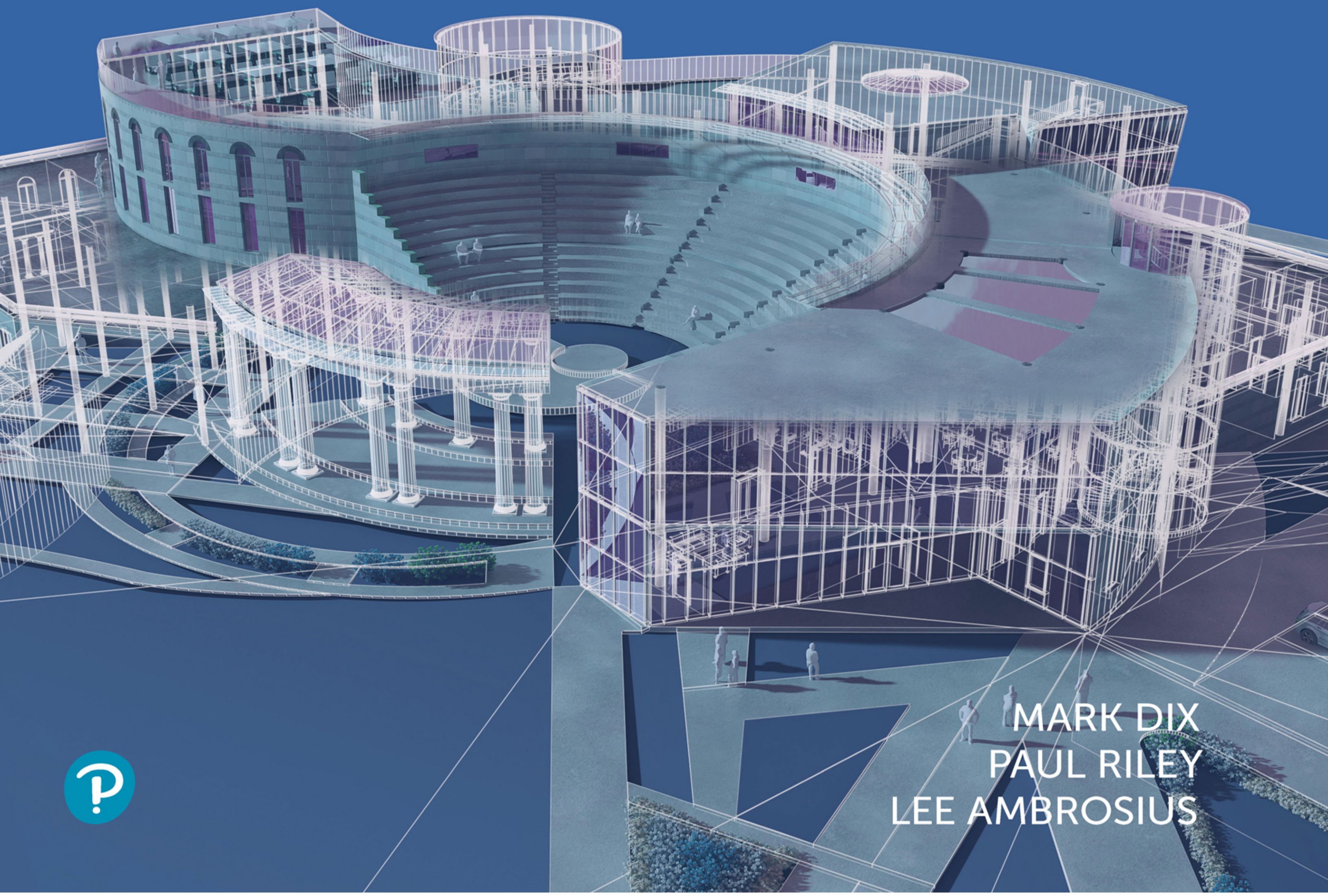
DISCOVERING
AutoCAD® 2024



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FREE SAMPLE CHAPTER

Discovering AutoCAD® 2024

Mark Dix

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Pearson

Discovering AutoCAD® 2024

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Preface

Get Active with *Discovering AutoCAD® 2024*

Designed for introductory AutoCAD users, *Discovering AutoCAD 2024* offers a hands-on, activity-based approach to the use of AutoCAD as a drafting tool—complete with techniques, tips, shortcuts, and insights designed to increase efficiency. Topics and tasks are carefully grouped to lead students logically through the AutoCAD command set, with the level of difficulty increasing steadily as skills are acquired through experience and practice. Straightforward explanations focus on what is relevant to actual drawing procedures, and illustrations show exactly what to expect on the computer screen when steps are correctly completed. Each chapter ends with drawing exercises that assess and reinforce the student's understanding of the material.

Features

The book uses a consistent format for each chapter that includes the following:

- Chapter Objectives and Introduction
- Exercises that introduce new commands and techniques
- Exercise instructions clearly set off from the text discussion
- Lots of illustrations with drawings and screenshots
- Twenty end-of-chapter Review Questions
- Four to eight realistic engineering drawing problems—fully dimensioned working drawings

High-quality working drawings include a wide range of applications that focus on mechanical drawings but also include architectural, civil, plumbing, general, and electrical drawings. Appendix A contains 21 drawing projects for additional review and practice. Appendixes B, C, and D cover material not required for drawing practice but highly relevant for any beginning CAD professional. These include information on customization features, basic programming procedures, and a summary of Autodesk cloud-based and file-sharing features.

Acknowledgments

The authors thank the following reviewers for their feedback: John Irwin, Michigan Technological University; Tony Graham, North Carolina A&T State University; Beverly Jaeger, Northeastern University; Daniel McCall, Amarillo College; and Susan Freeman, Northeastern University. Lastly, we would like to thank Jon Page (and his family) for his time in reviewing the content changes made in this edition of the book.

From Lee Ambrosius: I would like to thank my family for being by my side throughout the many stages of my career and during the writing of this book. Along with my family, I would like to give special thanks to my

instructors (Gary Magee, Kenneth Schulz, and Tricia Croyle) of the architectural program at Northeast Wisconsin Technical College (NWTC), where it all began. Without them, my career would likely be very different today.

Features New to This Edition

1. Updated to reflect the latest changes to AutoCAD with the 2024 release
2. Updated illustrations representing the newest AutoCAD interface
3. Coverage of the workflow of measuring objects in Chapter 6
4. New sections on counting and replacing blocks in Chapter 10
5. Expanded Appendix D with coverage of design review workflows

Style Conventions in *Discovering AutoCAD® 2024*

Text Element	Example
Key Terms—Boldface and italic on first mention (first letter lowercase, as it appears in the body of the text). Brief definition in margin alongside first mention. Full definition in Glossary available at peachpit.com/Discoverautocad2024.	Views are created by placing ***viewport*** objects in the paper space layout.
AutoCAD commands—Bold and uppercase.	Start the **LINE** command.
Ribbon and panel names, palette names, toolbar names, menu items, and dialog box names—Bold and follow capitalization convention in AutoCAD toolbar or pull-down menu. (Generally, the first letter is capitalized.)	The **Layer Properties Manager** palette
Panel tools, toolbar buttons, and dialog box controls/buttons/input items—Bold and follow the name of the item or the name shown in the AutoCAD tooltip.	Choose the **Line** tool from the **Draw** panel. Choose the **Symbols and Arrows** tab in the **Modify Dimension Style** dialog box. Choose the **New Layer** button in the **Layer Properties Manager** palette. In the **Lines and Arrows** tab, set the **Arrow size:** to **.125**.
AutoCAD prompts—Dynamic input prompts are set in a different font to distinguish them from the text. Command-line prompts are set to look like the text in the command line, including capitalization, brackets, and punctuation. Text following the prompt's colon specifies user input in bold.	*AutoCAD prompts you to specify first point:* `Specify center point for circle or [3P 2P Ttr (tan tan radius)]:` **3.5**
Keyboard Input—Bold with special keys in brackets.	Type **3.5 <Enter>**.

Download Instructor Resources from the Instructor Resource Center

Instructor materials are available from Pearson's Instructor Resource Center. Go to https://www.pearson.com/en-us/highered-educators.html to register or to sign in if you already have an account.

Contents

Appendix D and Glossary are available online at peachpit.com/Discoverautocad2024.

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11 chapter eleven

Isometric Drawing

CHAPTER OBJECTIVES

- Use isometric snap
- Switch isometric planes
- Use **COPY** and other edit commands
- Draw isometric circles with **ELLIPSE**
- Draw text aligned with isometric planes
- Draw ellipses in orthographic views
- Save and restore displays with **VIEW**

Introduction

Learning to create isometric drawings should be a pleasure at this point. There are very few new commands to learn, and anything you already know about manual isometric drawing makes it that much easier on the computer. Once you know how to get into the isometric mode in AutoCAD and change from plane to plane, you can rely on previously learned skills and techniques. Many of the commands from early chapters in this book will work readily, and you will find that using the isometric drawing planes is an excellent warm-up for 3D wireframe and solid modeling.

Using Isometric Snap

To begin drawing isometrically, you need to switch to the isometric snap style. You will find the grid and crosshairs behaving in ways that might seem odd at first, but you will quickly get used to them.

TIP

The following is a general procedure for entering isometric snap mode:

1. Click the **ISODRAFT** tool from the status bar.
2. Press **<F5>** to cycle through the three isoplanes.

✔ Begin a new drawing using decimal units and 18 × 12 limits. Use the *1B* template if you have it; otherwise, be sure to use the *acad.dwt* template.

✔ Check to see that the **Grid Mode** and **Snap Mode** tools are on.

✔ Click the **ISODRAFT** tool from the status bar, as shown in Figure 11-1.

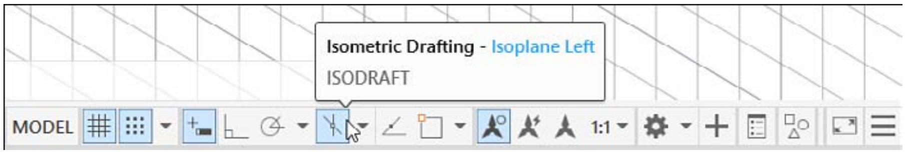


Figure 11-1
ISODRAFT tool

NOTE

Because isoplane settings are retained within the current session, it is possible that your crosshairs may be turned to the top or right isoplane instead of the left, as shown. It is not necessary to change the setting in this exercise.

At this point, your grid and crosshairs are reoriented, resembling Figure 11-2. *This is the 2D model space grid in isometric mode. Gridlines are drawn at 30°, 90°, and 150° angles from the horizontal, depending on which isoplane is being represented. The crosshairs are initially turned to define the left isometric plane, and gridlines are drawn to represent the left isoplane, with lines at 90° and 150°. The three* ***isoplanes*** *are discussed in the "Switching Isometric Planes" section.*

isoplane: One of three planes used for isometric drawing. In AutoCAD, these planes are named left, right, and top.

✔ To get a feeling for how this snap style works, start the **LINE** command and draw some boxes, as shown in Figure 11-3.

Make sure ***Ortho*** *is off and* ***Snap*** *is on, or you will be unable to draw the lines shown.*

Figure 11-2
Isometric grid, left isoplane

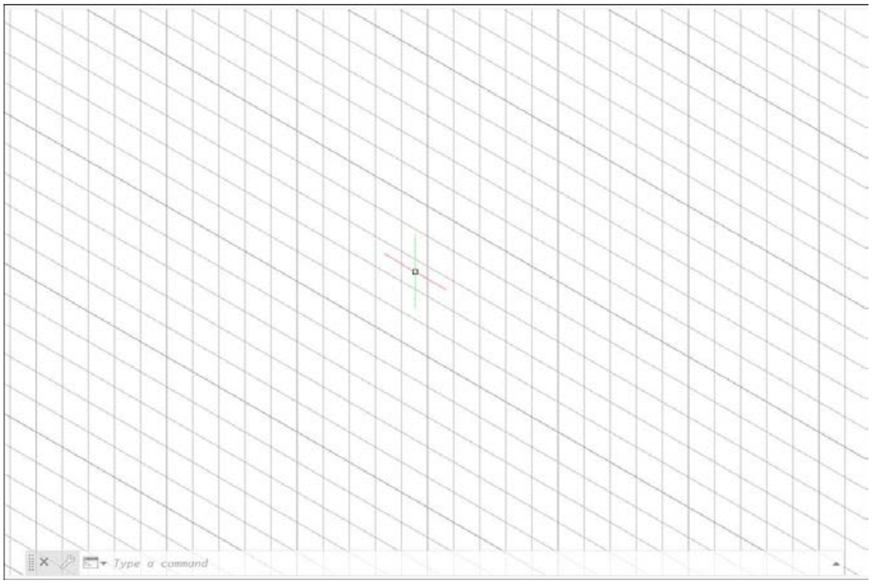
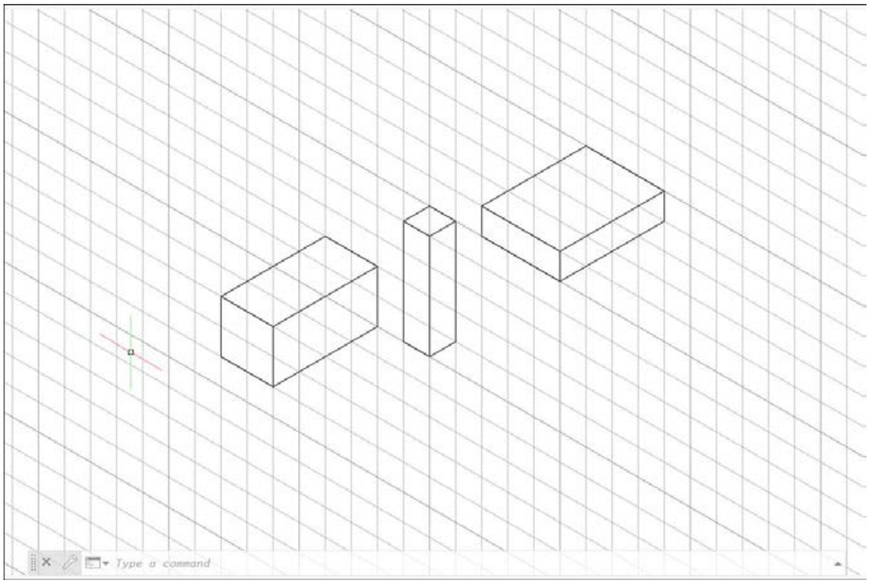


Figure 11-3
Isometric boxes



Switching Isometric Planes

If you tried to draw the boxes in the preceding section with **Ortho** on, you discovered that it is impossible. Without changing the orientation of the crosshairs, you can draw in only two of the three isometric planes. To utilize **Ortho** for accuracy and speed, you have to be able to switch planes. There are several ways to do this, but the simplest, quickest, and most convenient way is to use the **<F5>** key (or **<Ctrl>+E**).

Before beginning, take a look at Figure 11-4, which shows the three planes of a standard isometric drawing. These planes are often referred to as top, front, and right. However, AutoCAD's terminology is top, left, and right. While there are differences in the naming of the isometric planes, in this chapter, the AutoCAD terminology is used when referring to the isometric planes.

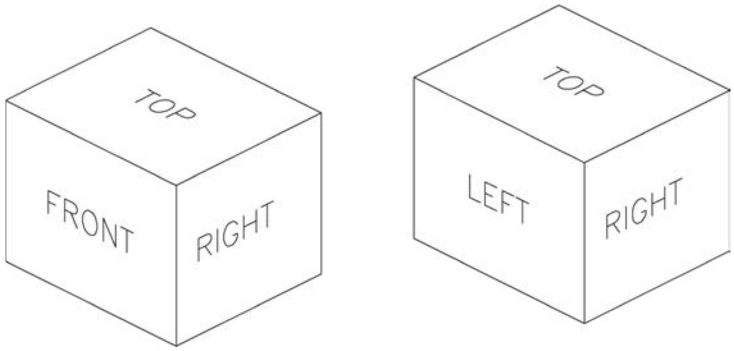


Figure 11-4
Isometric planes

Now look at Figure 11-5, you can see how the isometric crosshairs are oriented to draw in each of the planes. The gridlines change for each isoplane as well. They will be at 90° and 150° for the left isoplane, 30° and 150° for the top isoplane, and 30° and 90° for the right isoplane.

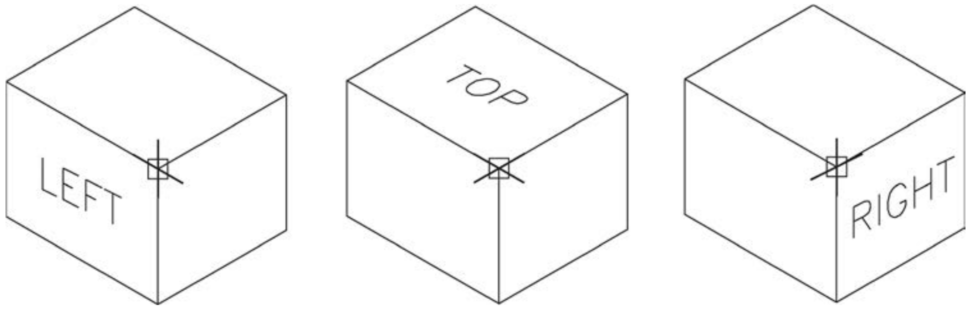


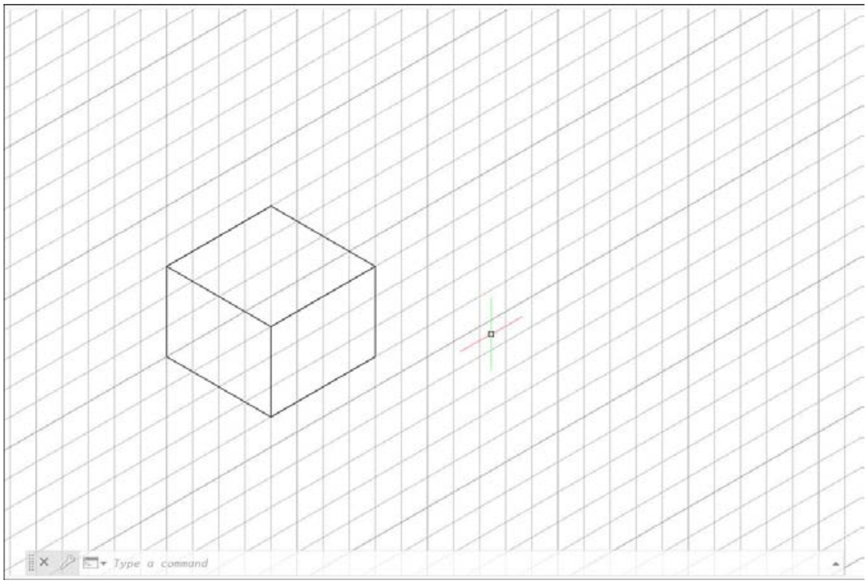
Figure 11-5
Isometric crosshairs

- ✔ Press **<F5>** (or **<Ctrl>+E**) to switch from left to top.

  *You can also open the Isoplane drop-down menu to the right of the **ISODRAFT** tool on the status bar to switch among the planes. The advantage of **<F5>** is that you can switch while drawing without moving your cursor away from the object you are drawing.*
- ✔ Press **<F5>** again to switch from top to right.
- ✔ Press **<F5>** once more to switch back to left.
- ✔ Now turn **Ortho** on and draw a box outline like the one in Figure 11-6.

  *You need to switch planes several times to accomplish this. Notice that you can switch planes using **<F5>** without interrupting the **LINE** command. If you find that you are in the wrong plane to construct a line, switch planes. Because every plane allows movement in two of the three directions, you can always move in the direction you want with one switch. However, you may not be able to hit the snap point you want. If you cannot, switch planes again.*

Figure 11-6
Isometric box outline



Using COPY and Other Edit Commands

Most commands work in the isometric planes just as they do in standard orthographic views. In this exercise, you construct an isometric view of a bracket using **LINE** and **COPY**. Then you draw angled corners using **CHAMFER**. In the next exercise, you will draw a hole in the bracket with **ELLIPSE**, **COPY**, and **TRIM**.

✔ Erase the boxes you previously drew and check to see that **Ortho** mode is on.

✔ Switch to the left isoplane.

✔ Click the **Line** tool on the ribbon.

✔ Draw the L-shaped object shown in Figure 11-7.
Notice that this is drawn in the left isoplane and that it is 3.00 units high by 4.00 units long by 1.00 unit wide.

✔ Next, you copy this object 4.00 units back to the right to create the back surface of the bracket.

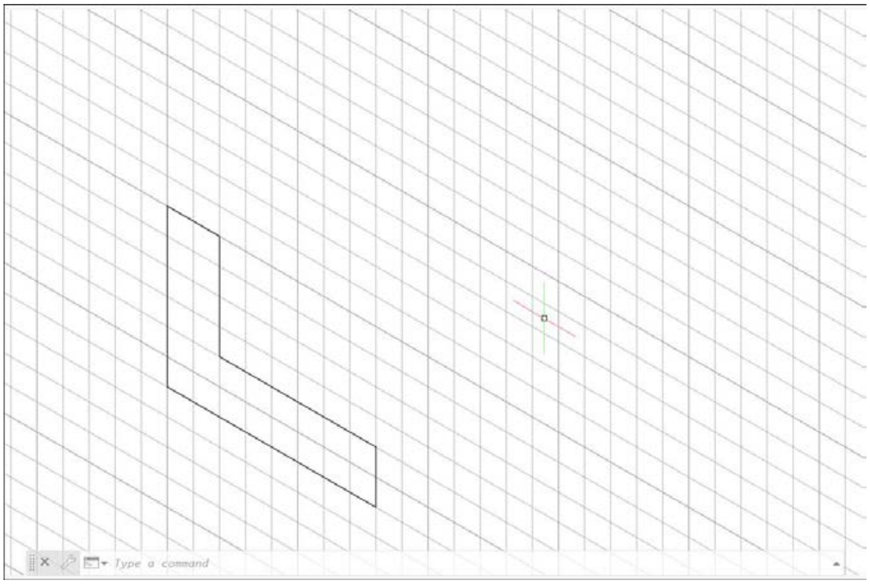
✔ Click the **Copy** tool on the ribbon.

✔ Select all the lines in the L.

✔ Right-click to end object selection.

✔ Specify a base point at the inside corner of the L.
*It is a good exercise to keep **Ortho** on, switch planes, and move the object around in each plane. You can move in two directions in each isoplane. To move the object back to the right, as shown in Figure 11-8, you must be in either the top or the right isoplane.*

Figure 11-7
Drawing an L shape



✔ Switch to the top or right isoplane and specify a second point of displacement 4.00 units back to the right, as shown in Figure 11-8.

✔ Press **<Enter>** to exit **COPY**.

✔ Enter the **LINE** command again and draw the connecting lines in the right plane, as shown in Figure 11-9.

Figure 11-8
Copying the L shape

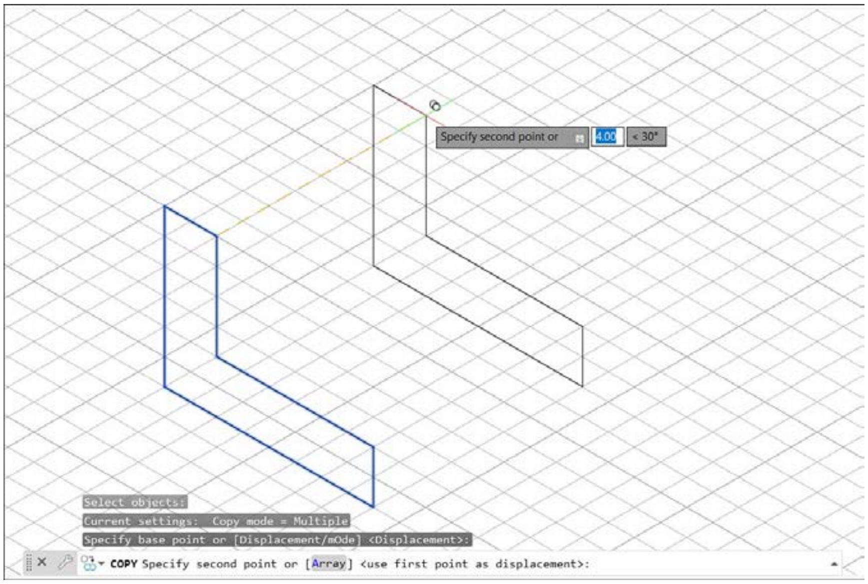
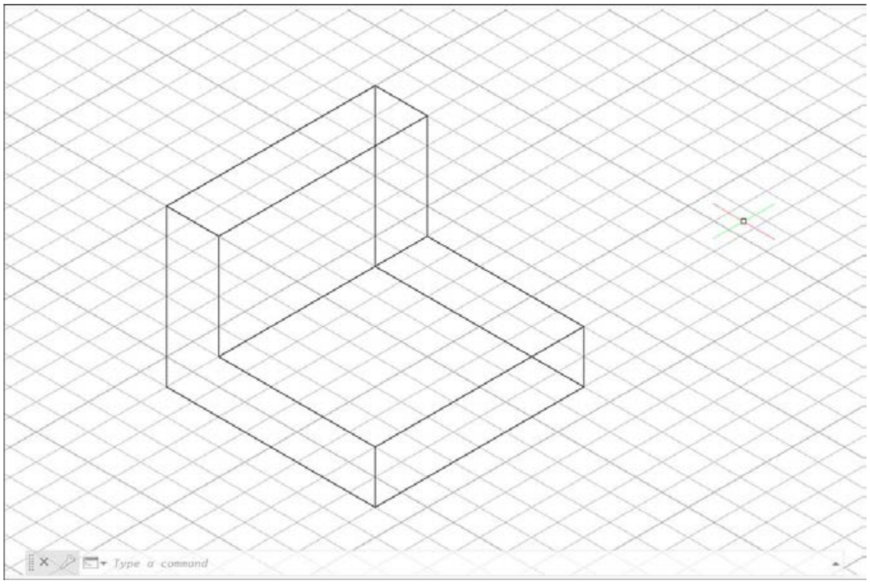


Figure 11-9
Drawing connecting lines



Creating Chamfers in an Isometric View

Keep in mind that inclined edges in an isometric view do not show true lengths. Edges of inclined planes must be drawn between endpoints located along vertical or horizontal paths in one of the three drawing planes. In this exercise, you create inclined edges by using the **CHAMFER** command to cut the corners of the bracket. This is no different from using **CHAMFER** in orthographic views.

✔ Click the **Chamfer** tool from the **Fillet/Chamfer** drop-down menu on the **Modify** panel on the ribbon's **Home** tab.

✔ Right-click and select **Distance** from the shortcut menu.
AutoCAD prompts for a first chamfer distance.

✔ Type **1 <Enter>**.

✔ Press **<Enter>** to accept 1.00 as the second chamfer distance.

✔ Select the top and left back edges of the bracket to create a chamfer, as shown in Figure 11-10.

✔ Repeat the **CHAMFER command**.

✔ Chamfer the other edges so that your drawing resembles Figure 11-10.

✔ To complete the bracket, start the **LINE** command and draw lines between the new chamfer edges.

✔ Finally, erase the two unseen lines on the back surface and the two corner lines left "in space" from the creation of the chamfers to produce what's shown in Figure 11-11.

Figure 11-10
Chamfering corners

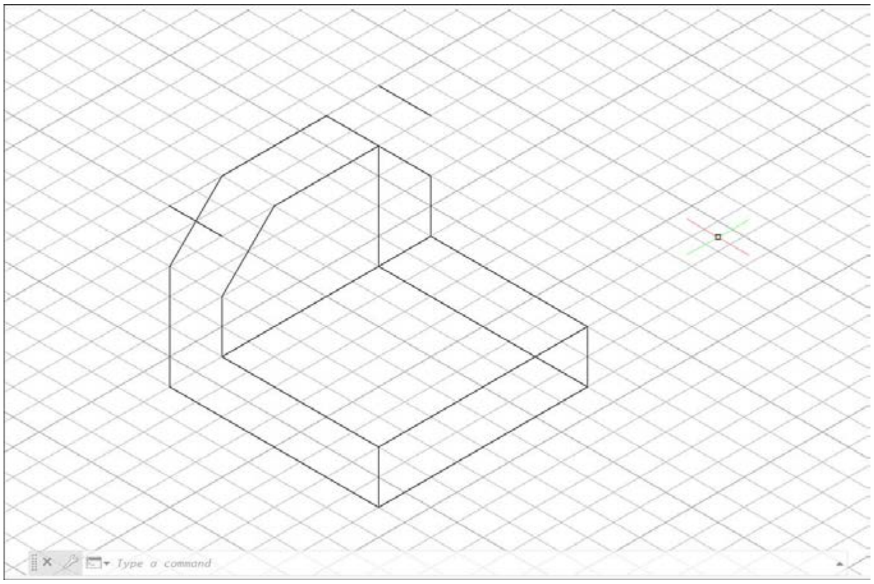
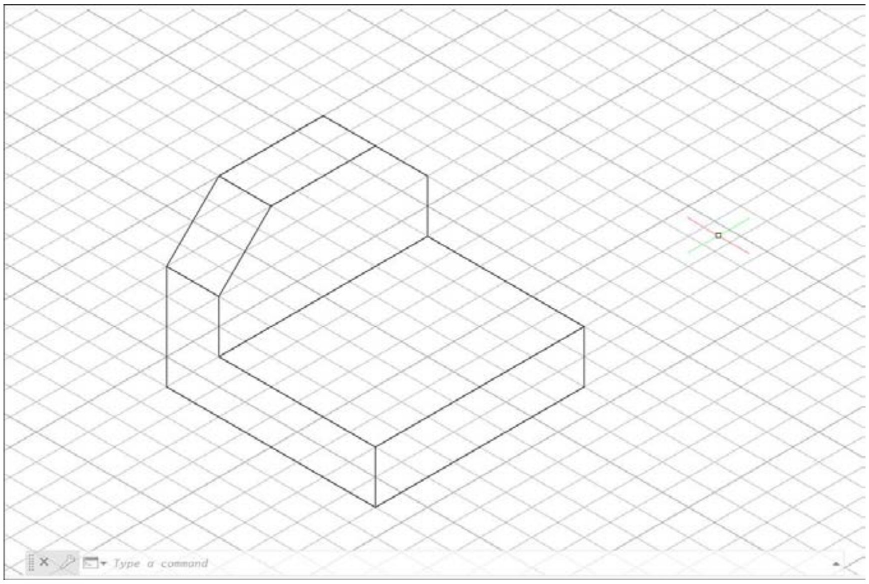


Figure 11-11
Finishing the chamfer and edges



ELLIPSE	
Command	ELLIPSE
Alias	El
Panel	Draw
Tool	

isocircle: The elliptical representation of a circle in an isometric drawing.

Drawing Isometric Circles with ELLIPSE

You can use the **ELLIPSE** command to draw true ellipses in orthographic views or ellipses that appear to be circles in isometric views (called ***isocircles*** in AutoCAD). In this exercise, you use the latter capability to construct a hole in the bracket.

✔ To begin, you should have the bracket shown in Figure 11-11 in your drawing.

*To draw an isocircle, you need a center point. Often, it is necessary to locate this point carefully using temporary lines, object snap tracking, or point filters. You must be sure that you can locate the center point before entering the **ELLIPSE** command.*

In this case, it is easy because the center point is on a snap point.

✔ Type **ellipse <Enter>** at the Command prompt.

*There is an **Ellipse** tool on the ribbon, but this automates an initial option and does not give you access to the **Isocircle** option.*

AutoCAD prompts:

```
Specify axis endpoint of ellipse or [Arc/Center/Isocircle]:
```

*The option you want is **Isocircle**. Ignore the others for the time being.*

✔ Select **Isocircle** from the command line, or right-click and select **Isocircle** from the shortcut menu.

AutoCAD prompts:

```
Specify center of isocircle:
```

If you could not locate the center point, you would have to exit the command now and start over.

✔ Use the **Snap Mode** and **Grid Mode** tools to specify the center of the ellipse, as shown in Figure 11-12. If you have drawn your object with the suggested dimensions, the center point will be over 2 units and back 2 units from the top-front corner of the bracket.

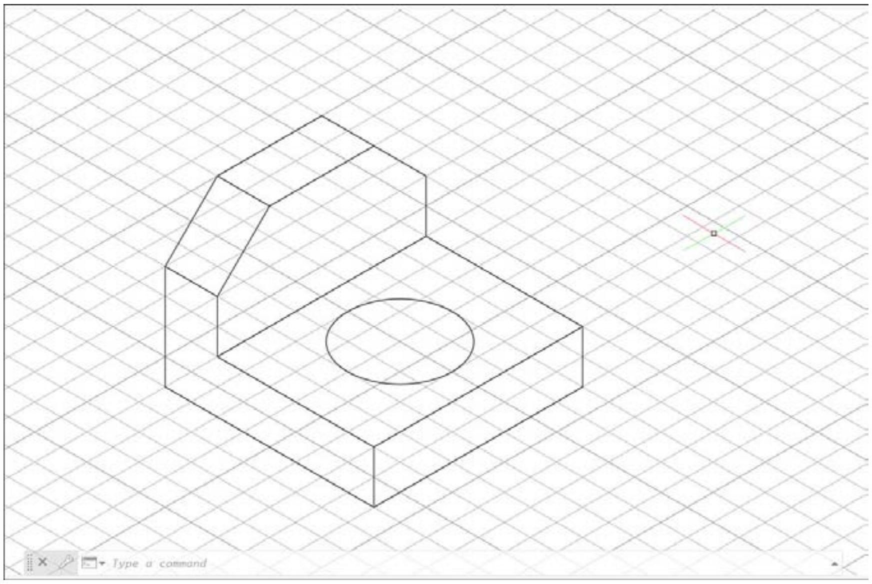


Figure 11-12
Ellipse isocircle

*AutoCAD presents you with an isocircle to drag, as in the **CIRCLE** command. The isocircle you see depends on the isoplane you are in. To understand this, try switching planes to see how the preview changes.*

✔ Drag the cursor to define the radius of the isocircle and then press **<F5>** to switch isoplanes. Observe the isocircle. Try this two or three times.

✔ Switch to the top isoplane before moving on.

AutoCAD prompts for a radius or diameter:

```
Specify radius of isocircle or [Diameter]:
```

*A radius specification is the default here, as it is in the **CIRCLE** command.*

✔ Specify a point so that your isocircle resembles the one in Figure 11-12.

*Next, you use the **COPY** and **TRIM** commands to create the bottom of the hole.*

✔ Start the **COPY** command.

✔ Select the isocircle.

✔ Right-click to end object selection.

✔ Specify the top-front corner of the bracket.

Any point can be used as the base point. By specifying the top-front corner, the bottom-front corner gives you the exact thickness of the bracket.

✔ Specify the bottom-front corner. Make sure you are in an isoplane that allows movement from top to bottom (the left or right isoplanes).

Your drawing should now resemble Figure 11-13. The last thing you must do is trim the hidden portion of the bottom of the hole.

✔ Press **<Enter>** to exit **COPY**.

✔ Start the **TRIM** command.

✔ Select **cuTting edges** from the command line, or right-click and select **cuTting edges** from the shortcut menu.

✔ Select the first isocircle as a cutting edge.

It may help to turn off snap to make these selections.

✔ Right-click to end cutting edge selection.

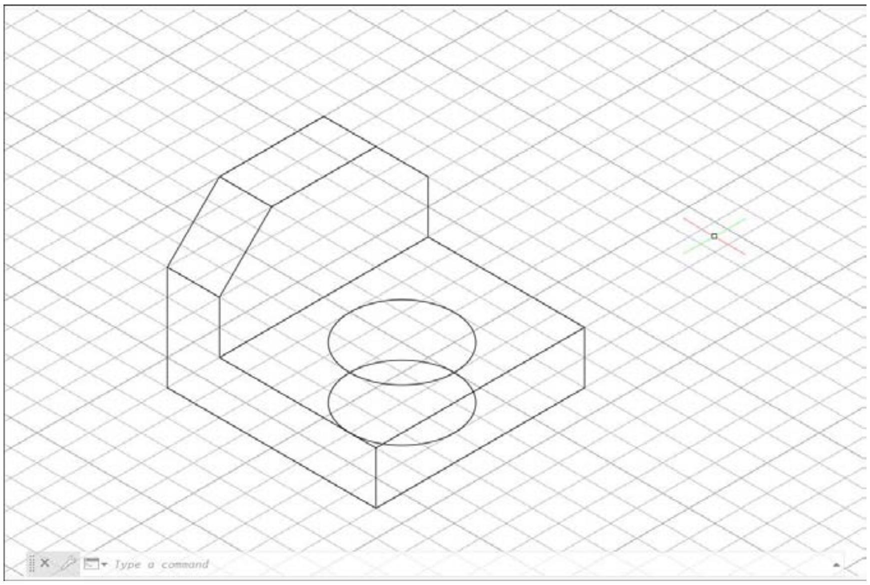


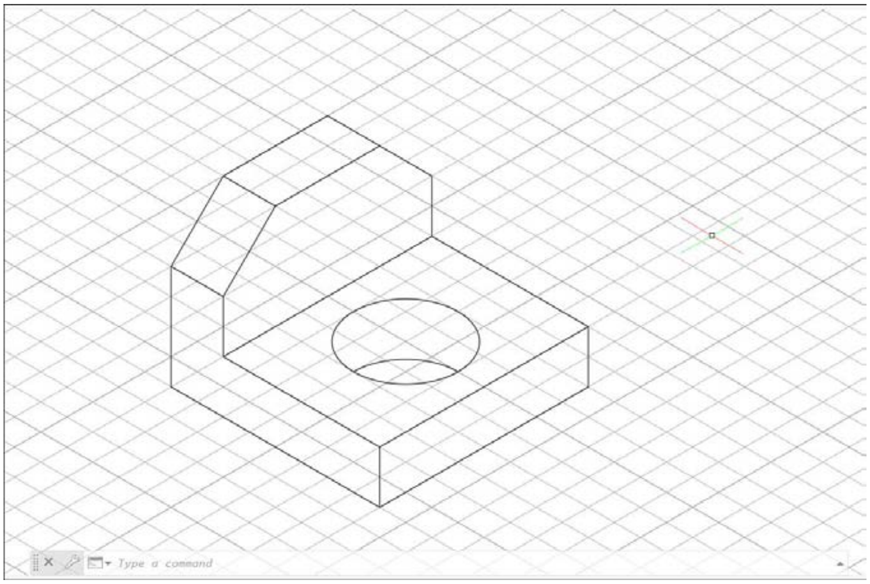
Figure 11-13
Copy isocircle

✔ Select the hidden section of the lower isocircle.

✔ Press **<Enter>** to exit **TRIM**.

The bracket is now complete, and your drawing should resemble Figure 11-14.

Figure 11-14
Trimming lower isocircle



Drawing Text Aligned with Isometric Planes

Adding text to isometric drawings has some challenges you may not have encountered previously. To create the appearance that text aligns with an isometric plane, it needs to be altered in two ways. First, the whole line of text needs to be rotated to align with one side of the plane. Second, the obliquing angle of individual characters needs to be adjusted to match the plane's tilt. Rotation angle, you recall, is handled through the command sequence of the **TEXT** command. Obliquing angle is set as a text style characteristic using the **STYLE** command.

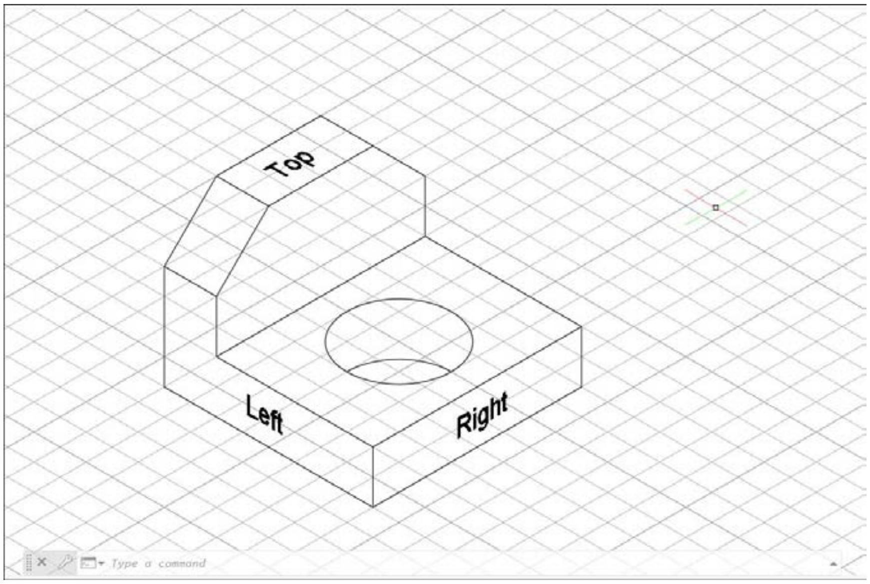
Typically, text in an isometric drawing aligns with one of the three isometric planes. In order to demonstrate how this works, you add a single-line text object to each of the planes of the bracket, as shown in Figure 11-15. Though you will be drawing on three planes, you can accomplish this with only two new text styles. These will be simple variations of the Standard text style, with the oblique angles needed for isometric alignment. The right isoplane will use a 30° oblique angle, while the top and left planes will use a –30° angle.

✔ To begin, you should be in the bracket drawing created in the previous sections. Isometric snap and grid modes should be enabled.

✔ Click the **Home** tab, expand the **Annotation** panel, and then click the **Text Style** tool in the top left, next to the name of the current text style (**Standard**).

This opens the ***Text Style*** *dialog box. The first new text style you create will be used for drawing text on the right isoplane. If you look at Figure 11-15, you can see that this text (the word* Right*) is rotated along the 30°* X*-axis of the isoplane. What may be less obvious is that the individual characters are also drawn at a 30° oblique angle. You enter the rotation angle when creating the text. Here, you set the oblique angle for this plane.*

Figure 11-15
Drawing text in isometric view



- ✔ In the **Text Style** dialog box, click the **New** button.
- ✔ In the **New Text Style** dialog box, type **isotext30**.
- ✔ Click **OK**.
- ✔ Change the **Oblique Angle** to **30**.
- ✔ Click **Apply**.

  You repeat these steps to create a style with –30° obliquing angle.
- ✔ Click the **New** button.
- ✔ In the **New Text Style** dialog box, type **isotext-30**.
- ✔ Click **OK**.
- ✔ Change the **Oblique Angle** to **–30**.
- ✔ Click **Apply**.
- ✔ Highlight **isotext30** in the **Styles** list.
- ✔ If you see a message saying the current text style has been modified, click **Yes**.

  *You should now be back in the drawing with **isotext30** as the current text style. You are now ready to add the single-line text.*
- ✔ Click the **Set Current** button.
- ✔ Click **Close**.
- ✔ Make sure to be on the right isoplane. Press **<F5>** to cycle through the isoplanes until you get to the right isoplane.
- ✔ On the **Annotation** panel, click the **Multiline Text/Single-line** text drop-down menu and choose the **Single Line** text tool.
- ✔ Use the **Justify** option and set the justification for the new text to **Middle Center (MC)**.

- ✔ Specify the middle point on the right front of the bracket, as shown by the placement of the word *Right* in Figure 11-15.
- ✔ Specify a text height of **.30**.
- ✔ Type **30 <Enter>** for the rotation angle.
- ✔ Type **Right <Enter>**.
- ✔ Press **<Enter>**.

    The word Right *should be drawn on the bracket, as shown in Figure 11-15. Now draw the word* Left *on the left isoplane, as shown. This will use the* ***isotext-30*** *style and a rotation angle of –30°.*

- ✔ Expand the **Annotation** panel and select **isotext-30** from the **Text Style** drop-down list.
- ✔ Press **<F5>** to set the left isoplane current.
- ✔ Click the **Single Line** text tool from the **Annotation** panel.
- ✔ Specify the middle point on the left side of the bracket, as shown by the placement of the word *Left* in Figure 11-15.
- ✔ Specify a text height of **.30**.
- ✔ Type **–30 <Enter>** for the rotation angle.
- ✔ Type **Left <Enter>**.
- ✔ Press **<Enter>**.

Finally, for text in the top isoplane, use **isotext-30** with a rotation angle of +30°.

- ✔ Press **<F5>** to set the top isoplane current.
- ✔ Click the **Single Line** text tool from the **Annotation** panel.
- ✔ Specify the middle point on the top of the bracket, as shown by the placement of the word *Top* in Figure 11-15.
- ✔ Press **<Enter>** for a text height of **.30**.
- ✔ Type **30 <Enter>** for the rotation angle.
- ✔ Type **Top <Enter>**.
- ✔ Press **<Enter>**.

    Your drawing should resemble Figure 11-15.

This completes the present discussion of isometric drawing. You can find more in the drawing suggestions at the end of this chapter.

Next, you go on to exploring the nonisometric use of the **ELLIPSE** command and saving named views with the **VIEW** command.

Drawing Ellipses in Orthographic Views

orthographic view: One of six standard views in which the observer's point of view is normal to the front, back, left, right, top, or bottom of the drawing plane.

The **ELLIPSE** command is important not only for drawing isocircles but also for drawing true ellipses in ***orthographic views***. There is also an option to create elliptical arcs.

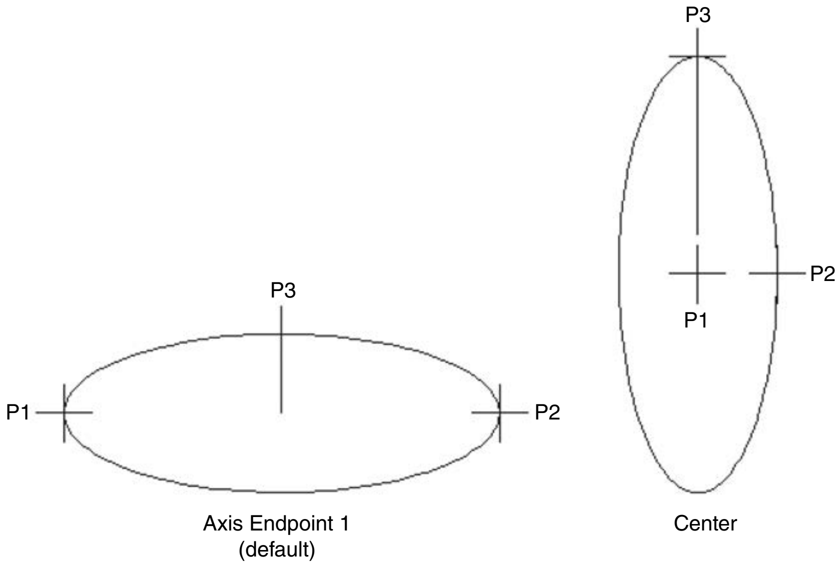
An ellipse is determined by a center point and two perpendicular axes of differing lengths. In AutoCAD, these specifications can be shown in two nearly identical ways, each requiring you to show three points (see Figure 11-16). In the default method, you show two endpoints of an axis and then show half the length of the other axis, from the midpoint of the first axis out. (The midpoint of an axis is also the center of the ellipse.) The other method allows you to specify the center point of the ellipse first, then the endpoint of one axis, followed by half the length of the other axis.

✔ In preparation for this exercise, return to the standard **Snap** mode and **Grid** mode by clicking the **ISODRAFT** tool on the status bar to turn off the **Isometric Snap** mode.

Your grid is returned to the standard pattern of lines, and the cross-hairs are horizontal and vertical again. Notice that this does not affect the isometric bracket you have just drawn.

*You briefly explore the **ELLIPSE** command and draw some standard ellipses.*

Figure 11-16
Ellipse axis and center



✔ **Ortho** should be off for this exercise.

✔ On the **Home** tab, click the **Ellipse** drop-down menu on the **Draw** panel and then select the **Axis**, **End** tool, as shown in Figure 11-17.

AutoCAD prompts:

```
Specify axis endpoint of ellipse or [Arc/Center]:
```

✔ Specify the axis endpoint, as shown by P1 on the ellipse at the lower left in Figure 11-18.

AutoCAD prompts for the other endpoint:

```
Specify other endpoint of axis:
```

✔ Specify the second endpoint, as shown by P2.

As you drag the rubber band, a preview of the ellipse is updated to show the length of the other axis. Only the length of the rubber band is significant; the angle is already determined to be perpendicular to the first axis. Because of this, the third point falls on the ellipse only if the rubber band happens to be exactly perpendicular to the first axis.

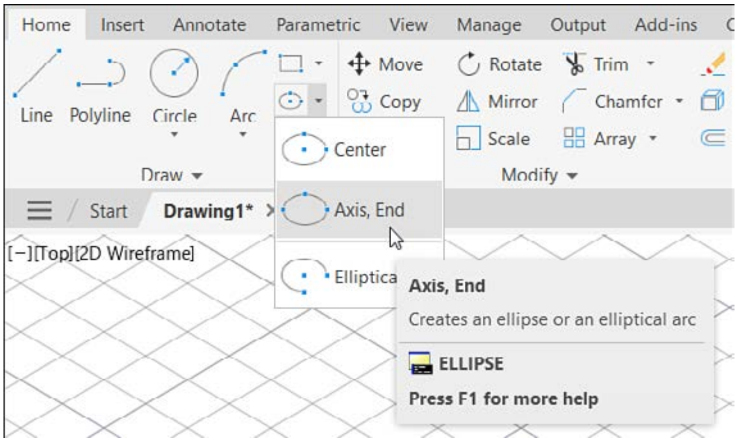


Figure 11-17
Ellipse Axis, End tool

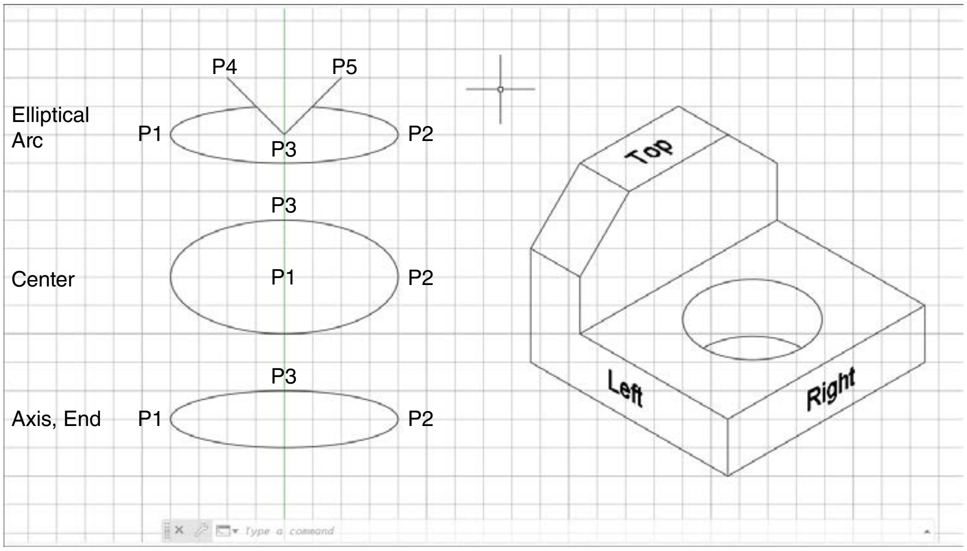


Figure 11-18
Drawing standard ellipses

As before, the following prompt allows you to specify the second axis distance or a rotation around the first axis:

```
Specify distance to other axis or [Rotation]:
```

*The **Rotation** option is awkward to use and is not explored here; see the AutoCAD Command Reference for more information.*

✔ Specify P3 as shown.

This point shows half the length of the other axis.

The first ellipse should now be complete. Next, you draw one by specifying the center point first, using the **Center** ellipse tool from the ribbon.

- ✔ Click the **Center** tool from the **Ellipse** drop-down menu on the ribbon's **Draw** panel.

  *This uses the **Center** option.*

  AutoCAD prompts you for a center point:

  ```
  Specify center of ellipse:
  ```

- ✔ Specify the center point, as shown by P1 at the middle left in Figure 11-18.

  Now, you have a rubber band stretching from the center to the end of an axis and the following prompt:

  ```
  Specify endpoint of axis:
  ```

- ✔ Specify an endpoint, as shown by P2 in Figure 11-18.

  The prompt that follows allows you to specify the second axis distance as before, or a rotation around the first axis:

  ```
  Specify distance to other axis or [Rotation]:
  ```

- ✔ Specify an axis distance, as shown by P3.

  Here again, the rubber band is significant for distance only. The point you specify falls on the ellipse only if the rubber band is stretched perpendicular to the first axis. Notice that it is not so in Figure 11-18.

Drawing Elliptical Arcs

Elliptical arcs can be drawn by trimming complete ellipses or using the **ELLIPSE** command's **Arc** option. Using the **Arc** option, you first construct an ellipse using one of the two methods shown previously and then define the arc of the ellipse that you want to keep.

- ✔ Click the **Elliptical Arc** tool from the **Ellipse** drop-down menu on the ribbon's **Draw** panel.
- ✔ Specify the first axis endpoint, as shown by P1 at the upper left in Figure 11-18.
- ✔ Specify the second endpoint, P2 in the figure.
- ✔ Specify P3 to indicate the second axis distance.

  *AutoCAD draws an ellipse as you have specified, but the object is only temporary. Now, you need to show the arc you want drawn. The two options are **Parameter** and **Included angle**. **Parameter** takes you into more options that allow you to specify your arc in different ways, similar to the options of the **ARC** command. Stick with the default option here.*

- ✔ Specify P4 to indicate the angle at which the elliptical arc begins.

  Move the cursor slowly along the ellipse's preview, and you can see all the arcs that are possible, starting from this angle.

- ✔ Specify P5 to indicate the end angle and complete the ellipse.

Saving and Restoring Displays with VIEW

VIEW	
Command	VIEW
Alias	V
Panel	Views
Tool	

The word *view* in connection with the **VIEW** command has a special significance in a drawing. It refers to any set of display boundaries that have been named and saved using the **VIEW** command. It also refers to a defined 3D viewpoint that has been saved with a name. Saved views can be restored by direct reference rather than by redefining the location, size, or viewpoint of the area to be displayed. **VIEW** can be useful in creating drawing layouts and when you know you will be returning frequently to a certain area of a large drawing. It saves you from having to zoom out to look at the complete drawing and then zoom in again on the area you want. It can also save time in creating a 3D viewpoint. In this chapter, you learn to use 2D views only.

Imagine that you have to complete some detail work on the area around the hole in the bracket and also on the top corner. You can define each of these as a view and jump back and forth at will.

✔ To begin this exercise, you should have the bracket in your drawing, as shown in Figure 11-19.

✔ Type **view <Enter>** or click **View** > **Named Views** > **View Manager** from the ribbon**,** as shown in Figure 11-20.

*This displays the **View Manager** dialog box shown in Figure 11-21. At the left is a list of views, including **Current**, **Model Views**, **Layout Views**, and **Preset Views**. In this chapter, you only work with the **Current** view, which you define and name.*

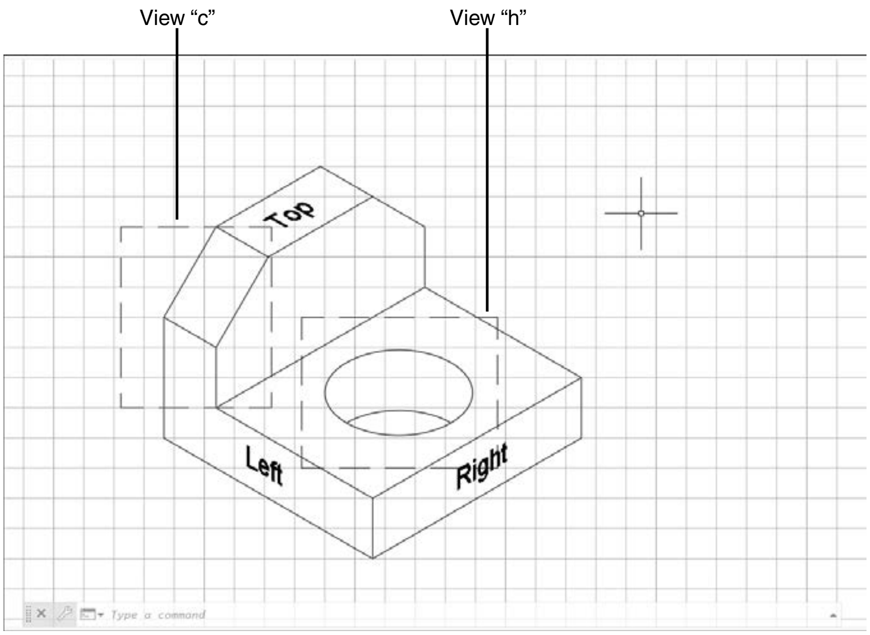


Figure 11-19
Defining views

Figure 11-20
View Manager tool

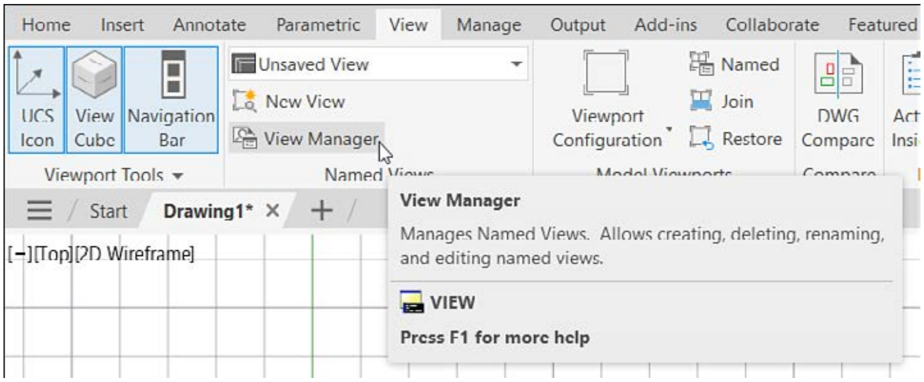
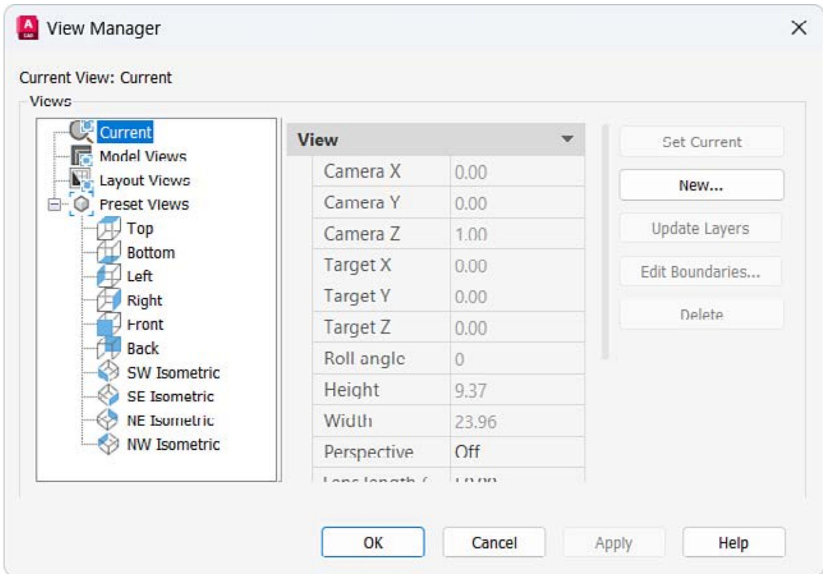


Figure 11-21
View Manager dialog box



✔ Click the **New** button.

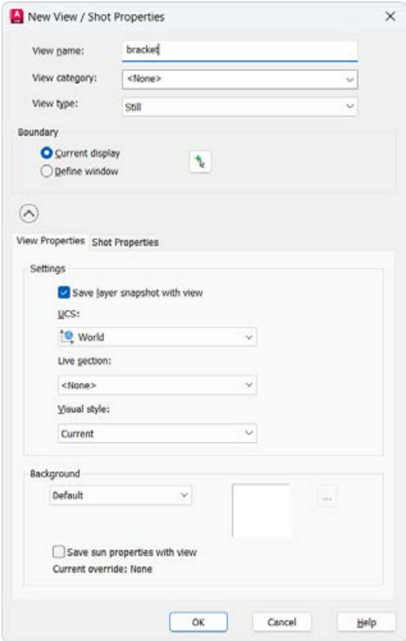
*This takes you to the **New View/Shot Properties** dialog box shown in Figure 11-22. Notice that the **Current Display** option is selected in the **Boundary** panel. All you have to do is give the current display a name to save it as a named view.*

✔ Type **bracket** in the **View Name** edit box.

✔ Click **OK**.

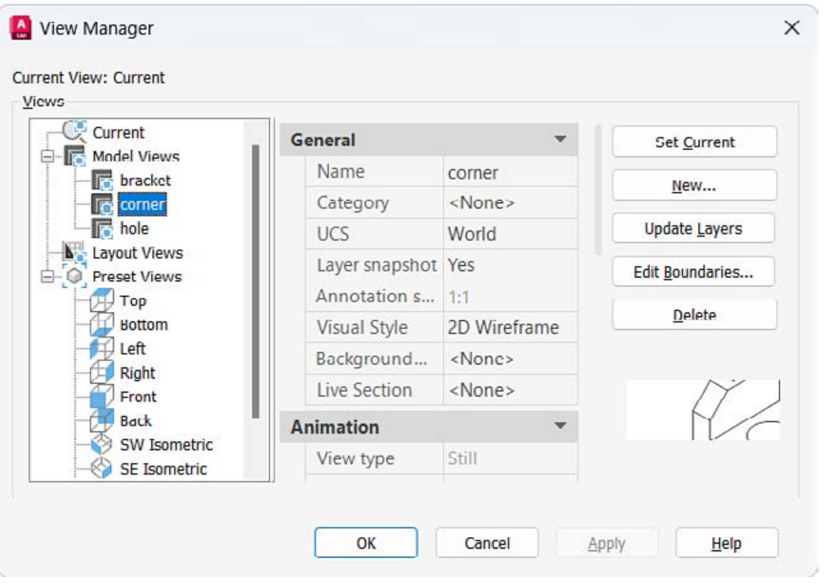
*The **View Manager** dialog box reappears, with **bracket** now showing under the **Model Views** heading. All views defined in model space will be listed as **Model Views**. Views defined in paper space will be listed as **Layout Views**. Next, you use a window to define a model space with a smaller area as a view.*

Figure 11-22
New View/Shot Properties
dialog box



- ✔ Click the **New** button to return to the **New View/Shot Properties** dialog box.
- ✔ Type **hole** in the **View Name** edit box.
  This view zooms in on the hole.
- ✔ Select the **Define Window** option in the **Boundary** panel.
  The dialog box closes, giving you access to the drawing area where the current view is outlined. The rest of the drawing is grayed out.
- ✔ Specify the first and second corners to define a window around the hole in the bracket, as shown previously in Figure 11-19.
  A window outline of the new view is shown, with the rest of the drawing grayed out.
- ✔ Press **<Enter>** to return to the **New View/Shot Properties** dialog box.
- ✔ Click **OK** to save the named view.
  *You are now back in the **View Manager** dialog box with **bracket** and **hole** in the list of **Model Views**. Define one more view to show the upper-left corner of the bracket, as shown in Figure 11-19.*
- ✔ Click the **New** button.
- ✔ Type **corner** for the view name.
- ✔ Select the **Define Window** option.
- ✔ Define a window, as shown in Figure 11-19.
- ✔ Press **<Enter>** to return to the dialog box.
- ✔ Click **OK** to close the **New View/Shot Properties** dialog box.
  *You have now defined three model views. To see the views in action, you must set them as current. Notice that the new view names are now displayed in a list on the **Views** panel, as shown in Figure 11-23.*

Figure 11-23
New view list on the **Views** panel



✔ Double-click **hole** in the **Views** list.

✔ *Click* **OK**.

Your drawing should resemble Figure 11-24.

Now, switch to the corner view.

✔ Instead of using the **View Manager** again to set a view current, click the **Views** drop-down list on the ribbon's **Named Views** panel.

✔ Select **corner** from the **Views** list.

Your drawing should resemble Figure 11-25.

Figure 11-24
Hole view

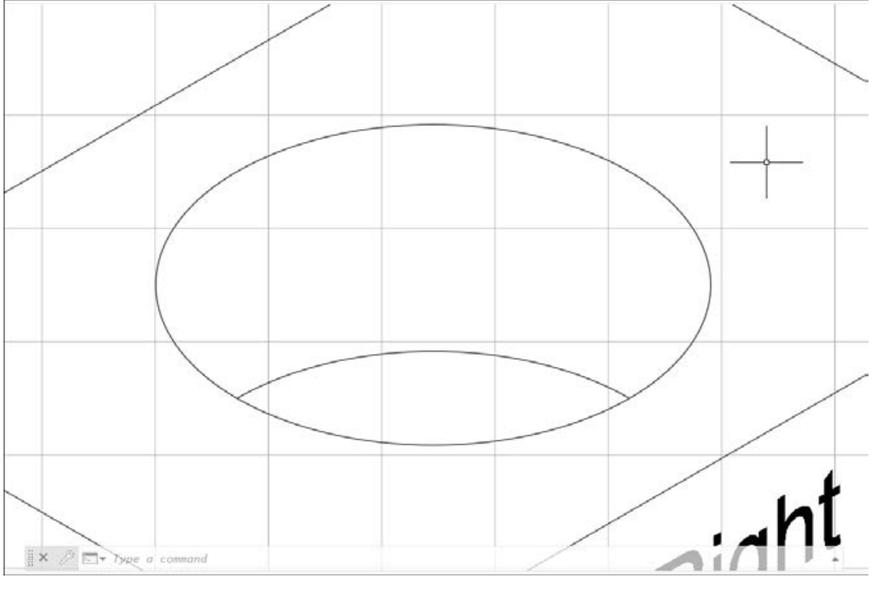
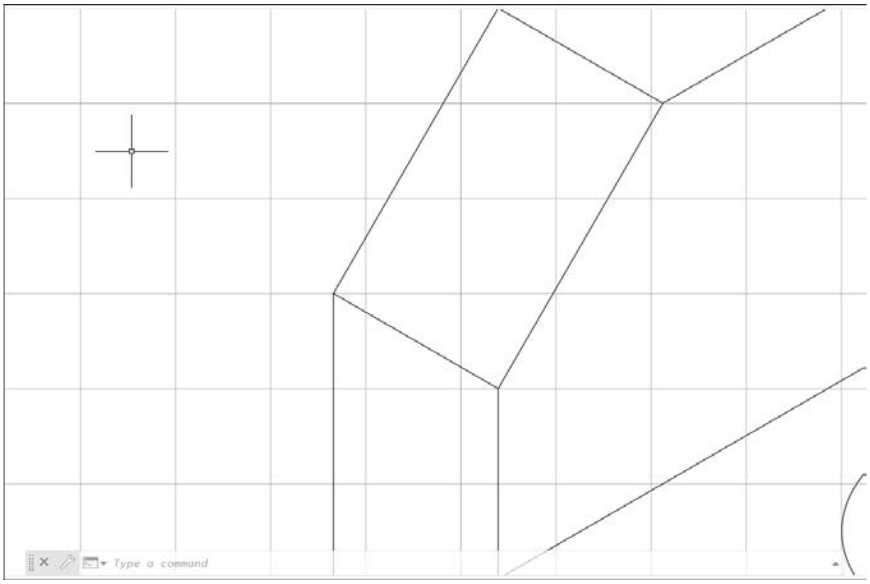


Figure 11-25
Corner view



chapter**eleven**

Chapter Summary

In this chapter, you were introduced to using the grid display and **Snap** mode to facilitate drawing geometry other than the standard orthographic 2D representation. You learned the use of the isometric grid to create an isometric drawing. In this mode, grid lines were shown at 30°, 90°, and 150° angles, and the crosshairs were oriented to the top, left, and right isoplanes. You used the **<F5>** key to switch among these planes, and the **LINE**, **COPY**, and **TRIM** commands to draw simple objects in isometric orientation. You also drew single-line text aligned with each of the three isoplanes. You also learned the use of the **ELLIPSE** command to create ellipses that appear as circles in isometric planes and also drew ellipses in standard orthographic views. Finally, you were introduced to the **VIEW** command, which allows you to define any display as a named view that can be restored at any time.

Chapter Test Questions

Multiple Choice

Circle the correct answer.

6. Change to this to align the grid with isometric planes:

a. Change grid to polar
b. Change grid to isometric
c. Change grid to 2D model space
d. Change snap to 2D model space

7. Which of these does **not** name an isoplane in AutoCAD?

a. Right
b. Left
c. Front
d. Top

8. The command used to draw circles in isometric views is

a. **ISOCIRCLE**
b. **ELLIPSE**
c. **ISOPLANE**
d. **CIRCLE**

9. To align text with isometric planes, make changes to

a. Rotation and oblique angles
b. Text style and alignment
c. Alignment style and rotation
d. Rotation angle and grid style

10. The minimum number of points required to define an elliptical arc are

a. 2
b. 3
c. 4
d. 5

Matching

Write the number of the correct answer on the line.

a. Isoplane switch ______
b. Isocircle ______
c. Right isoplane text ______
d. Top isoplane text ______
e. Default view ______

1. Ellipse
2. 30°, 30°
3. 30°, –30°
4. **<F5>**
5. Current display

True or False

Circle the correct answer.

1. **True or False:** Isometric drawings are two-dimensional.
2. **True or False:** Isometric drawings show no true distances.
3. **True or False:** To draw an isometric circle, it is necessary to specify a center point.
4. **True or False:** To switch isoplanes, you must open the **Drafting Settings** dialog box.
5. **True or False:** Drawing isocircles is the same as drawing ellipses in orthographic views.

Questions

1. What are the angles of the crosshairs and grid lines in an isometric grid?
2. What are the names for the isometric planes in AutoCAD?
3. What is an isocircle? Why are isocircles drawn with the **ELLIPSE** command?
4. How many different isocircles can you draw with the same radius and the same center point?
5. What rotation angle and oblique angle are used to align text with each of the three isoplanes?

Drawing Problems

1. Using the isometric grid, draw a 4 × 4 square in the right isoplane.
2. Copy the square back 4.00 units along the left isoplane.
3. Connect the corners of the two squares to form an isometric cube. Erase any lines that would be hidden in this object.
4. Use text rotation and obliquing to draw the word *Top* in the top plane of the cube so that the text is centered on the face and aligned with its edges. The text should be 0.5-unit high.
5. In a similar manner, draw the word *Left* at the center of the left side and the word *Right* at the center of the right side. All text should align with the face on which it is located.

11 chapter eleven

Chapter Drawing Projects

M

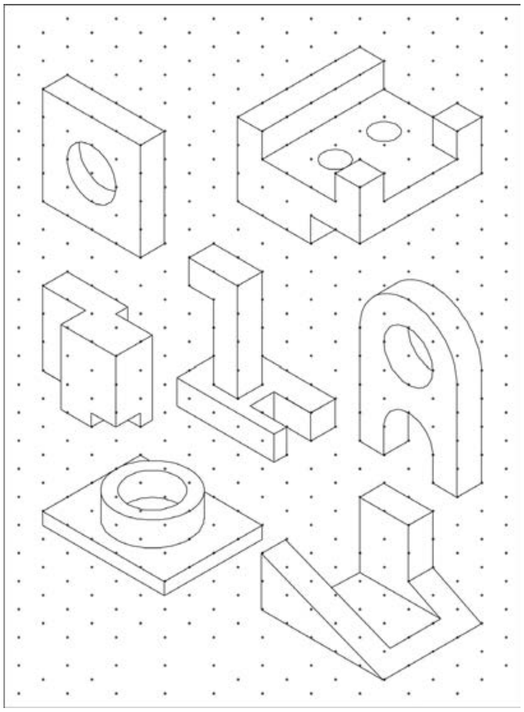
Drawing 11-1: *Isometric Projects* [INTERMEDIATE]

This drawing is a direct extension of the exercises in the chapter. It gives you practice in basic AutoCAD isometrics and in transferring dimensions from orthographic to isometric views.

Drawing Suggestions

- Set your grid to **.50** and your snap to **.25** to create all the objects in this project. Your grid should match the grid of this drawing. Notice that some lines do not fall on grid points, but halfway between.
- There is no **Arc** option when you use **ELLIPSE** to draw isocircles, so semicircles such as those at the back of the holes must be constructed by first drawing isocircles and then trimming or erasing unwanted portions.
- To draw the portion of the isocircle that shows the depth of a circle, copy the isocircle down or back, snapping from endpoint to endpoint of other lines in the view that show the depth.
- Often, when you try to select a group of objects to copy, there are many crossing lines that you do not want to include in the copy. This is an ideal time to use the **Remove** option in object selection. First, window the objects you want along with those nearby that are unavoidable, and then remove the unwanted objects one by one.
- Sometimes, you may get unexpected results when you try to trim an object in an isometric view. For example, AutoCAD divides an ellipse into a series of arcs and trims only a portion. If you do not get the results you want, use a **Nearest** object snap to control how the object is trimmed.

Drawing 11-1
Isometric projects



M

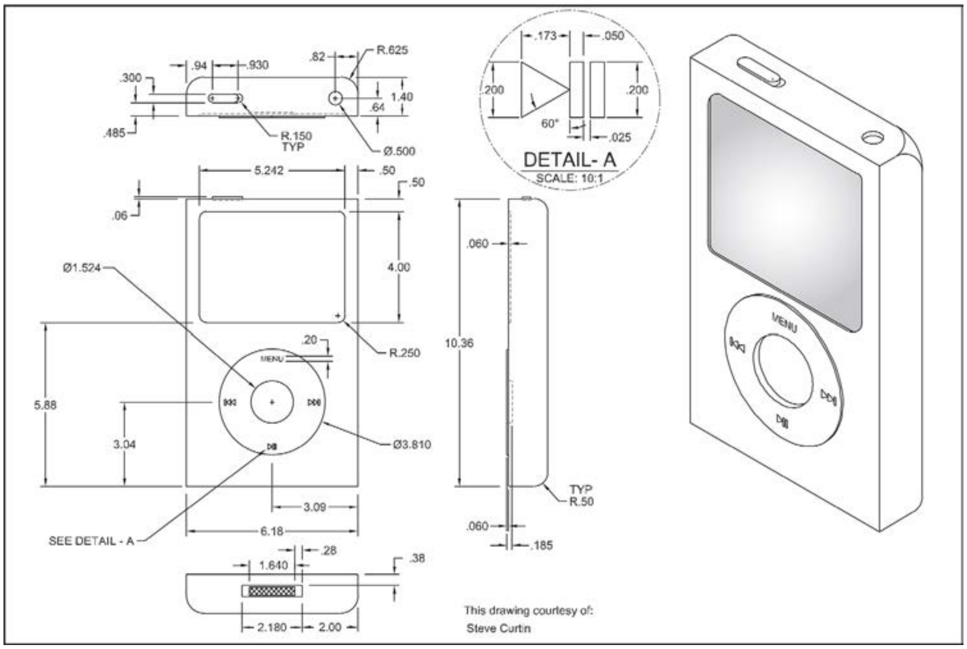
Drawing 11-2: *MP3 Player* [ADVANCED]

This drawing introduces text and combines a complete set of 2D views with an isometric representation of the object. Placing objects on different layers so they can be turned on and off during **TRIM** and **ERASE** procedures makes things considerably less messy.

Drawing Suggestions

- Use the box method to create the isometric view of this drawing. That is, begin with an isometric box according to the overall outside dimensions of the MP3 player. Then trim and add the details.
- The dial is made from isocircles with copies to show thickness. You can use the **Tangent** object snap to draw the front-to-back connecting lines.
- Use a gradient hatch for the video window area.
- Use the **isotext30** and **isotext-30** text styles created in this chapter for drawing the text on the left and right isoplanes, as shown.

Drawing 11-2
MP3 player



M

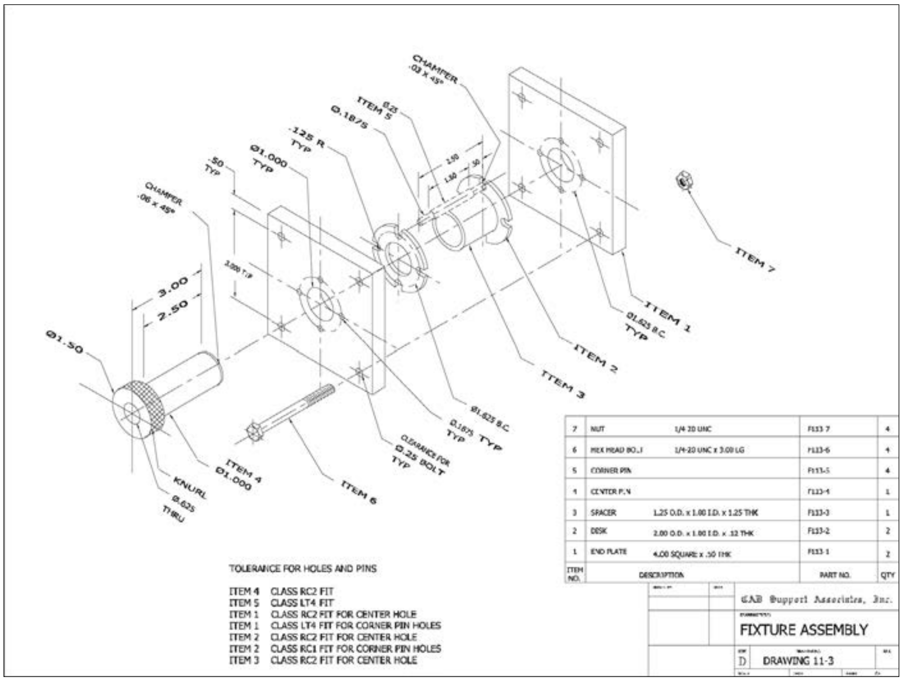
Drawing 11-3: *Fixture Assembly* [ADVANCED]

This is a difficult drawing. It takes time and patience but teaches you a great deal about isometric drawing in AutoCAD.

Drawing Suggestions

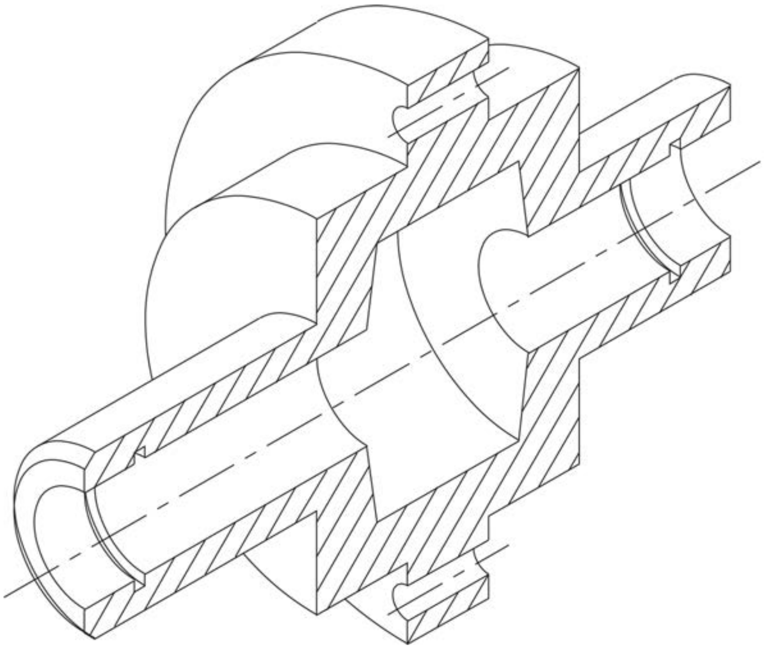
- This drawing can be completed either by drawing everything in place as you see it or by drawing the parts and moving them into place along the common centerline that runs through the middle of all the items. If you use the former method, draw the centerline first and use it to locate the center points of isocircles and as base points for other measures.
- As you go, look for pieces of objects that can be copied from other objects. Avoid duplicating efforts by editing before copying. In particular, when one object covers part of another, be sure to copy it before you trim or erase the covered sections.
- To create the chamfered end of Item 4, begin by drawing the 1.00-diameter cylinder 3.00 long with no chamfer. Then, copy the isocircle at the end forward 0.125. The smaller isocircle is 0.875 (7/8) because 0.0625 (1/16) is cut away from the 1.00 circle all around. Draw this smaller isocircle and trim away everything that is hidden. Then draw the slanted chamfer lines using **LINE**, not **CHAMFER**. Use the same method for Item 5.
- In both the screw and the nut, you need to create hexes around isocircles. Use the dimensions from a standard bolt chart.
- Use three-point arcs to approximate the curves on the screw bolt and the nut. Your goal is a representation that looks correct. It is impractical and unnecessary to achieve exact measures on these objects in the isometric view.

Drawing 11-3
Fixture assembly



M Drawing 11-4: *Flanged Coupling* [ADVANCED]

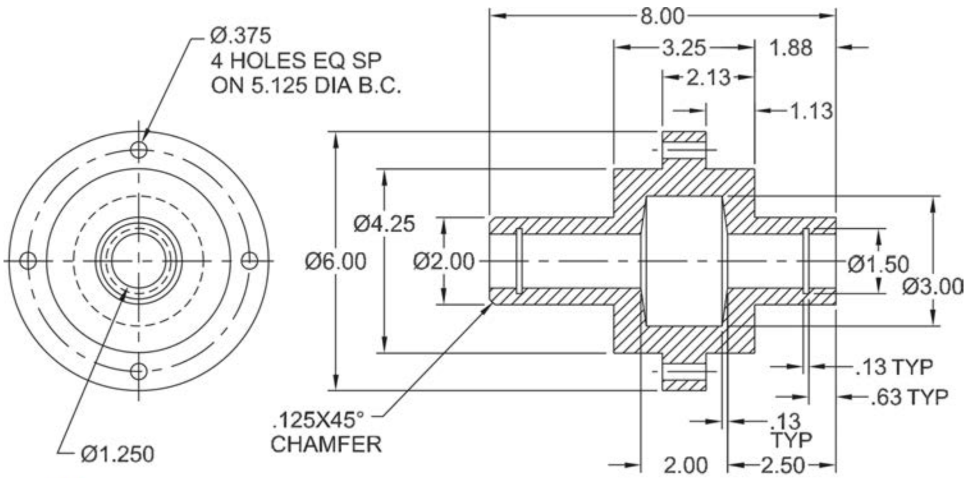
The isometric view in this three-view drawing must be completed working off the centerline.



Drawing Suggestions

- Draw the major centerline first. Then, draw vertical centerlines at every point where an isocircle is to be drawn. Make sure to draw these lines extra long so they can be used to trim the isocircles in half. By starting at the back of the object and working forward, you can take dimensions directly from the right-side view.
- Draw the isocircles at each centerline and then trim them to represent semicircles.
- Use the **Endpoint**, **Intersection**, and **Tangent** object snaps to draw horizontal lines.
- Trim away all obstructed lines and parts of isocircles.
- Draw the four slanted lines in the middle as vertical lines first. Then, with **Ortho** off, change their endpoints, moving them 0.125 closer.
- Remember, **MIRROR** does not work in the isometric view, although it can be used effectively in the right-side view.
- Use **HATCH** to create the crosshatching.
- If you have made a mistake in measuring along the major centerline, **STRETCH** can be used to correct it. Make sure **Ortho** is on and you are in an isoplane that lets you move the way you want.

Drawing 11-4
Flanged coupling



A

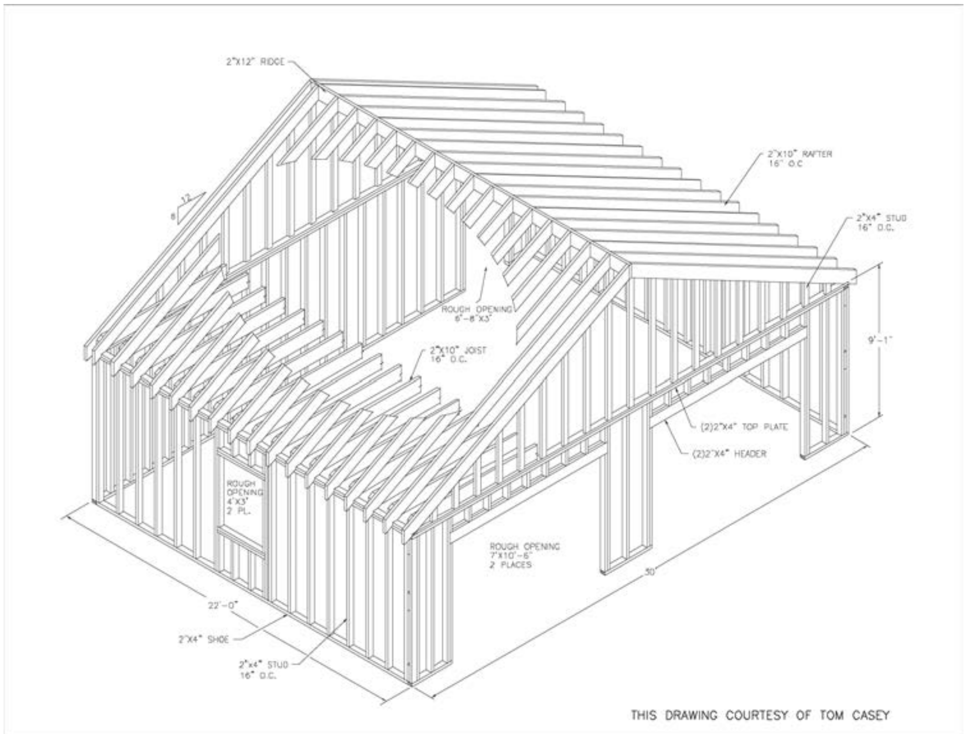
Drawing 11-5: *Garage Framing* [ADVANCED]

This is a fairly complex drawing that takes lots of trimming and careful work. Changing the **SNAPANG** (snap angle) variable so that you can draw slanted arrays is a method that can be used frequently in isometric drawing.

Drawing Suggestions

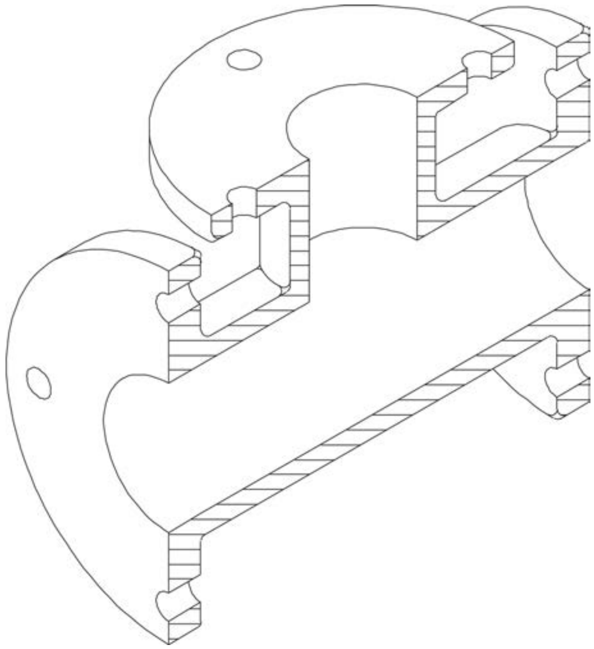
- You will find yourself using **COPY**, **ZOOM**, and **TRIM** a great deal. **OFFSET** also works well.
- You may want to create some new layers with different colors. Keeping different parts of the construction walls, rafters, and joists on different layers allows you to have more control over them and adds a lot of clarity to what you see on the screen. Turning layers on and off can considerably simplify trimming operations.
- You can cut down on repetition in this drawing by using arrays on various angles. For example, if the **SNAPANG** variable is set to 150°, the 229 wall in the left isoplane can be created as a rectangular array of studs with 1 row and 17 columns set 160 apart. To do so, follow this procedure:
  1. Type **snapang** and press **<Enter>**.
  2. Enter a new value so that rectangular arrays are created at isometric angles (30° or 150°).
  3. Enter the **ARRAY** command and create the array. Use negative values where necessary.
  4. Trim the opening for the window.
- One alternative to this array method is to set your snap to **16"** temporarily and use **COPY** to create the columns of studs, rafters, and joists. Another alternative is to use the grip edit offset snap method beginning with an offset snap of 16" (i.e., press **<Shift>** when you show the first copy displacement and continue to hold down **<Shift>** as you make other copies).
- The cutaway in the roof that shows the joists and the back door is drawn using the standard nonisometric **ELLIPSE** command. Then, the rafters are trimmed to the ellipse, and the ellipse is erased. Do this procedure before you draw the joists and the back wall. Otherwise, you will need to trim these as well.
- Use **CHAMFER** to create the chamfered corners on the joists.

Drawing 11-5
Garage framing



P Drawing 11-6: *Cast Iron Tee* [ADVANCED]

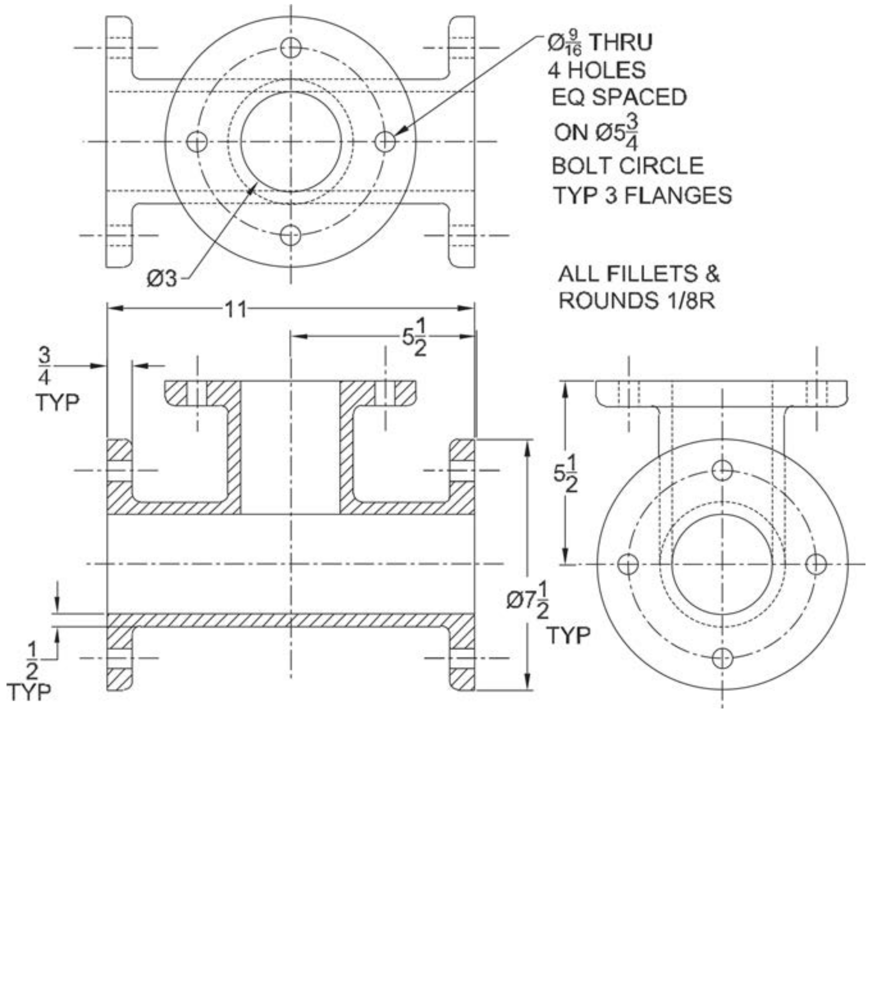
The objective of this exercise is to complete the isometric view of the tee using dimensions from the three-view drawing. Begin this isometric by working off the centerline.



Drawing Suggestions

- Be sure **Ortho** is on and you are in an isoplane that is correct for the lines you want to draw. Take full advantage of object snaps as you complete this drawing.
- First, draw the two major centerlines to exact length, as shown in the isometric view. Then, draw vertical centerlines at every point where an isocircle is to be drawn. These centerlines should be drawn longer so the isocircles can be trimmed more easily. Notice that **OFFSET** and **MIRROR** do not work very well in isometric drafting.
- After establishing the centerlines, draw the isocircles for the three flanges.
- When you have completed the flanges, draw the isocircles for the wall of the tee.
- Draw all horizontal and vertical lines and trim away all nonvisible lines and parts of the isocircles. Fillet the required intersections.
- After completing the outline of the tee, use **HATCH** to create the cross-hatching.

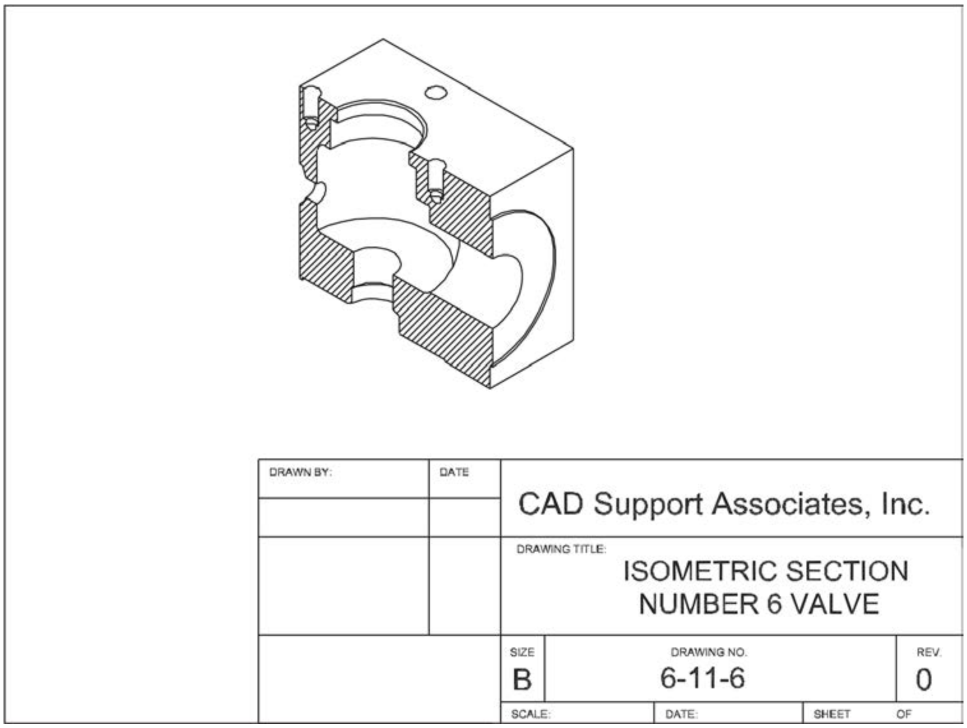
Drawing 11-6
Cast iron tee



P

Drawing 11-7: *Valve* [ADVANCED]

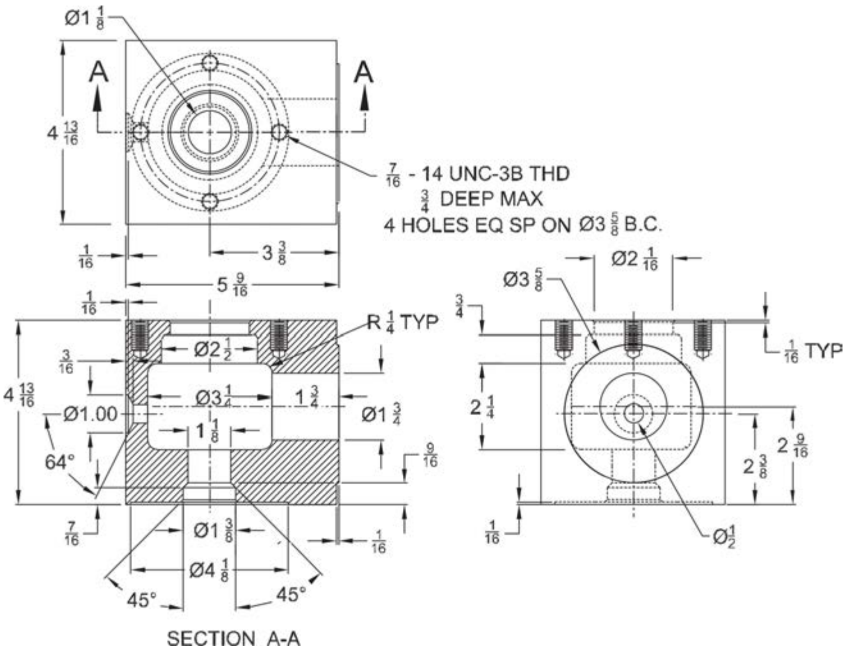
For the purposes of this chapter, the isometric view is most important. The three detail views, the title block, and the border can be included or not, as assigned.



Drawing Suggestions

- Use the box method to create the isometric view in this drawing. Begin with an isometric box according to the overall outside dimensions of the valve. Then go back and cut away the excess so the drawing becomes half the valve, exposing the interior details of the object.
- As in all section drawings, no hidden lines are shown.
- In addition to flat surfaces indicated by hatching, the interior is made up of isocircles of different sizes on different planes.
- Keep all construction lines and centerlines until the drawing is complete. (Draw them on a separate layer, and you can turn off that layer when you no longer need them.)
- The tapped holes are drawn with a series of isocircles that can be arrayed. This is only a representation of a screw thread, so it is not drawn to precise dimensions. Draw one thread, and copy it to the other side.

Drawing 11-7
Valve



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Index

Symbols

A

B

C

D

K

L

M

N

T

U

V

W

X–Y–Z