given views. The hole pattern defined in Figure P4-61 also applies to Figure P4-62.

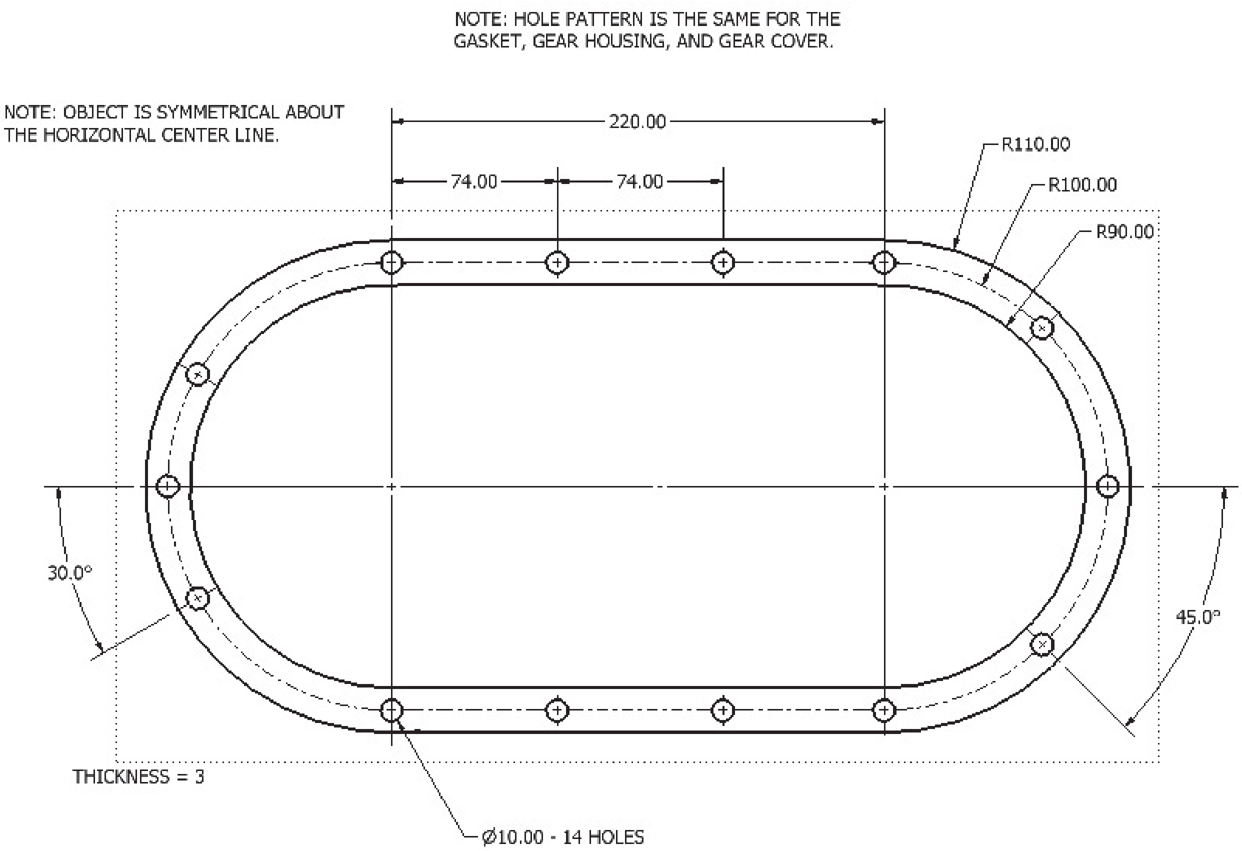



**Figure P4-61**
MILLIMETERS

**Figure P4-62**

MILLIMETERS

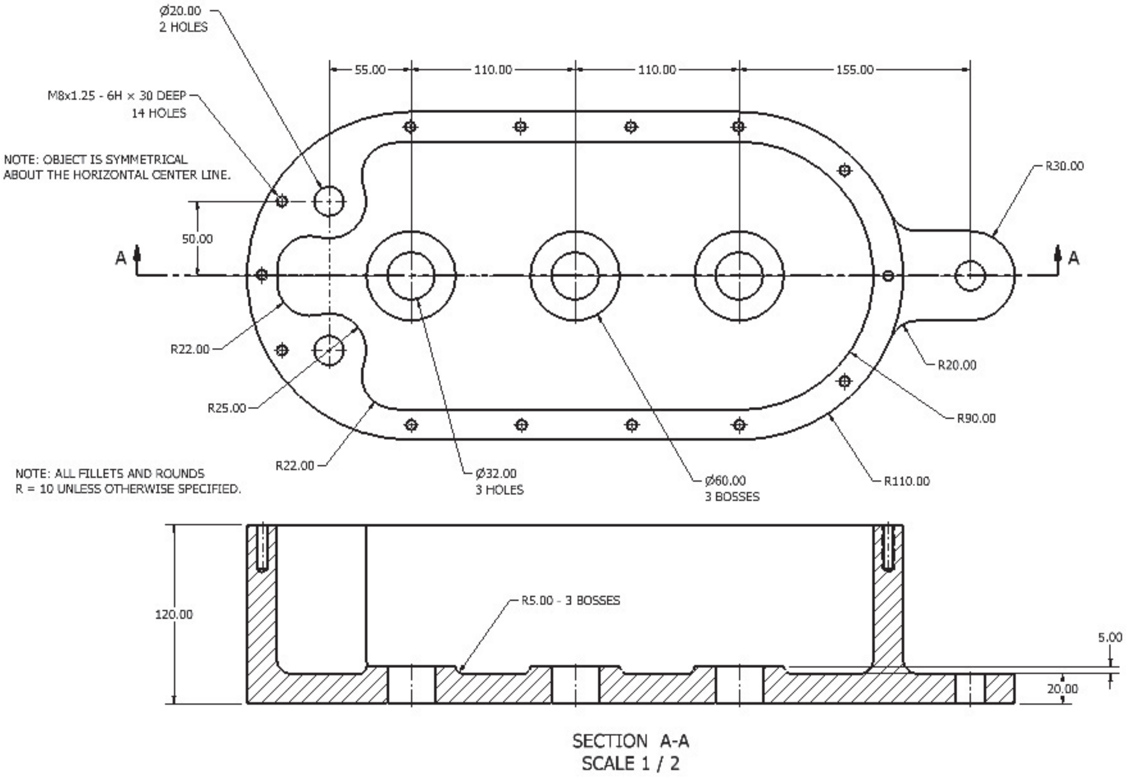
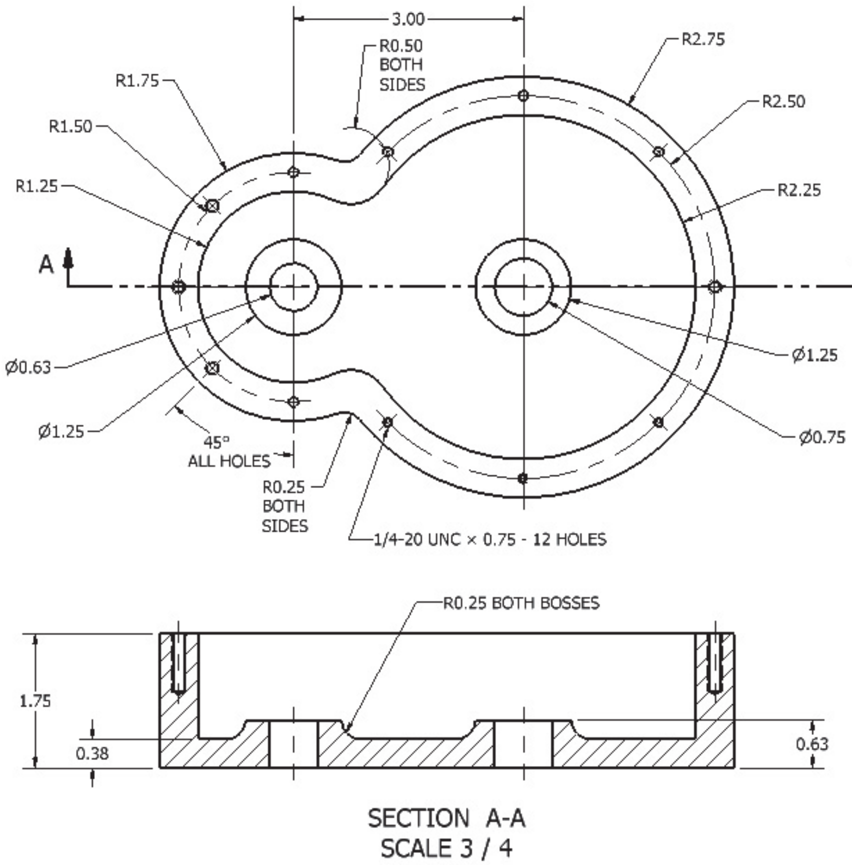
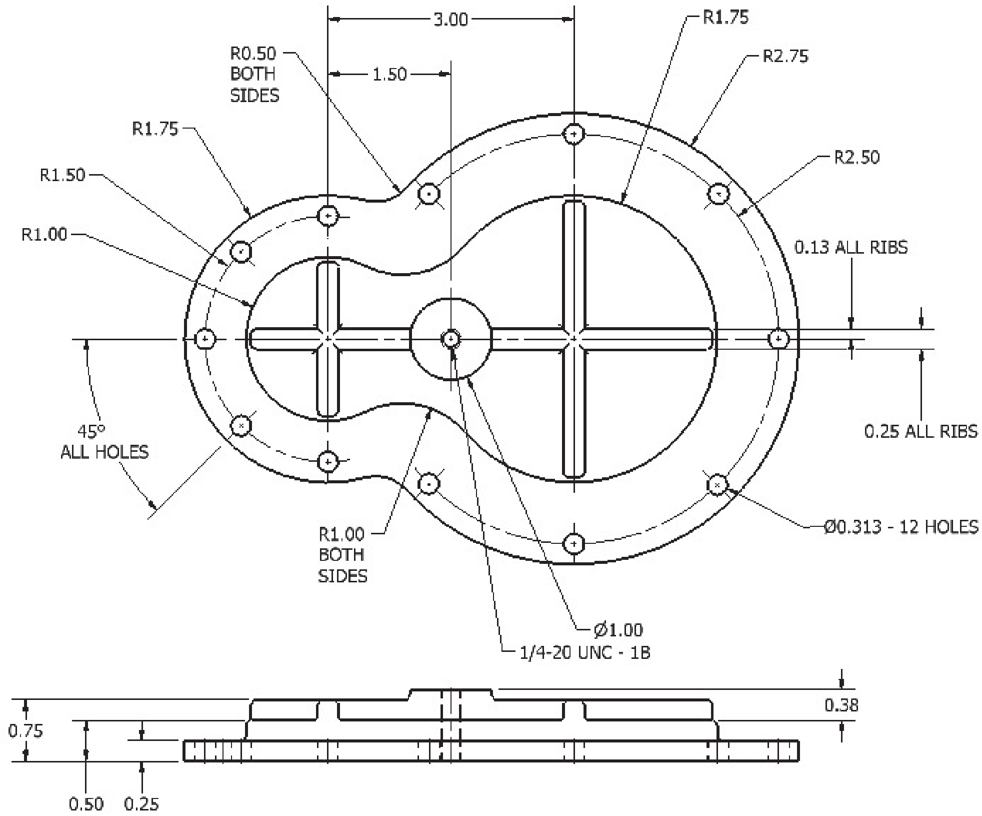



**Figure P4-63**

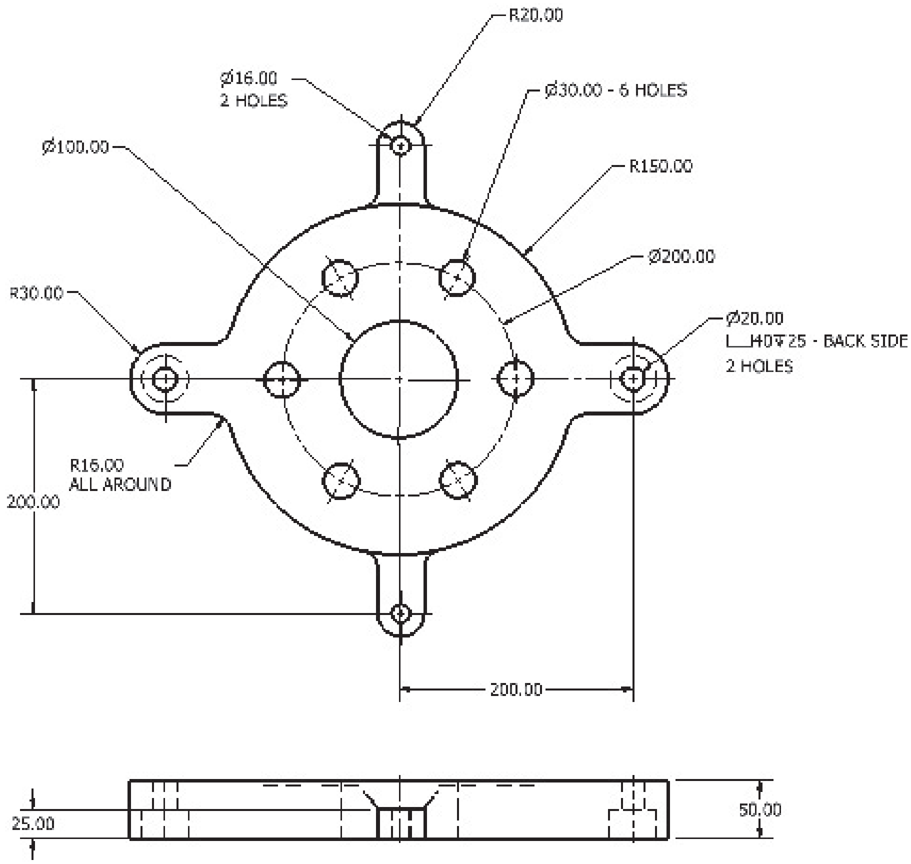
INCHES



## Figure P4-64
INCHES




## Figure P4-65
MILLIMETERS



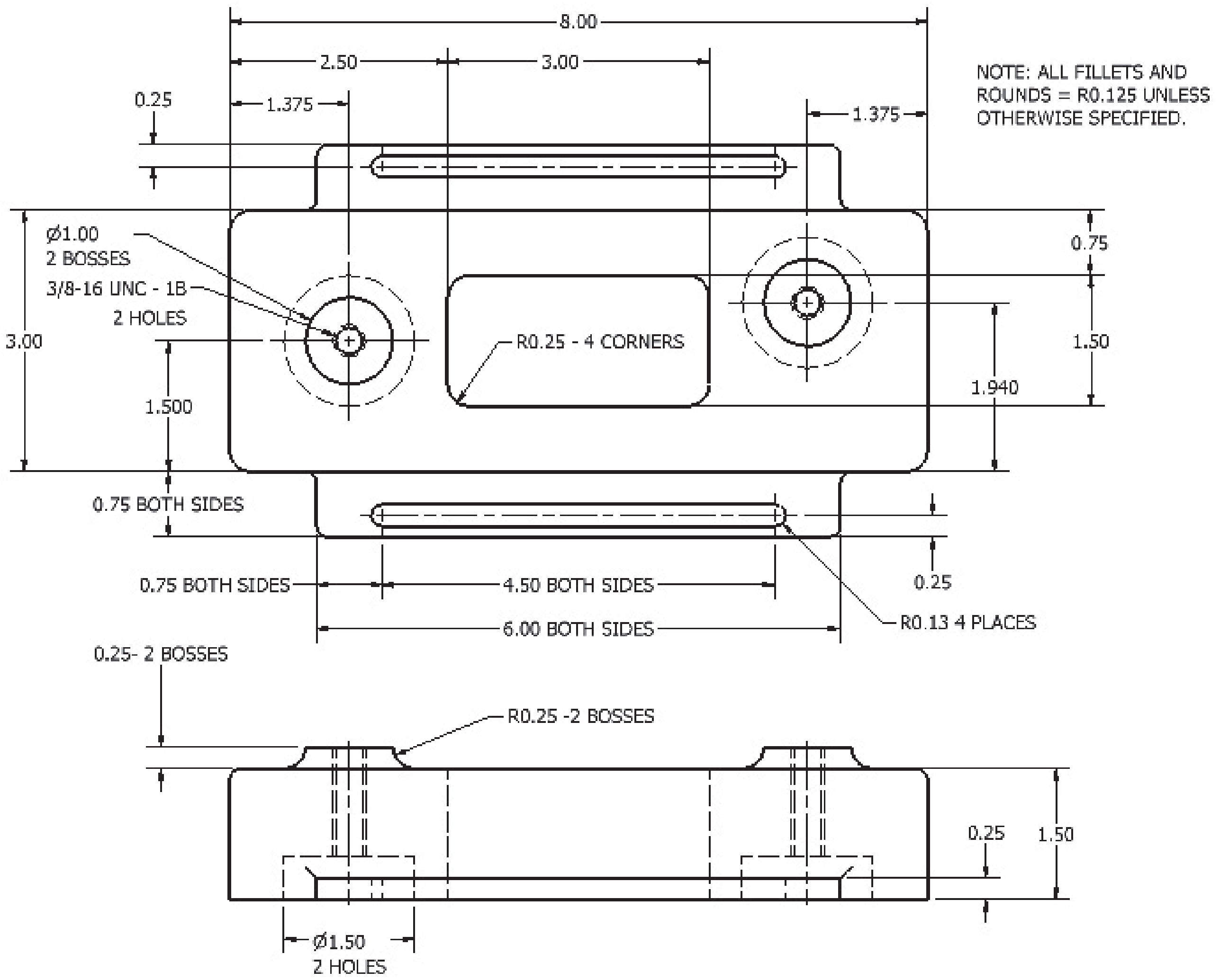



**Figure P4-66**
INCHES

# Project 4-9

Figures P4-67 through P4-71 are presented using first-angle projection and ISO conventions.

A. Create a solid model from the given orthographic views.

B. Draw front, top, and right-side orthographic views of the objects using third-angle projection and ANSI conventions.

**Figure P4-67**
MILLIMETERS

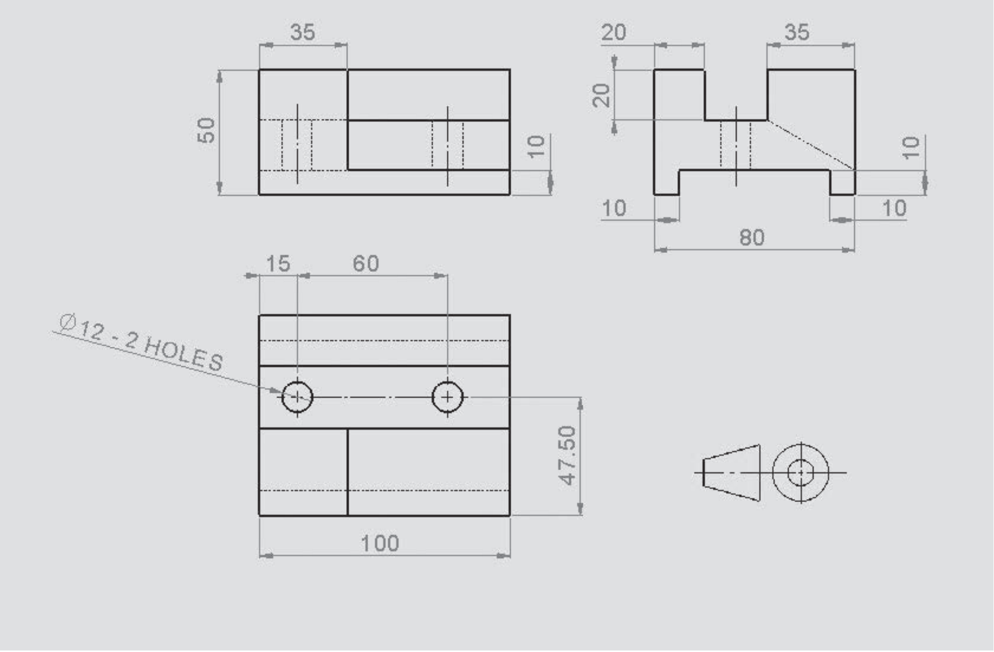



**Figure P4-68**
MILLIMETERS

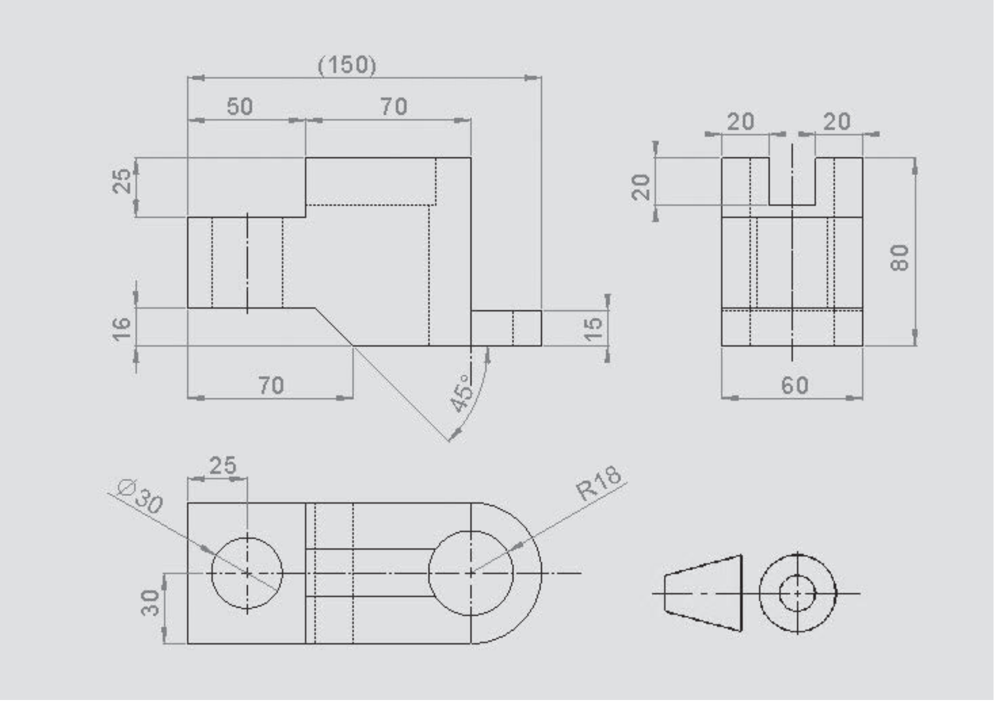


Figure P4-69
MILLIMETERS

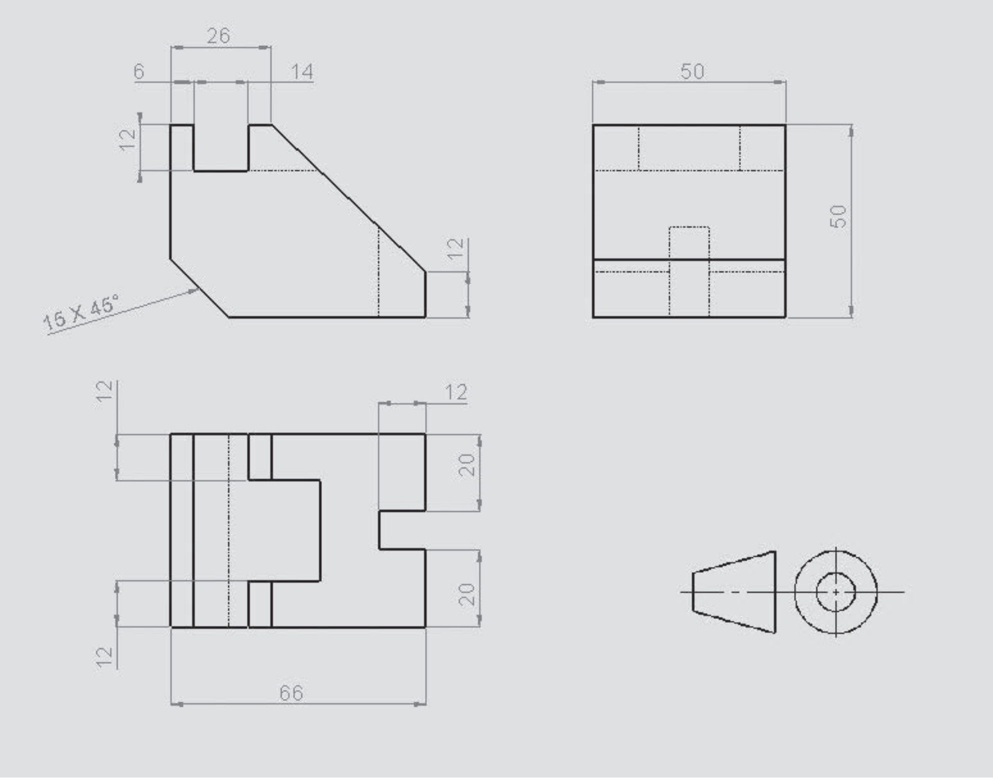



Figure P4-70
MILLIMETERS

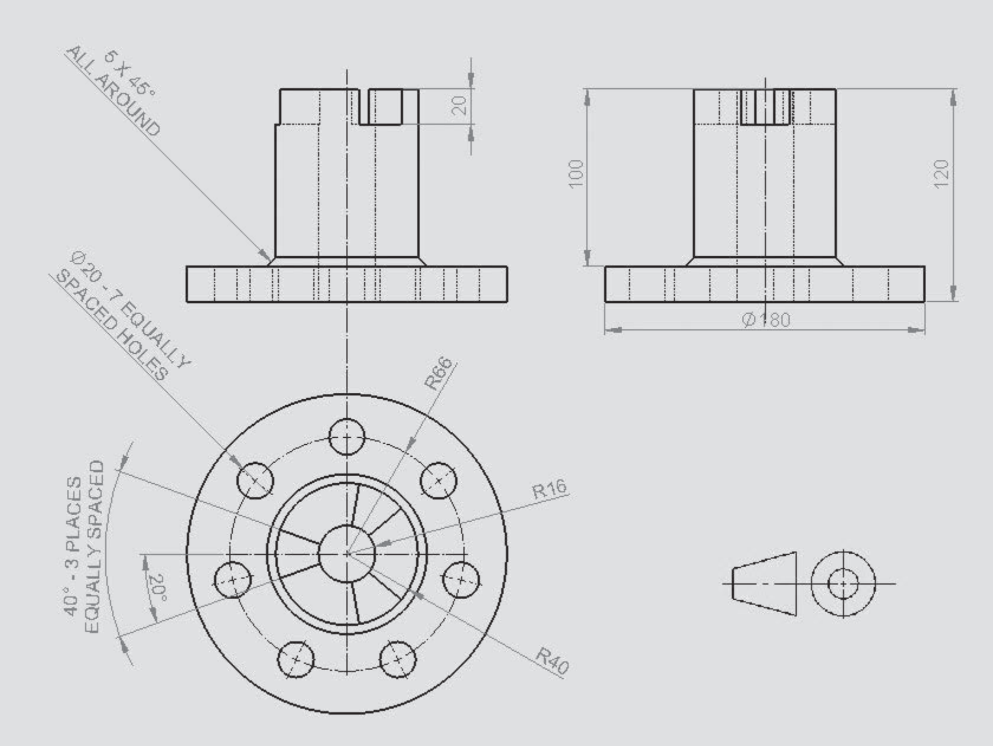
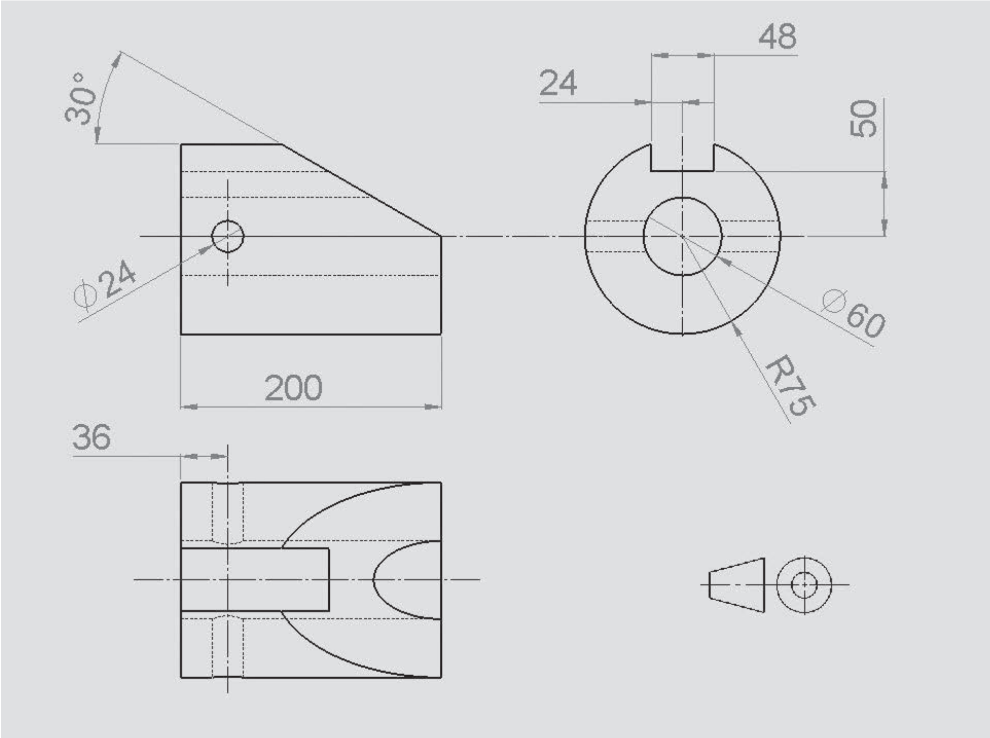


**Figure P4-71**
MILLIMETERS



# Index

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## H



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## K



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## M

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