

Isometric Drawings

ISOMETRIC BASICS

Isometric drawings are a means of drawing an object in picture form for better clarifying the object's appearance. These types of drawings resemble a picture of an object that is drawn in two dimensions. As a result, existing AutoCAD commands such as `LINE` and `COPY` are used for producing isometric drawings. This chapter will explain isometric basics including how regular, angular, and circular objects are drawn in isometric. Numerous isometric aids such as snap and isometric axes will be explained to assist in the construction of isometric drawings.

Isometric drawings consist of two-dimensional drawings that are tilted at some angle to expose other views and give the viewer the illusion that what he or she is viewing is a three-dimensional drawing. The tilting occurs with two 30° angles that are struck from the intersection of a horizontal baseline and a vertical line (see Figure A-1). The directions formed by the 30° angles represent actual dimensions of the object; this may be either the width or depth. The vertical line in most cases represents the height dimension.

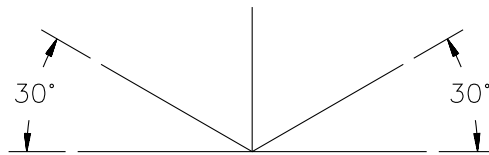


Figure A-1

Figure A-2 is a very simple example of how an object is aligned to the isometric axis. Once the horizontal baseline and vertical line are drawn, the 30° angles are projected from this common point, which becomes the reference point of the isometric view. In this example, once the 30° lines are drawn, the baseline is no longer needed and it is usually discarded through erasing. Depending on how the object is to be viewed, width and depth measurements are made along the 30° lines. Height is measured along the vertical line. Figure A-2 has the width dimension measured off to the left

30° line while the depth dimension measures to the right along the right 30° line. Once the object is blocked with overall width, depth, and height, details are added, and lines are erased and trimmed, leaving the finished object. Holes in isometric drawings no longer appear as full circles but rather as ellipses. Techniques of drawing circles in isometric will be discussed later in this chapter.

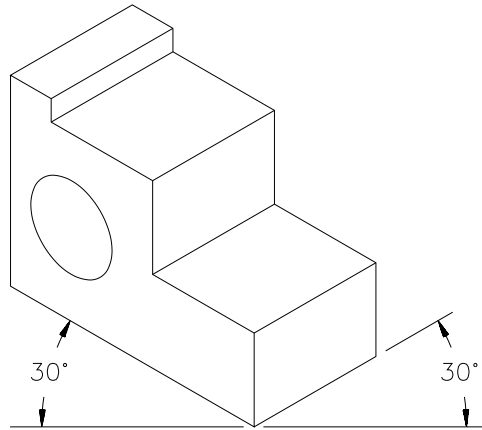


Figure A-2

Notice that the objects in Figure A-3 both resemble the object in Figure A-2 except they appear from a different vantage point. The problem with isometric drawings is that if an isometric of an object is drawn from one viewing point and you want an isometric from another viewing point, an entirely different isometric drawing must be generated from scratch. Complex isometric drawings from different views can be very tedious to draw. Another interesting observation concerning the objects in Figure A-3 is that one has hidden lines while the other does not. Usually only the visible surfaces of an object are drawn in isometric, with hidden lines left out. Although this is considered the preferred practice, there are always times when hidden lines are needed to clarify very complex isometric drawings.

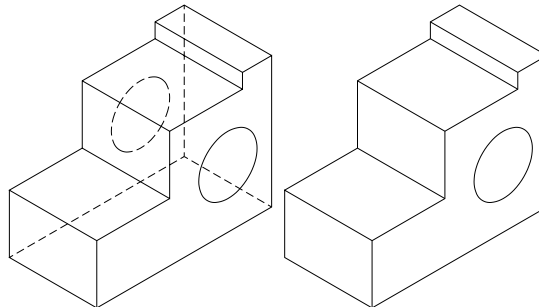


Figure A-3

CREATING AN ISOMETRIC GRID

Figure A-4 shows the current AutoCAD screen complete with cursor and grid on. In manual drawing and sketching days, an isometric grid was used to lay out all lines before the lines were transferred to paper or Mylar for pen and ink drawings. An isometric grid may be defined in an AutoCAD drawing through the **SNAP** command. This would be the same grid found on isometric grid paper.

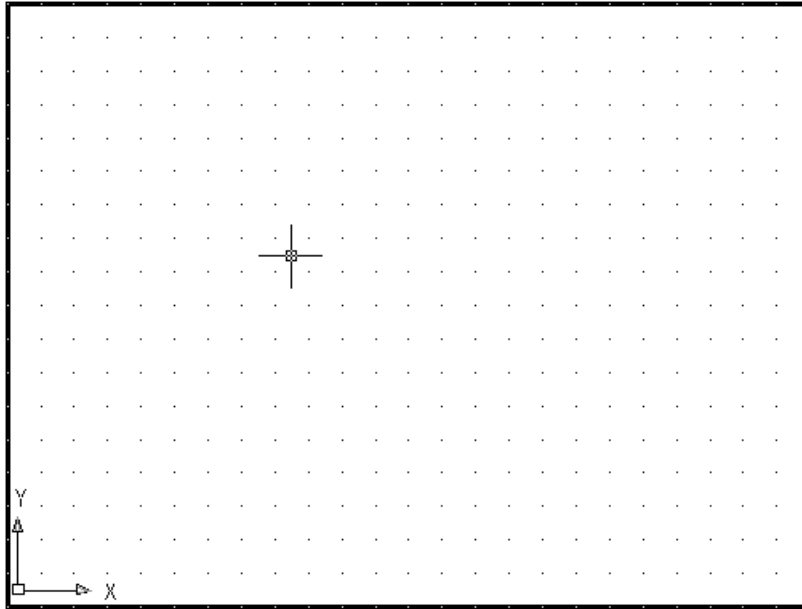


Figure A-4

The display screen in Figure A-5 reflects the use of the **SNAP** command and how this command affects the current grid display:

Command: **SN** (For **SNAP**)

Snap spacing or ON/OFF/Aspect/Rotate/Style <0.2500>: **S** (for **Style**)

Standard/Isometric <S>: **I** (for **Isometric**)

Vertical spacing <0.5000>: (Press **ENTER** to accept default value)

Choosing an isometric style of snap changes the grid display from orthographic to isometric, shown in Figure A-5. The grid distance conforms to a vertical spacing height that you specify. As the grid changes, notice the display of the typical AutoCAD cursor; it conforms to an isometric axis plane and is used as an aid in constructing isometric drawings.

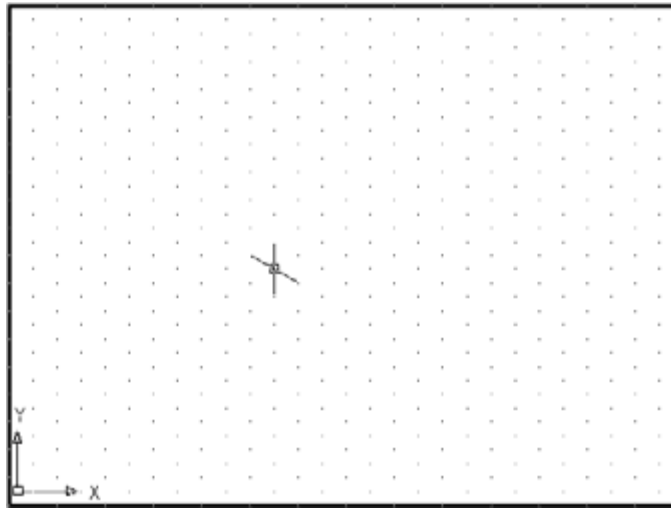


Figure A-5

To see how this vertical spacing distance affects the grid, changing it to isometric, see Figure A-6. The grid dot at “A” becomes the reference point where the horizontal baseline is placed followed by the vertical line represented by the dot at “B.” At dots “A” and “B,” 30° lines are drawn; points “C” and “D” are formed where they intersect. This is how an isometric screen display is formed.

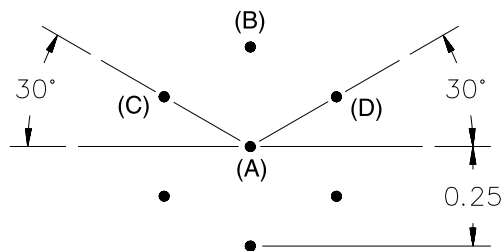


Figure A-6

ISOPLANE MODES

The AutoCAD cursor has always been the vehicle for drawing objects or constructing windows for object selection mode. Once in isometric snap mode, AutoCAD supports three axes to assist in the construction of isometric drawings. The first axis is the Left axis and may control that part of an object falling into the left projection plane. The Left axis cursor displays a vertical line intersected by a 30°-angle line, which is drawn to the left. This axis is displayed in the illustration in Figure A-7 and in the following drawing:

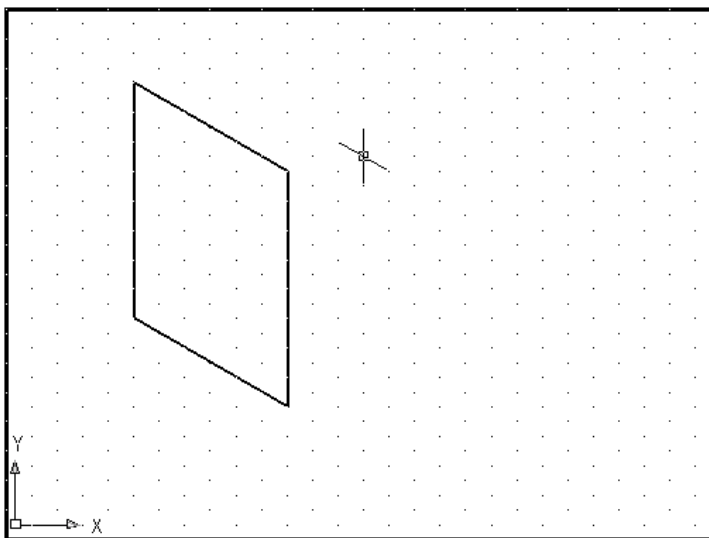


Figure A-7

The next isometric axis is the Top mode. Objects falling into the top projection plane may be drawn using this isometric axis. This cursor consists of two 30°-angle lines intersecting each other forming the center of the cursor. This mode is displayed in Figure A-8 and in the following drawing:

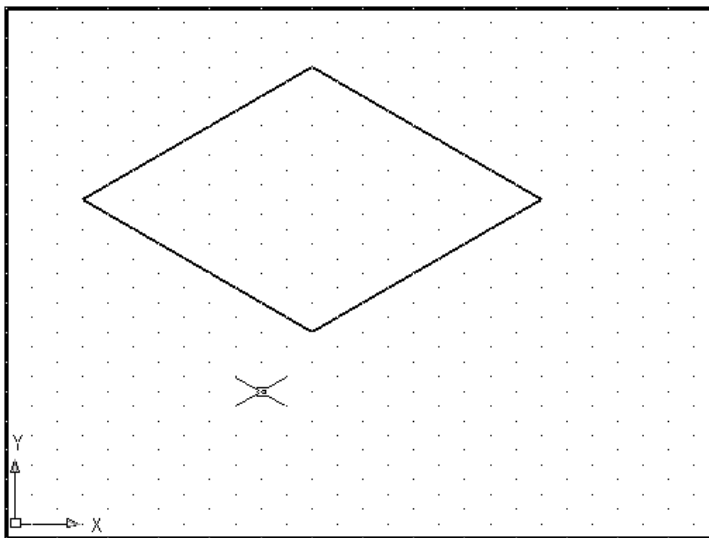


Figure A-8

The final isometric axis is called the Right mode and is formed by the intersection of a vertical line and a 30° angle drawn off to the right. As with the previous two modes, objects that fall along the right projection plane of an isometric drawing may be drawn with this cursor. It is displayed in Figure A-9 and in the following drawing. The current Ortho mode affects all three modes. If Ortho is on, and the current isometric axis is Right, lines and other operations requiring direction will be forced to be drawn vertical or at a 30° angle to the right, as shown in the following drawing:

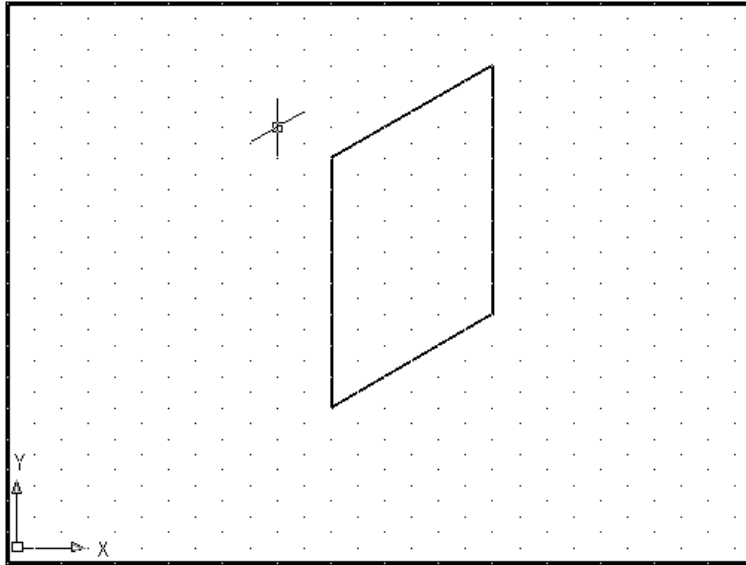


Figure A-9

RESETTING GRID AND SNAP TO THEIR DEFAULT VALUES

Once an isometric drawing is completed, it may be necessary to change the grid, snap, and cursor back to normal. This might result from the need to place text on the drawing, and the isometric axis now confuses instead of assists in the drawing process. Use the following prompts to reset the snap back to the Standard spacing. See Figure A-10.

Command: **SN** (For SNAP)

Snap spacing or ON/OFF/Rotate/Style <0.2500>: **S** (for Style)

Standard/Isometric <I>: **S** (for Standard)

Spacing/Aspect <0.5000>: (Press ENTER to accept the default value)

Notice that when you change the snap style to Standard, the AutoCAD cursor reverts to its original display.

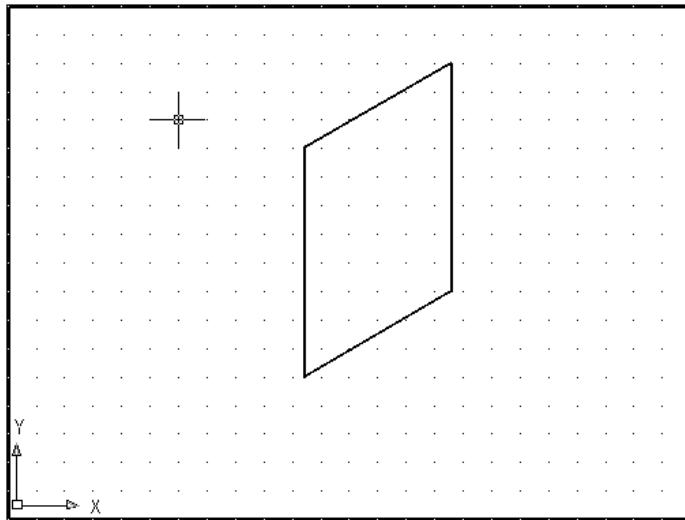


Figure A-10

ISOMETRIC AIDS IN THE DRAFTING SETTINGS DIALOG BOX
 Choosing Drafting Settings... from the Tools pull-down menu in Figure A-11A displays the Drafting Settings dialog box in Figure A-11B. Here, in addition to Snap and Grid, an Isometric area exists to automatically set up an isometric grid by clicking in the radio button next to Isometric Snap. This has the same effect as using the SNAP-Style-Isometric option.

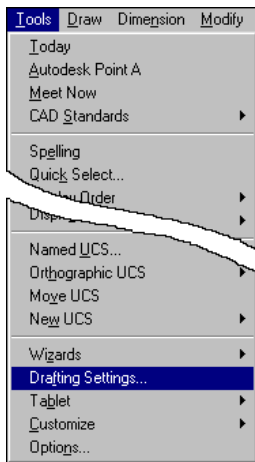


Figure A-11A

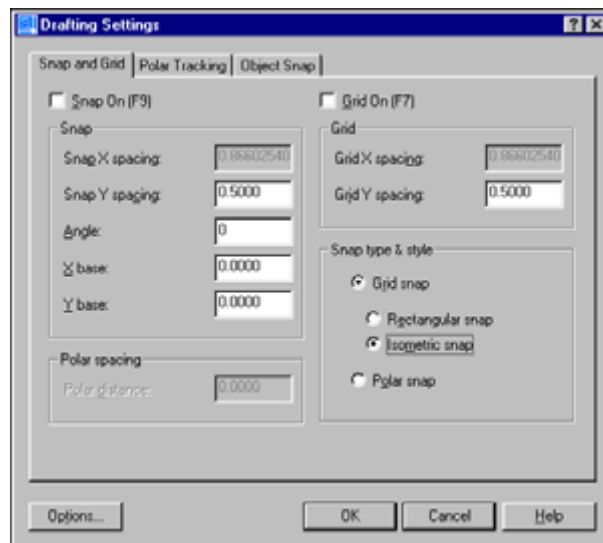


Figure A-11B

Various methods exist to switch from one isometric axis mode to another. By default, after you set up an isometric grid, the Left isometric axis mode is active (see Figure A-12). When you press **CTRL+E**, the Left axis mode changes to the Top axis mode. Pressing **CTRL+E** again changes from the Top axis mode to the Right axis mode. Pressing **CTRL+E** a third time changes from the Right axis mode back to the Left axis mode, and the pattern repeats from here. Using this keyboard entry, it is possible to switch or toggle from one mode to another.

The **F5** function key also allows you to scroll through the different Isoplane modes.

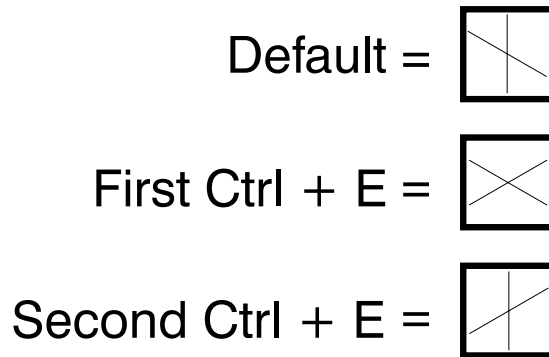


Figure A-12

CREATING ISOMETRIC CIRCLES

Circles appear as ellipses when drawn in any of the three isometric axes. The **ELLIPSE** command has a special **Isocircle** option to assist in drawing isometric circles; the **Isocircle** option will appear in the **ELLIPSE** command only if the current **SNAP-Style** is **Isometric** (see Figure A-13). The prompt sequence for this command is:

Command: **EL** (For **ELLIPSE**)
 Arc/Center/Isocircle/<Axis endpoint 1>: **I** (for **Isocircle**)
 Center of circle: (Select a center point)
 <Circle radius>/Diameter: (Enter a value for the radius or type "D" for diameter and enter a value)

When you draw isometric circles using the **ELLIPSE** command, it is important to match the isometric axis with the isometric plane the circle is to be drawn in. Figure A-14 shows a cube displaying all three axes.

Try It! - Open the drawing file **A_Iso Ellipse1**. This drawing is similar to Figure A-14. Using the three isometric planes, construct ellipses with a diameter of 2.00 units using the "X" points as centers.

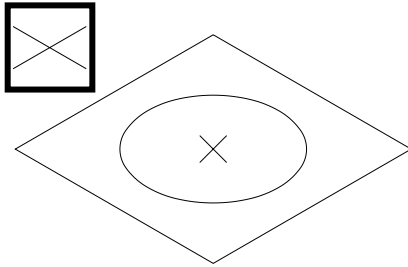


Figure A-13

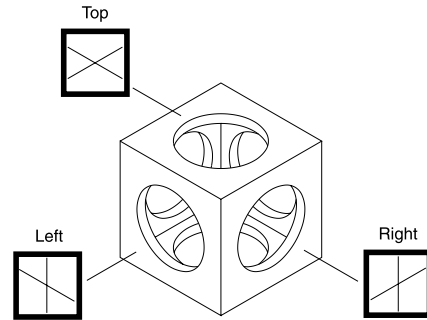


Figure A-14

Figure A-15 shows the result of drawing an isometric circle using the wrong isometric axis. The isometric box is drawn in the Top isometric plane while the current isometric axis mode is Left. An isometric circle can be drawn to the correct size, but notice that it does not match the box it was designed for. If you notice halfway through the ELLIPSE command that you are in the wrong isometric axis, press CTRL+E until the correct axis appears.

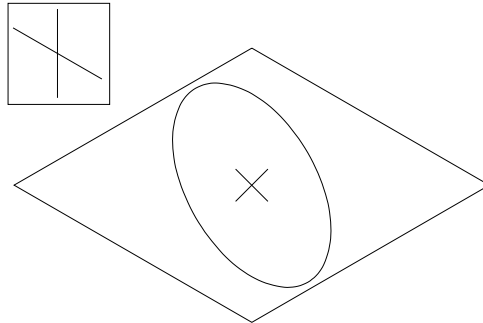


Figure A-15

BASIC ISOMETRIC CONSTRUCTION

Any isometric drawing, no matter how simple or complex, has an overall width, height, and depth dimension. Start laying out the drawing with these three dimensions to create an isometric box illustrated in the example in Figure A-16A. Some techniques rely on piecing the isometric drawing together by views; unfortunately, it is very easy to get lost in all of the lines using this method. Once a box is created from overall dimensions, somewhere inside the box is the object.

With the box as a guide, begin laying out all visible features in the primary planes. Use the Left, Top, or Right isometric axis modes to assist you in this construction process. See Figure A-16B.

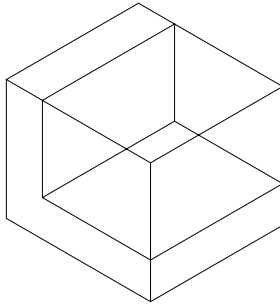


Figure A-16A

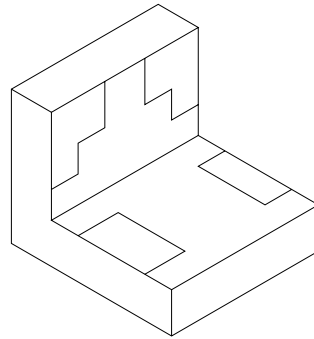


Figure A-16B

You may use existing AutoCAD editing commands, especially `COPY`, to duplicate geometry to show depth of features, as shown in Figure A-16C. The `OFFSET` command should not be attempted for performing isometric drawings. Next, use the `TRIM` command to partially delete geometry where objects are not visible. Remember, most isometric objects do not require hidden lines.

Use the `LINE` command to connect intersections of surface corners. The resulting isometric drawing is illustrated in Figure A-16D.

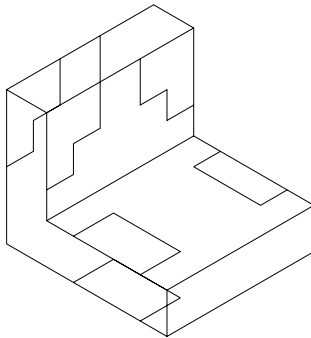


Figure A-16C

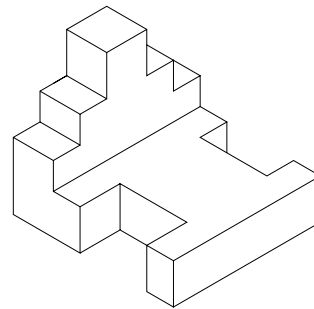


Figure A-16D

Try It! - Open the drawing file `A_Iso Basic` in Figure A-16B. Continue copying and trimming line segments until you achieve the desired results in Figure A-16D.

CREATING ANGLES IN ISOMETRIC—METHOD #1

Drawing angles in isometric is a little tricky but not impossible. The two-view drawing in Figure A-17 has an angle of unknown size; however, one endpoint of the angle measures 2.06 units from the top horizontal line of the Front view at “B,” and the

other endpoint measures 1.00 unit from the vertical line of the Front view at “A.” This is more than enough information to lay out the endpoints of the angle in isometric. You can use the MEASURE command to lay out these distances easily. The LINE command is then used to connect the points to form the angle.

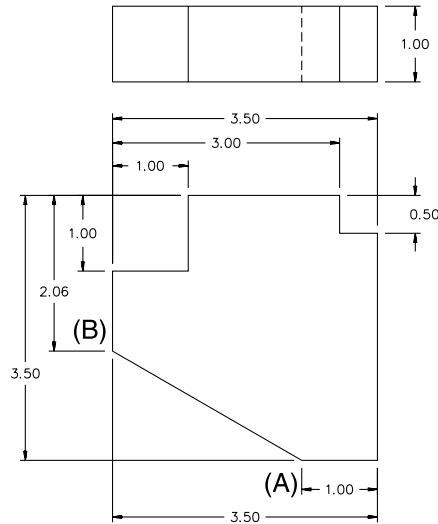


Figure A-17

Before using the MEASURE command, set the PDMODE variable to a new value of 3. Points will appear as an X instead of a dot, as shown in Figure A-18A. Points may also be set through the Point Style dialog box found in the Format menu. Now use the MEASURE command to set off the two distances.

Command: **PDMODE**
New value for PDMODE <0>: **3**

Command: **ME** (For MEASURE)
Select object to measure: (Select the inclined line at “A”)
<Segment length>/Block: **1.00**

Command: **ME** (For MEASURE)
Select object to measure: (Select the vertical line at “B”)
<Segment length>/Block: **2.06**

When using the MEASURE command, the measuring will occur at the nearest endpoint of the line where the line was selected. Once the points have been placed, the LINE command is used to draw a line from one point to the other using the OSNAP-Node option (see Figure A-18B).

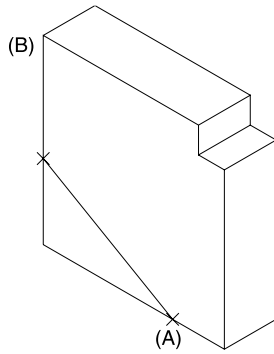


Figure A-18A

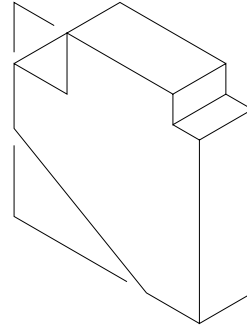


Figure A-18B

Try It! - Open the drawing file A_Iso Angle1. Use the previous figures as guides in constructing the angle in isometric mode in Figure A-18B. Notice that you may get more points than needed. Draw the line from the nodes nearest the end of the line you selected with the MEASURE command.

CREATING ANGLES IN ISOMETRIC—METHOD #2

The same two-view drawing is illustrated in Figure A-19. This time, one distance is specified along with an angle of 30° . Even with the angle given, the position of the isometric axes makes any angle construction by degrees inaccurate. The distance XY is still needed for constructing the angle in isometric. Use the MEASURE command to find distance XY, place a point, and connect the first distance with the second to form the 30° angle in isometric. It is always best to set the PDMODE system variable to a new value in order to visibly view the point. Points may also be set through the Point Style dialog box found in the Format menu. A new value of 3 will assign the point as an X.

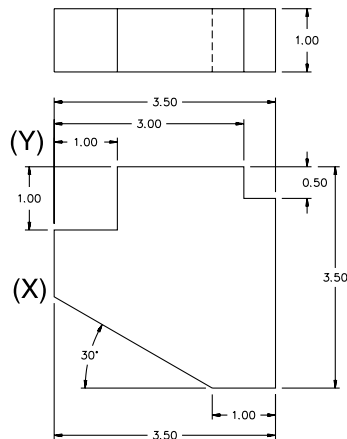


Figure A-19

See Figures 16–20A, 16–20B, and the following prompt sequences to change the point mode and measure the appropriate distances in the previous step.

Command: **PDMODE**

New value for PDMODE <0>: **3**

Command: **ME** (For MEASURE)

Select object to measure: (Select the inclined line at “A” in Figure A–20A)

<Segment length>/Block: **1.00**

Command: **ME** (For MEASURE)

Select object to measure: (Select the vertical line at “B” in Figure A–20A)

<Segment length>/Block: **End**

of (Select the endpoint of the line at X in the 2-view drawing in Figure A–19)

Second point: **Int**

of (Select the intersection at Y in the 2-view drawing in Figure A–19)

Line “B” is selected as the object to measure (see Figure A–20A). Since this distance is unknown, the MEASURE command may be used to set off the distance XY by identifying an endpoint and intersection from the Front view in Figure A–19. This means the view must be constructed only enough to lay out the angle and project the results to the isometric using the MEASURE command and the preceding prompts. The result is illustrated in Figure A–20B.

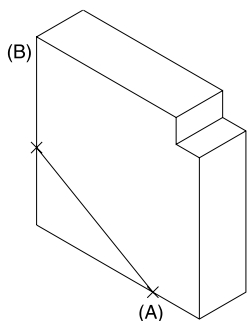


Figure A–20A

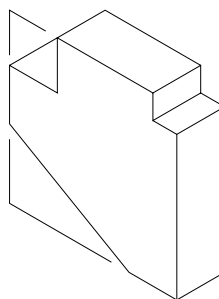


Figure A–20B

Try It! - Open the drawing file A_Iso Angle2. Use the previous figures as guides in constructing the angle in isometric mode in Figure A–20B. You can check the accuracy of your construction by comparing distances using the DISTANCE command on the Front view of the two-view drawing and on the same line in the Isometric. The values should be the same.

ISOMETRIC CONSTRUCTION USING ELLIPSES

Constructing circles as part of isometric drawing is possible using one of the three isometric axis positions. It is up to you to decide which axis to use. Before this, how-

ever, an isometric box consisting of overall distances is first constructed, as shown in Figure A-21A. Use the `ELLIPSE` command to place the isometric circle at the base. To select the correct axis, press `CTRL+E` or `F5` until the proper axis appears in the form of the cursor. Place the ellipses. Lay out any other distances.

For ellipses at different positions, press `CTRL+E` or `F5` to select another isometric axis (see Figure A-21B).

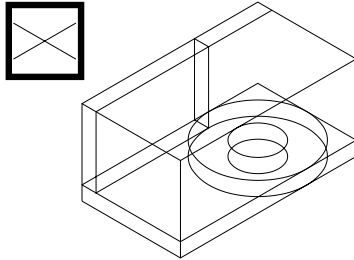


Figure A-21A

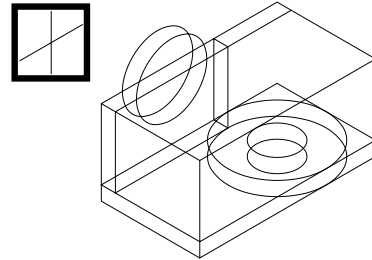


Figure A-21B

Use the `TRIM` command to trim away any excess objects that are considered unnecessary, as shown in Figure A-21C.

Use the `LINE` command to connect endpoints of edges that form surfaces and from tangents of arcs to tangents of arcs on parallel planes, as shown in Figure A-21D.

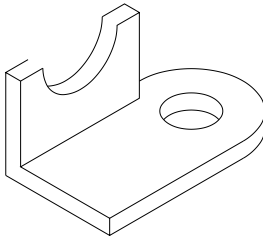


Figure A-21C

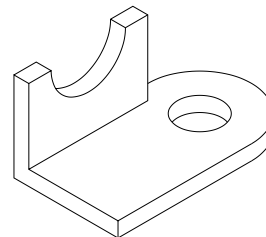


Figure A-21D

Try It! - Open the drawing file `A_Iso Ellipse2`. Complete the isometric drawing until your image is similar to Figure A-21D.

CREATING ISOMETRIC SECTIONS

In some cases, it is necessary to cut an isometric drawing to expose internal features. The result is an isometric section similar to a section view formed from an orthographic drawing. The isometric section differs, however, because the cutting plane is usually along one of the three isometric axes. Figure A-22A displays an orthographic section in addition to the isometric drawing.

The isometric in Figure A-22B has additional lines representing surfaces cut by the cutting plane line. The lines to define these surfaces are formed through the `LINE` command

in addition to a combination of Top and Right isometric axis modes. During this process, Ortho mode is toggled on and off numerous times depending on the axis direction. Use the TRIM command to remove objects from the half that will eventually be discarded.

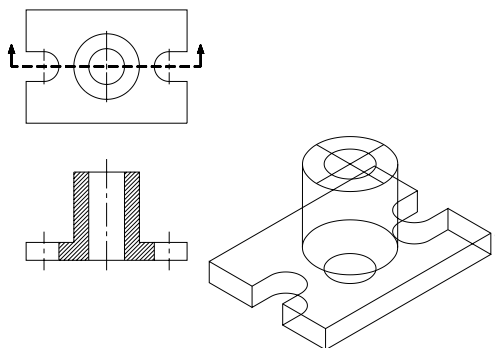


Figure A-22A

Once ellipses are trimmed, the remaining objects representing the front half of the isometric are removed, exposing the back half. The front half is then discarded. This has the same effect as conventional section views where the direction of sight dictates which half to keep. For a full section, the BHATCH command with the angle set at 15 degrees is used to draw section lines on the surfaces being cut by the cutting plane line, as in Figure A-22C. Surfaces designated holes or slots not cut are not cross-hatched. This same procedure is followed for converting an object to a half section.

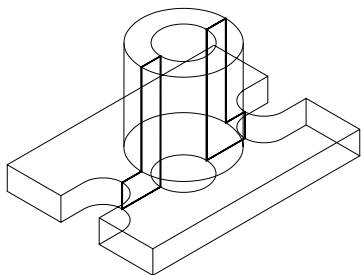


Figure A-22B

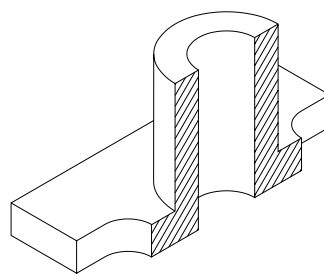


Figure A-22C

Try It! - Open the drawing file A_Iso Section. Use the previous figures as guides in constructing the isometric section in Figure A-22C.

EXPLODED ISOMETRIC VIEWS

Isometric drawings are sometimes grouped together to form an exploded drawing showing how a potential or existing product is assembled (see Figure A-23). This involves aligning parts that fit with line segments, usually in the form of centerlines. Bubbles identifying the part number are attached to the drawing. Exploded isometric drawings

come in handy for creating bill of material information and, for this purpose, they have an important application to manufacturing. Once the part information is identified in the drawing and title block area, this information is extracted and brought into a third-party business package where important data collection information is used to actually track the status of parts in production in addition to the shipping date for all finished products.

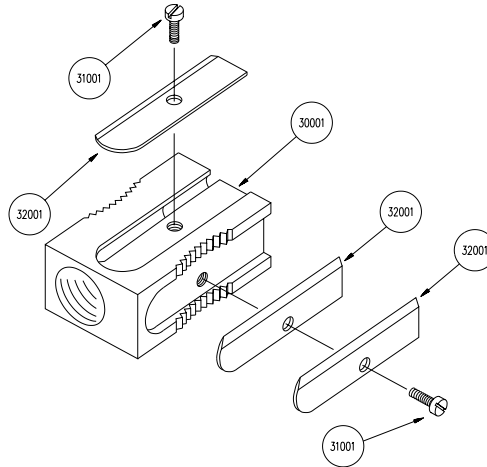


Figure A-23

ISOMETRIC ASSEMBLIES

Assembly drawings show the completed part as if it were to be assembled (see Figure A-24). Sometimes this drawing has an identifying number placed with a bubble for bill of material needs. Assembly drawings are commonly placed on the same display screen as the working drawing. With the assembly alongside the working drawing, you have a pictorial representation of what the final product will look like, and this can help you to understand the orthographic views.

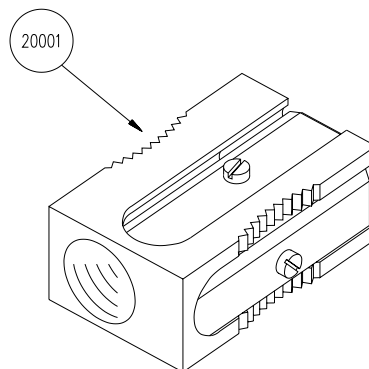


Figure A-24

TUTORIAL EXERCISE: PLATE.DWG

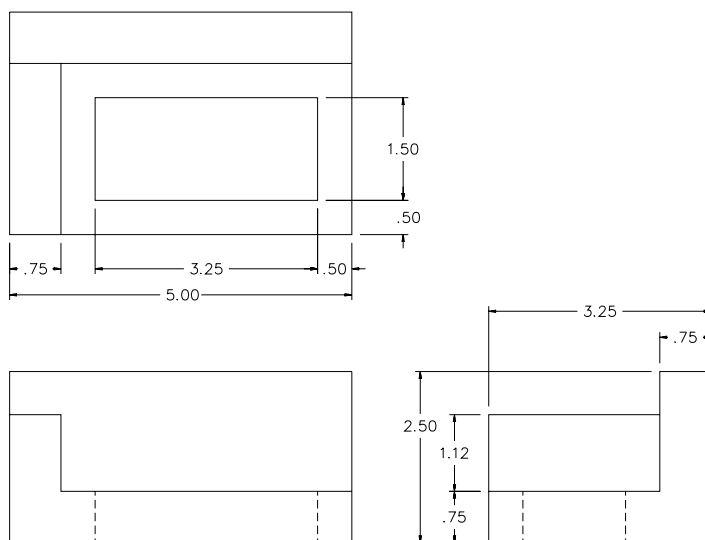


Figure A-25

Purpose

The purpose of this tutorial exercise is to use a series of coordinates along with AutoCAD editing commands to construct an isometric drawing of the plate in Figure A-25.

System Settings

Begin a new drawing called "Plate." Use the Drawing Units dialog box to change the number of decimal places past the zero from four to two. Keep the remaining default unit values. Using the LIMITS command, keep (0,0) for the lower left corner and change the upper right corner from (12.00,9.00) to (10.50,8.00). Perform a ZOOM-ALL after changing limits. Use the GRID command and change the grid spacing from 0.50 to 0.25 units. Do not turn the Snap or Ortho modes on. Check to see that the following Object Snap modes are already set: Endpoint, Extension, Intersection, and Center.

Layers

Create a layer called "Object" and assign this layer the color Green.

Suggested Commands

Begin this exercise by changing the grid from the standard display to an isometric display using the SNAP-Style option. Use absolute and polar coordinates to lay out the base of the plate. Then begin using the COPY command followed by TRIM to duplicate objects and clean up or trim unnecessary geometry.

Whenever possible, substitute the appropriate command alias in place of the full AutoCAD command in each tutorial step. For example, use "CP" for the COPY command, "L" for the LINE command, and so on. The complete listing of all command aliases is located in Chapter 1, Table 1-2.



STEP 1

Set a new polar tracking angle of 30° through the Polar Tracking tab of the Drafting Settings dialog box in Figure A–26A. Make the Object layer current. Then use the **LINE** command to draw the object in Figure A–26B. Notice, in the prompt sequence, that the Direct Distance mode is used because of the polar tracking angle setting of 30°. Also, be sure to turn Polar mode on by clicking on the **POLAR** button in the status bar.

Command: **L** (For **LINE**)

Specify first point: **5.50,0.75**

Specify next point or [Undo]: (Move your cursor up and to the right until the polar angle tooltip reads 30 degrees and enter **3.25**)

Specify next point or [Undo]: (Move your cursor up and to the left until the polar angle tooltip reads 150 degrees and enter **5.00**)

Specify next point or [Close/Undo]: (Move your cursor down and to the left until the polar angle tooltip reads 210 degrees and enter **3.25**)

Specify next point or [Close/Undo]: **C**
(To close the shape and exit the command)

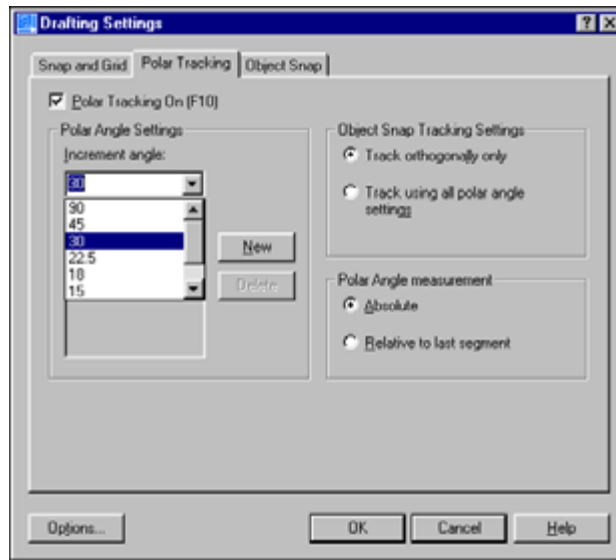


Figure A–26A

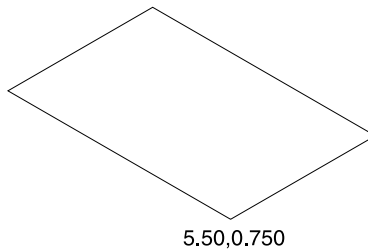


Figure A–26B

STEP 2

Copy the four lines drawn in Step 1 up a distance of 2.50 units in the 90° direction in Figure A-27.

Command: **CP** (For **COPY**)

Select objects: (Select lines "A", "B", "C", and "D")

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or
[Multiple]: (Pick the endpoint of the line at "A")

Specify second point of displacement or
<use first point as displacement>:
(Move your cursor up until the polar angle tooltip reads 90° and enter 2.50)

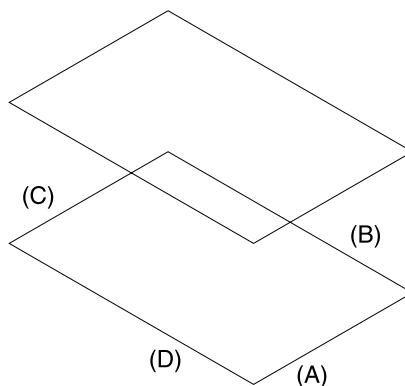


Figure A-27

STEP 3

Connect the top and bottom isometric boxes with line segments, as shown in Figure A-28. Draw one segment using the **LINE** command. Repeat this process using the **LINE** command at points "C" and "D," or use the **COPY** command and Multiple option to duplicate and form the remaining segments at "C" and "D." Erase the two dashed lines because they are not visible in an isometric drawing.

Command: **L** (For **LINE**)

Specify first point: (Select the endpoint of the line at "A")

Specify next point or [Undo]: (Select the endpoint of the line at "B")

Specify next point or [Undo]: (Press **ENTER** to exit this command)

Command: **E** (For **ERASE**)

Select objects: (Select the two dashed lines in Figure A-28)

Select objects: (Press **ENTER** to execute this command)

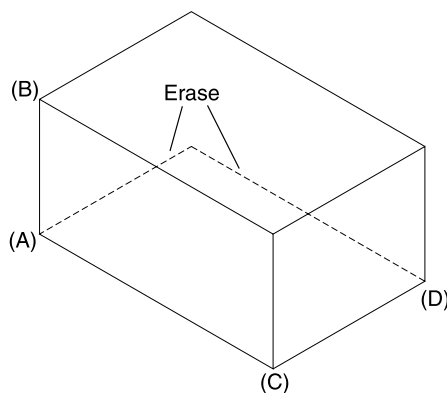


Figure A-28

STEP 4

Copy the two dashed lines in Figure A-29 a distance of 0.75 in the 210° direction.

Command: **CP** (For COPY)

Select objects: (Select the two dashed lines in Figure A-29)

Select objects: (Press ENTER to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:
(Move your cursor down and to the left until the polar angle tooltip reads 210° and enter 0.75)

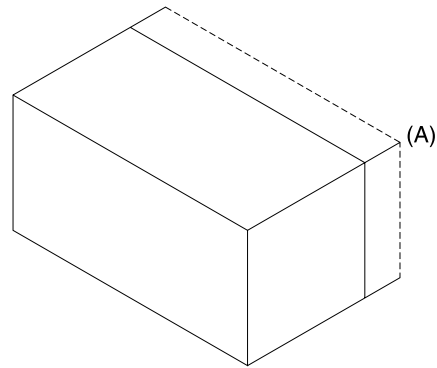


Figure A-29

STEP 5

Copy the two dashed lines in Figure A-30 a distance of 0.75 in the 90° direction. This forms the base of the plate.

Command: **CP** (For COPY)

Select objects: (Select the two dashed lines in Figure A-30)

Select objects: (Press ENTER to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:
(Move your cursor up until the polar angle tooltip reads 90° and enter 0.75)

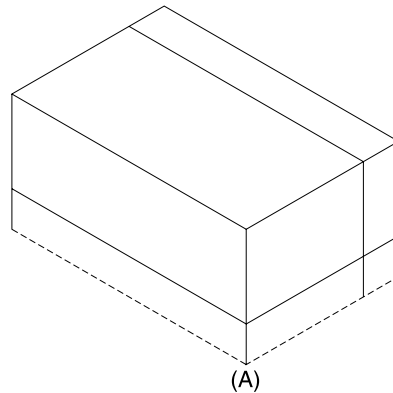


Figure A-30

STEP 6

Copy the dashed line in Figure A-31 a distance of 0.75 in the 330° direction.

Command: **CP** (For COPY)

Select objects: (Select the dashed line in Figure A-31)

Select objects: (Press ENTER to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:
(Move your cursor down and to the right
until the polar angle tooltip reads
330° and enter 0.75)

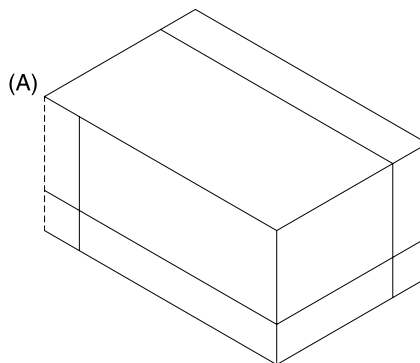


Figure A-31

STEP 7

Use the FILLET command to place a corner between the two dashed lines at "A" and "B" and at "C" and "D" in Figure A-32. Set the fillet radius to a value of 0 to accomplish this.

Command: **F** (For FILLET)

Current settings: Mode = TRIM, Radius = 0.50

Select first object or [Polyline/Radius/Trim]: **R** (For Radius)

Specify fillet radius <0.50>: **0**

Select first object or [Polyline/Radius/Trim]: (Select line "A")

Select second object: (Select line "B")

Command: **F** (For FILLET)

Current settings: Mode = TRIM, Radius = 0.00

Select first object or [Polyline/Radius/Trim]: (Select line "C")

Select second object: (Select line "D")

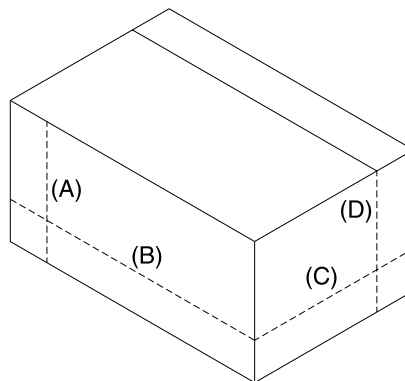


Figure A-32

STEP 8

Copy the dashed line in Figure A-33 to begin forming the top of the base.

Command: **CP** (For COPY)

Select objects: (Select the dashed line in Figure A-33)

Select objects: (Press ENTER to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:
(Select the endpoint at "B")

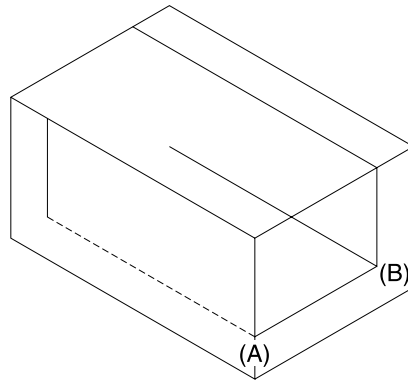


Figure A-33

STEP 9

Copy the dashed line in Figure A-34. This forms the base of the plate.

Command: **CP** (For COPY)

Select objects: (Select the dashed line in Figure A-34)

Select objects: (Press ENTER to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:
(Select the endpoint at "B")

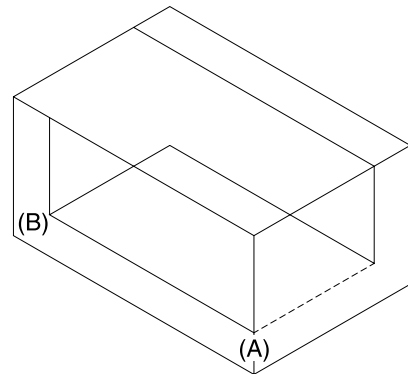


Figure A-34

STEP 10

Use the **TRIM** command to clean up the excess lines in Figure A-35.

Command: **TR** (For **TRIM**)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select the three dashed lines shown in Figure A-35)

Select objects: (Press **ENTER** to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the inclined line at "A")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the inclined line at "B")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the inclined line at "C")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press **ENTER** to exit this command)

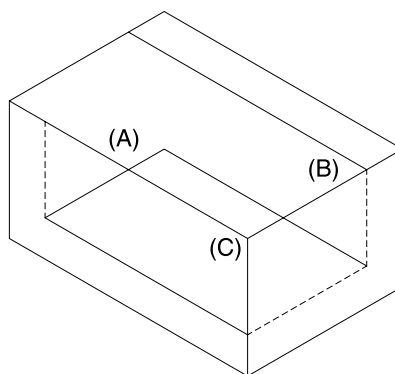


Figure A-35

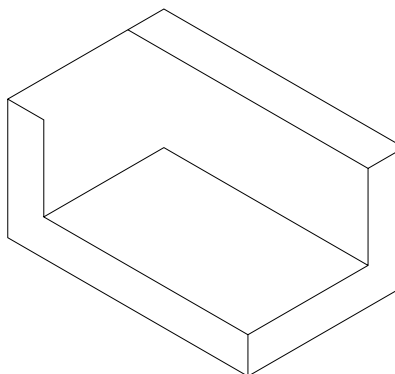


Figure A-36

STEP 11

Your display should be similar to Figure A-36.

STEP 12

Copy the dashed line in Figure A-37 a distance of 1.12 units in the 90° direction.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-37)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or [Multiple]: (Select the endpoint at "A")

Specify second point of displacement or <use first point as displacement>:
(Move your cursor up until the polar angle tooltip reads 90° and enter 1.12)

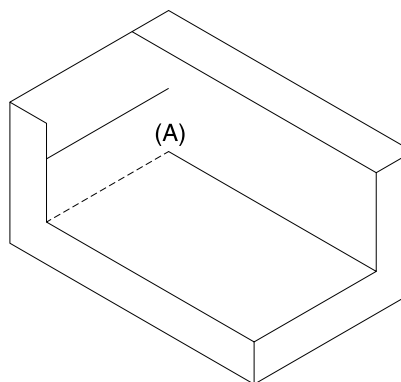


Figure A-37

STEP 13

As shown in Figure A–38, copy the dashed line at “A” to new positions at “B” and “C.” Then delete the line at “A” using the ERASE command.

Command: **CP** (For COPY)

Select objects: (Select the dashed line in Figure A–38)

Select objects: (Press ENTER to continue)

Specify base point or displacement, or [Multiple]: **M** (For Multiple)

Specify base point: (Select the endpoint at “A”)

Specify second point of displacement or <use first point as displacement>:
(Select the endpoint at “B”)

Specify second point of displacement or <use first point as displacement>:
(Select the endpoint at “C”)

Specify second point of displacement or <use first point as displacement>:
(Press ENTER to exit this command)

Command: **E** (For ERASE)

Select objects: (Select the dashed line at “A”)

Select objects: (Press ENTER to execute this command)

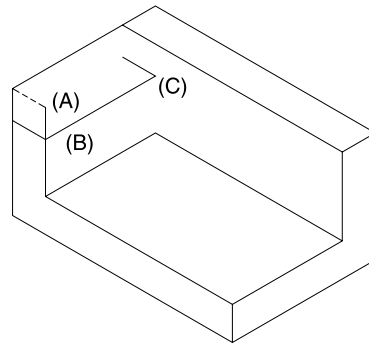


Figure A–38

STEP 14

Use the TRIM command to clean up the excess lines in Figure A–39.

Command: **TR** (For TRIM)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select the two dashed lines in Figure A–39)

Select objects: (Press ENTER to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the vertical line at “A”)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the vertical line at “B”)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the inclined line at “C”)

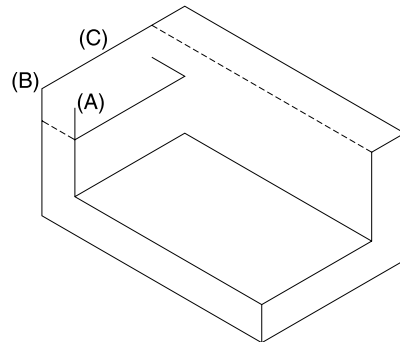


Figure A–39

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press ENTER to exit this command)

STEP 15

Use the **COPY** command to duplicate the dashed line in Figure A-40 from the endpoint of "A" to the endpoint at "B."

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-40)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:
(Select the endpoint at "B")

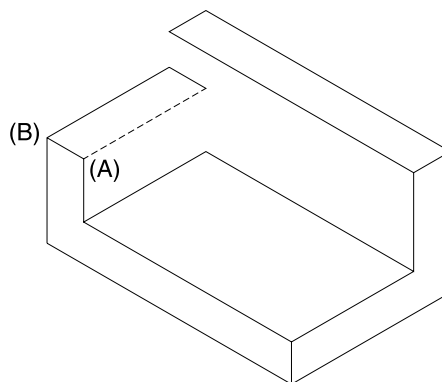


Figure A-40

STEP 16

Use the **COPY** command to duplicate the dashed line in Figure A-41 from the endpoint of "A" to the endpoint at "B."

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-41)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:
(Select the endpoint at "B")

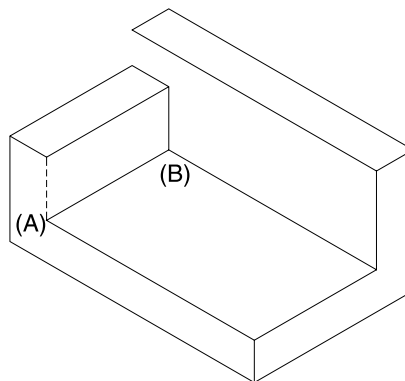


Figure A-41

STEP 17

Use the **LINE** command to connect the endpoints of the segments at “A” and “B” in Figure A-42.

Command: **L** (For **LINE**)

Specify first point: (Select the endpoint at “A”)

Specify next point or [Undo]: (Select the endpoint at “B”)

Specify next point or [Undo]: (Press **ENTER** to exit this command)

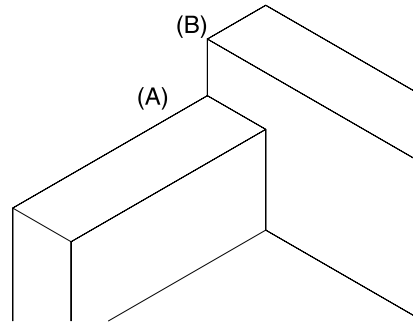


Figure A-42

STEP 18

Use the **COPY** command to duplicate the dashed line in Figure A-43 from the endpoint of “A” a distance of 0.50 units in the 210° direction. This value begins the outline of the rectangular hole through the object.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-43)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or [Multiple]: (Select the endpoint at “A”)

Specify second point of displacement or <use first point as displacement>:
(Move your cursor down and to the left until the polar angle tooltip reads 210° and enter **0.50**)

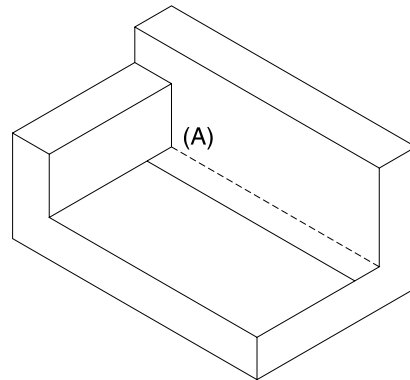


Figure A-43

STEP 19

Use the **COPY** command to duplicate the dashed line in Figure A-44 from the endpoint of "A" a distance of 0.50 in the 330° direction.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-44)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or

[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:

(Move your cursor down and to the right until the polar angle tooltip reads 330° and enter **0.50**)

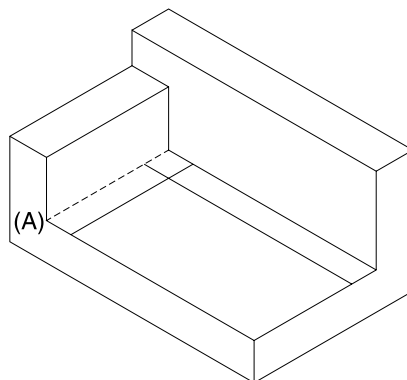


Figure A-44

STEP 20

Use the **COPY** command to duplicate the dashed line in Figure A-45 from the endpoint of "A" a distance of 0.50 in the 30° direction.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-45)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or

[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:

(Move your cursor up and to the right until the polar angle tooltip reads 30° and enter **0.50**)

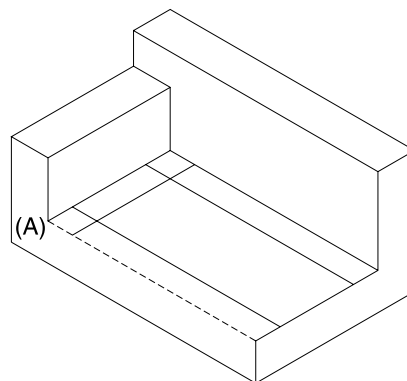


Figure A-45

STEP 21

Use the **COPY** command to duplicate the dashed line in Figure A-46 from the endpoint of "A" a distance of 0.50 in the 150° direction.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-46)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or [Multiple]: (Select the endpoint at "A")

Specify second point of displacement or <use first point as displacement>:

(Move your cursor up and to the right until the polar angle tooltip reads 150° and enter **0.50**)

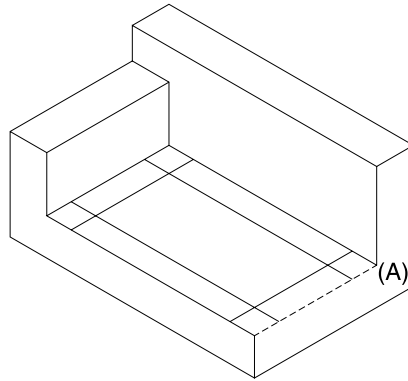


Figure A-46

STEP 22

Use the **FILLET** command with a radius of 0 to corner the four dashed lines in Figure A-47. Use the **MULTIPLE** command to remain in the **FILLET** command. To exit the **FILLET** command prompts, press **ESC** to cancel the command when finished.

Command: **MULTIPLE**

Enter command name to repeat: **F** (For **FILLET**)

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Polyline/Radius/Trim]: (Select the line at "A")

Select second object: (Select the line at "B")

FILLET

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Polyline/Radius/Trim]: (Select the line at "B")

Select second object: (Select the line at "C")

FILLET

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Polyline/Radius/Trim]: (Select the line at "C")

Select second object: (Select the line at "D")

FILLET

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Polyline/Radius/Trim]: (Select the line at "D")

Select second object: (Select the line at "A")

FILLET

Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Polyline/Radius/Trim]: (Press **ESC** to cancel this command)

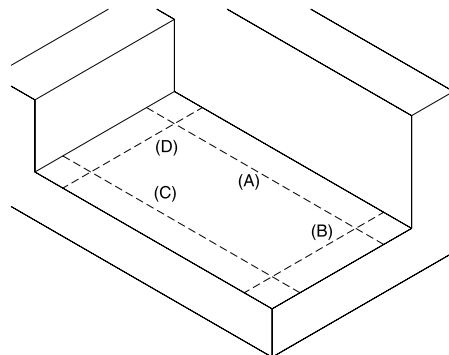


Figure A-47

STEP 23

Use the **COPY** command to duplicate the dashed line in Figure A-48. This will begin forming the thickness of the base inside the rectangular hole.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-48)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or

[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:

(Select the endpoint at "B")

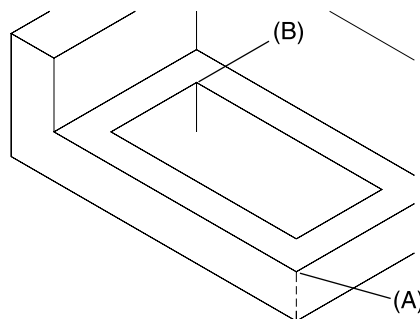


Figure A-48

STEP 24

Use the **COPY** command to duplicate the dashed lines in Figure A-49. These lines form the inside surfaces to the rectangular hole.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed lines in Figure A-49)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or

[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:

(Select the endpoint at "B")

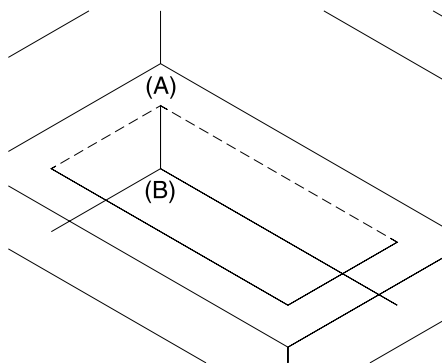


Figure A-49

STEP 25

Use the **TRIM** command to clean up excess lines in Figure A-50.

Command: **TR** (For **TRIM**)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select the two dashed lines in Figure A-50)

Select objects: (Press **ENTER** to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the line at "A")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the line at "B")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press **ENTER** to exit this command)

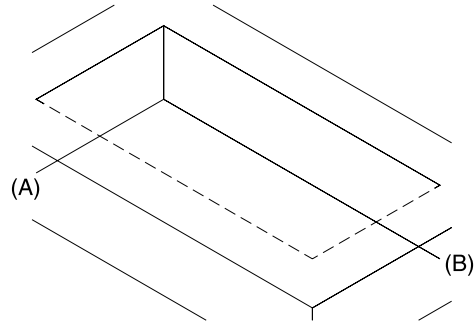


Figure A-50

STEP 26

The completed isometric is illustrated in Figure A-51. This drawing may be dimensioned with the **Oblique** option of the **DIMEDIT** command. Consult your instructor if this next step is necessary.

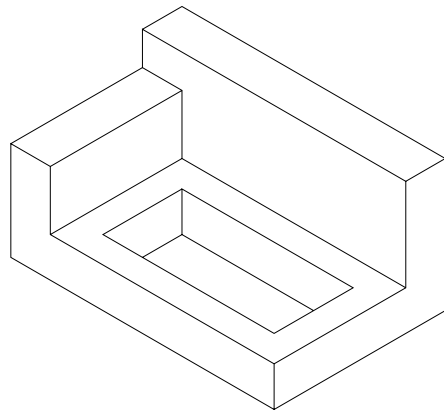
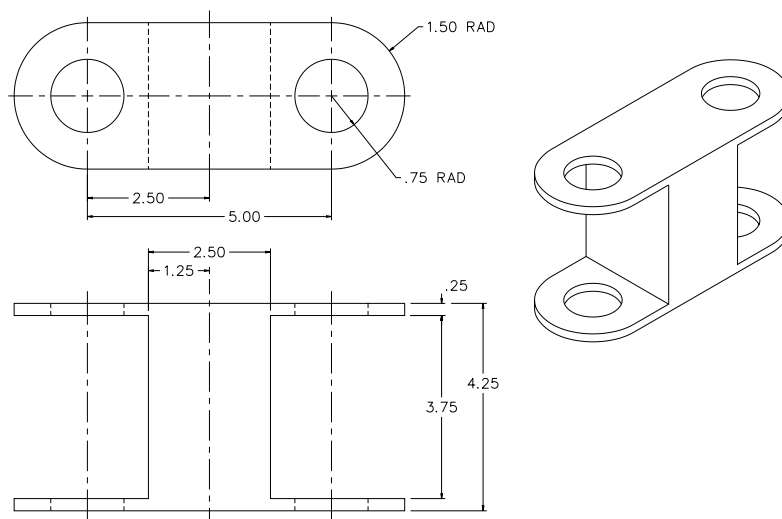


Figure A-51

TUTORIAL EXERCISE: HANGER.DWG**Figure A-52****Purpose**

The purpose of this tutorial exercise is to use a series of coordinates along with AutoCAD editing commands to construct an isometric drawing of the hanger in Figure A-52.

System Settings

Begin a new drawing called "Hanger." Use the Drawing Units dialog box to change the number of decimal places past the zero from four to two. Keep the remaining default unit values. Using the LIMITS command, keep (0,0) for the lower left corner and change the upper right corner from (12,9) to (15.50,9.50). Perform a ZOOM-ALL after changing Limits. Use the GRID command and change the grid spacing from 0.50 to 0.25 units. Turn Snap mode off; turn Ortho mode on. Check to see that the following Object Snap modes are already set: End-point, Extension, Intersection, Center.

Layers

Create a layer called "Object" and assign this layer the color Green.

Suggested Commands

Begin this exercise by changing the grid from the standard display to an isometric display using the SNAP-Style option. Remember, both the grid and snap can be manipulated by the SNAP command only if the current grid value is 0. Use absolute and polar coordinates to lay out the base of the hanger. Then begin using the COPY command followed by TRIM to duplicate objects and clean up or trim unnecessary geometry.

Whenever possible, substitute the appropriate command alias in place of the full AutoCAD command in each tutorial step. For example, use "CP" for the COPY command, "L" for the LINE command, and so on. The complete listing of all command aliases is located in Chapter 1, Table 1-2.



STEP 1

Make the Object layer current. Then set the SNAP-Style option to Isometric with a vertical spacing of 0.25 units. Press CTRL+E or F5 to switch to the Top Isoplane mode. Use the LINE command to draw the rectangular isometric box representing the total depth of the object along with the center-to-center distance of the holes and arcs that will be placed in the next step. See Figure A-53.

Command: **SN** (For SNAP)

Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <0.50>: **S** (For Style)

Enter snap grid style [Standard/Isometric] <S>: **I** (For Isometric)

Specify vertical spacing <0.50>: **0.25**

Command: (Press CTRL+E to switch to the Top Isoplane mode)

Command: **L** (For LINE)

Specify first point: **6.00,0.50**

Specify next point or [Undo]: (Move your cursor up and to the right and enter 5.00)

Specify next point or [Undo]: (Move your cursor up and to the left and enter 3.00)

Specify next point or [Close/Undo]: (Move your cursor down and to the left and enter 5.00)

Specify next point or [Close/Undo]: **C** (To close the shape and exit the command)

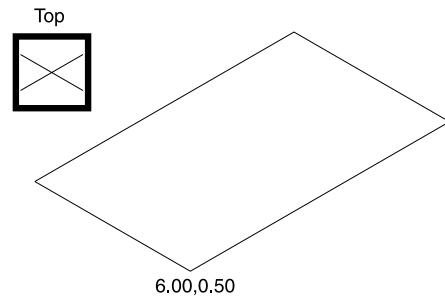


Figure A-53

STEP 2

While in the Top Isoplane mode, use the ELLIPSE command to draw two isometric ellipses of 0.75 and 1.50 radii each in Figure A-54. Identify the midpoint of the inclined line at "A" as the center of the first ellipse. To identify the center of the second ellipse, use the @ option, which stands for "last point" and will identify the center of the small circle as the same center as the large circle.

Command: **EL** (For ELLIPSE)

Specify axis endpoint of ellipse or [Arc/Center/Isocircle]: **I** (For Isocircle)

Specify center of isocircle: **Mid**

of (Pick the midpoint of the line at "A")

Specify radius of isocircle or [Diameter]: **0.75**

Command: **EL** (For ELLIPSE)

Specify axis endpoint of ellipse or [Arc/Center/Isocircle]: **I** (For Isocircle)

Specify center of isocircle: **@** (For the last point)

Specify radius of isocircle or [Diameter]: **1.50**

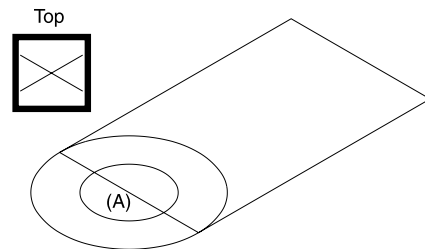


Figure A-54

STEP 3

Copy both ellipses from the endpoint of the inclined line at "A" to the endpoint of the inclined line at "B" in Figure A-55. Be sure Snap is turned off.

Command: **CP** (For COPY)

Select objects: (Select both ellipses in Figure A-55)

Select objects: (Press ENTER to continue)

Specify base point or displacement, or [Multiple]: (Select the endpoint at "A")

Specify second point of displacement or <use first point as displacement>: (Select the endpoint at "B")

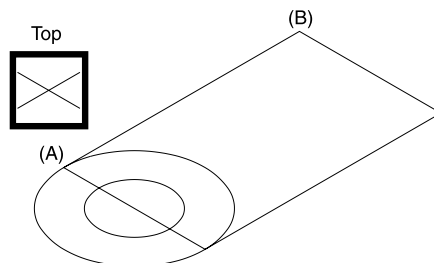


Figure A-55

STEP 4

Use the TRIM command to delete segments of the ellipses in Figure A-56.

Command: **TR** (For TRIM)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select dashed lines "A" and "B")

Select objects: (Press ENTER to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the ellipse at "C")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the ellipse at "D")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press ENTER to exit this command)

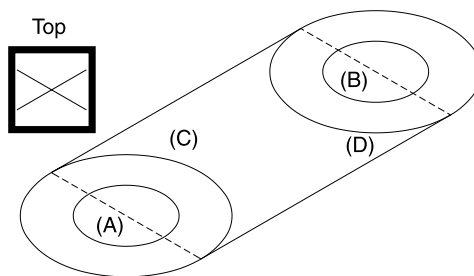


Figure A-56

STEP 5

Copy all objects in Figure A-57 up the distance of 0.25 units to form the bottom base of the hanger. Pressing CTRL+E or F5 can activate the Right Isoplane mode.

Command: *(Press CTRL+E To switch to the Right Isoplane mode)*

Right Isoplane

Command: **CP** *(For COPY)*

Select objects: *(Select all objects in Figure A-57)*

Select objects: *(Press ENTER to continue)*

Specify base point or displacement, or
[Multiple]: *(Select the endpoint at "A")*

Specify second point of displacement or
<use first point as displacement>:
(Move your cursor up and enter a value of 0.25)

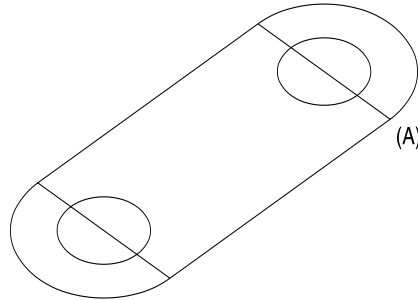


Figure A-57

STEP 6

Use the ERASE command to delete the three dashed lines in Figure A-58. These lines are not visible from this point of view and should be erased.

Command: **E** *(For ERASE)*

Select objects: *(Select the three dashed lines in Figure A-58)*

Select objects: *(Press ENTER to execute this command)*

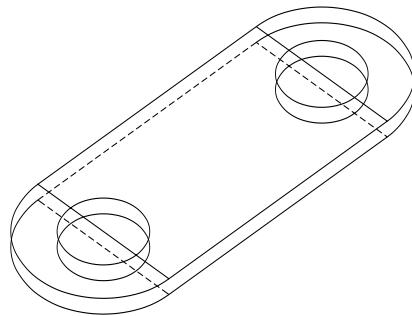
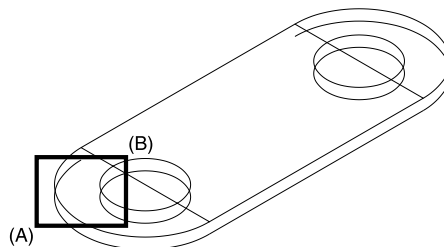


Figure A-58

STEP 7

Your display should appear similar to the illustration in Figure A-59. Begin partially deleting other objects to show only visible features of the isometric drawing. Check that the snap has been turned off to better assist in the next series of operations. The next few steps refer to the area outlined in Figure A-59. Use the ZOOM-Window option to magnify this area.

**Figure A-59**

Command: **Z** (For ZOOM)

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: (Pick a point at "A")

Specify opposite corner: (Pick a point at "B")

STEP 8

Carefully draw a line tangent to both ellipses. Use the OSNAP-Quadrant option to assist in constructing the line. See Figure A-60.

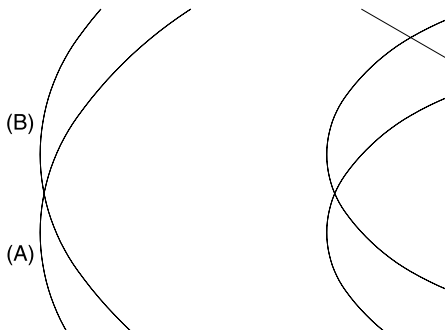
Command: **L** (For LINE)

Specify first point: **Qua**

of (Select the quadrant at "A")

Specify next point or [Undo]: (Pick the polar intersection at "B")

Specify next point or [Undo]: (Press ENTER to exit this command)

**Figure A-60**

STEP 9

Use the **TRIM** command, select the short dashed line as the cutting edge, and select the arc segment in Figure A-61 to trim.

Command: **TR** (For **TRIM**)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select the dashed line at "A")

Select objects: (Press **ENTER** to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the ellipse at "B")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press **ENTER** to exit this command)

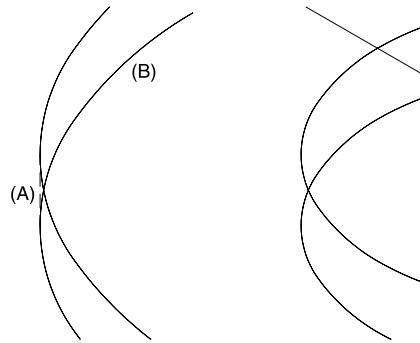


Figure A-61

STEP 10

The completed operation is in Figure A-62. Use **ZOOM** command and the **Previous** option to return to the previous display.

Command: **Z** (For **ZOOM**)

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: **P** (For **Previous**)

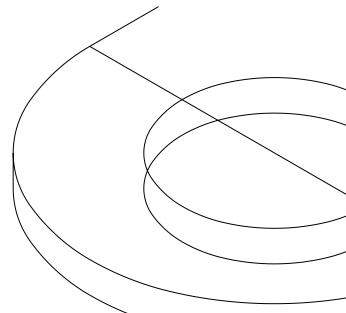


Figure A-62

STEP 11

Use **ZOOM-Window** to magnify the right half of the base in Figure A-63. Prepare to construct the tangent edge to the object using the procedure in Step 8.

Command: **Z** (For **ZOOM**)

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window] <real time>: (Pick a point at "A")

Specify opposite corner: (Pick a point at "B")

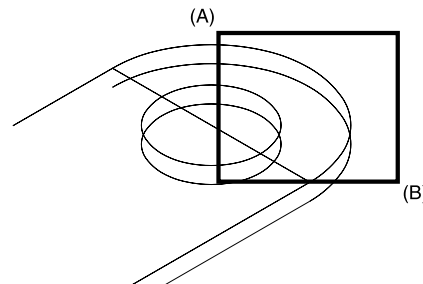
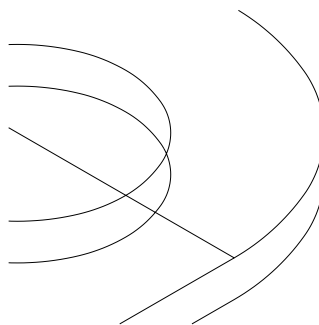


Figure A-63

STEP 12

Follow the same procedure as in Step 8 to construct a line from the quadrant point on the top ellipse to the polar intersection on the bottom ellipse. Then use the TRIM command in Step 9 to clean up any excess objects. Use the ERASE command to delete any elliptical arc segments that may have been left untrimmed. (See Figure A-64.)

**Figure A-64****STEP 13**

Zoom to the Previous display. Use the TRIM command to partially delete the ellipses in Figure A-65 to expose the thickness of the base.

Command: **TR** (For TRIM)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

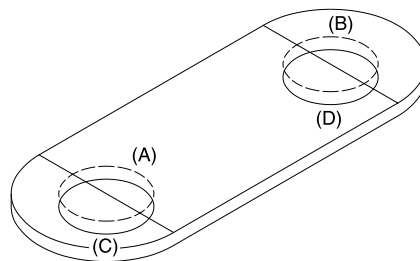
Select objects: (Select dashed ellipses "A" and "B")

Select objects: (Press ENTER to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the lower ellipse at "C")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the lower ellipse at "D")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press ENTER to exit this command)

**Figure A-65**

STEP 14

Use the **COPY** command to duplicate the bottom base and form the upper plate of the hanger as shown in Figure A-66. Copy the base a distance of 4 units straight up.

Command: **CP** (For **COPY**)

Select objects: (Select all dashed objects in Figure A-66)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint of the ellipse at "A")

Specify second point of displacement or
<use first point as displacement>:
(Move your cursor up and enter a value of **4.00**)

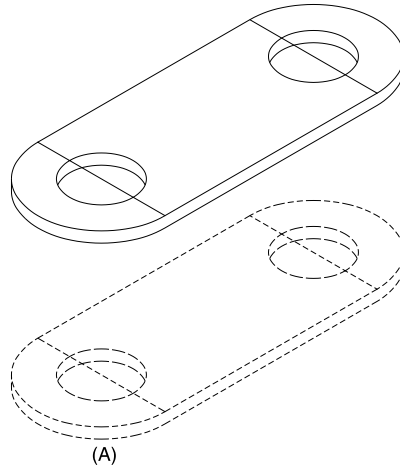


Figure A-66

STEP 15

Use the **COPY** command to duplicate the inclined line at "A" the distance of 2.50 units to form the line represented by a series of dashes in Figure A-67. This line happens to be located at the center of the object.

Command: **CP** (For **COPY**)

Select objects: (Select the line at "A")

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or
[Multiple]: (Select the endpoint at "A")

Specify second point of displacement or
<use first point as displacement>:
(Move your cursor up and to the right and enter **2.50**)

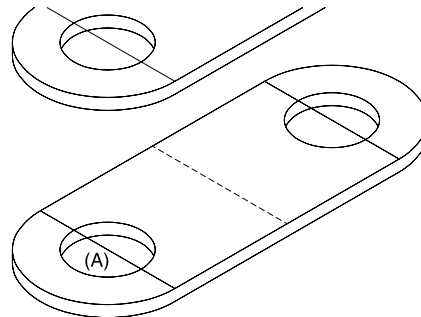


Figure A-67

STEP 16

Duplicate the line represented by dashes in Figure A-68 to form the two inclined lines at "B" and "C." These lines will begin the construction of the sides of the hanger. Use the **COPY** command along with the Multiple option to accomplish this.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-68)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or [Multiple]: **M** (For Multiple)

Specify base point: (Select the endpoint at "A")

Specify second point of displacement or <use first point as displacement>:

(Move your cursor up and to the right and enter **1.25**)

Specify second point of displacement or <use first point as displacement>:

(Move your cursor down and to the left and enter **1.25**)

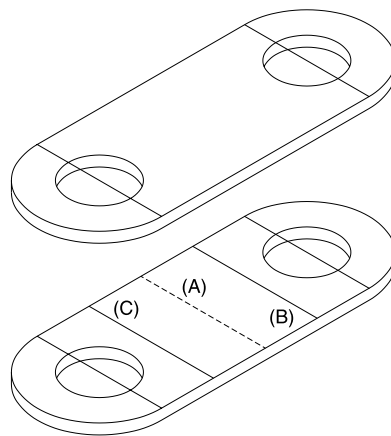


Figure A-68

Specify second point of displacement or <use first point as displacement>:
(Press **ENTER** to exit this command)

STEP 17

Use the **COPY** command to duplicate the two dashed lines in Figure A-69 straight up at a distance of 3.75 units. Use the **Polar** coordinate mode to accomplish this. Make sure the **Polar** mode is turned On.

Command: **CP** (For **COPY**)

Select objects: (Select both dashed lines in Figure A-69)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or [Multiple]: (Select the endpoint at "A")

Specify second point of displacement or <use first point as displacement>: (Pick the Polar Intersection at "B")

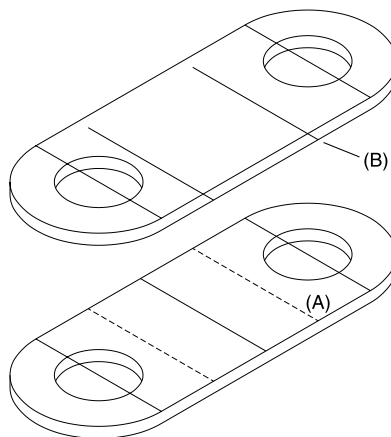


Figure A-69

STEP 18

Use the **LINE** command along with the **OSNAP-Endpoint** option to draw a line from endpoint “A” to endpoint “B” in Figure A-70.

Command: **L** (For **LINE**)

Specify first point: (Select the endpoint at “A”)

Specify next point or [Undo]: (Select the endpoint at “B”)

Specify next point or [Undo]: (Press **ENTER** to exit this command)

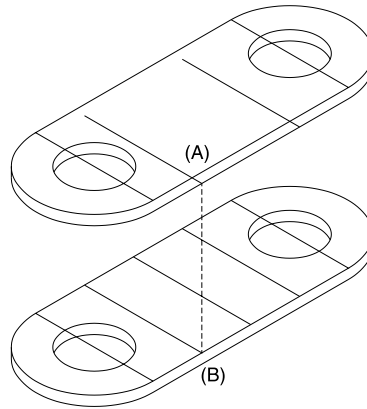


Figure A-70

STEP 19

Use the **COPY** command and the **Multiple** option to copy the dashed line at “A” to “B,” “C,” and “D” in Figure A-71.

Command: **CP** (For **COPY**)

Select objects: (Select the dashed line in Figure A-71)

Select objects: (Press **ENTER** to continue)

Specify base point or displacement, or [Multiple]: **M** (For **Multiple**)

Specify base point: (Select the endpoint of the vertical line at “A”)

Specify second point of displacement or <use first point as displacement>:
(Select the endpoint of the line at “B”)

Specify second point of displacement or <use first point as displacement>:
(Select the endpoint of the line at “C”)

Specify second point of displacement or <use first point as displacement>:
(Select the endpoint of the line at “D”)

Specify second point of displacement or <use first point as displacement>:
(Press **ENTER** to exit this command)

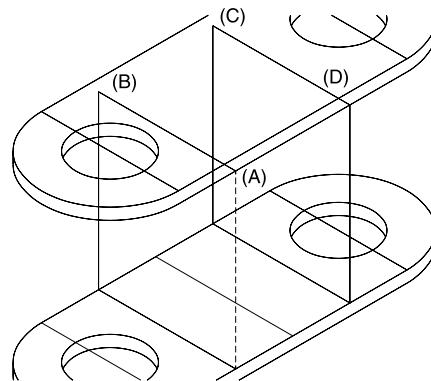


Figure A-71

STEP 20

Use the **ERASE** command to delete all lines represented as dashed lines in Figure A-72.

Command: **E** (For **ERASE**)

Select objects: (Select all dashed objects in Figure A-72)

Select objects: (Press **ENTER** to execute this command)

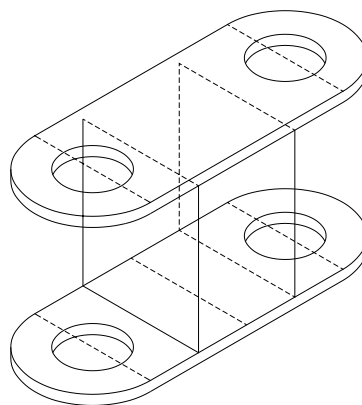


Figure A-72

STEP 21

Use the **TRIM** command to partially delete the vertical line in Figure A-73. The segment to be deleted is hidden and not shown in an isometric drawing.

Command: **TR** (For **TRIM**)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select the dashed objects at "A" and "B")

Select objects: (Press **ENTER** to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the vertical line at "C")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press **ENTER** to exit this command)

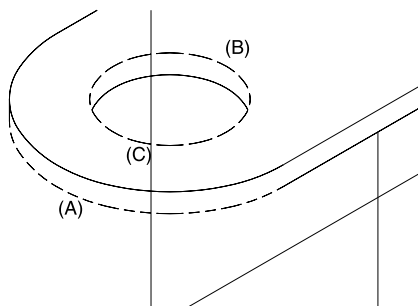


Figure A-73

STEP 22

Use the **TRIM** command to partially delete the vertical line in Figure A-74. The segment to be deleted is hidden and not shown in an isometric drawing.

Command: **TR** (For *TRIM*)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select the dashed elliptical arc at "A")

Select objects: (Press **ENTER** to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the vertical line at "B")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press **ENTER** to exit this command)

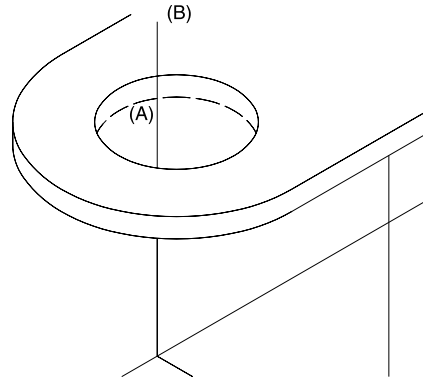


Figure A-74

STEP 23

Use the **TRIM** command to partially delete the inclined line in Figure A-75.

Command: **TR** (For *TRIM*)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select the dashed lines "A" and "B")

Select objects: (Press **ENTER** to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the line at "C")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the line at "D")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press **ENTER** to exit this command)

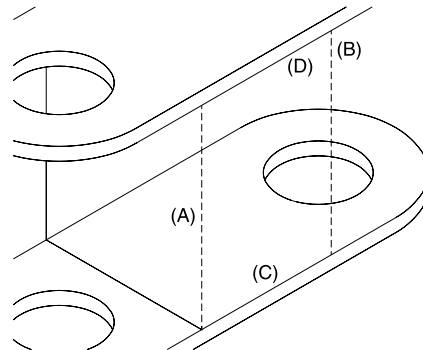


Figure A-75

STEP 24

Use the **TRIM** command to partially delete the objects in Figure A-76. The segments to be deleted are hidden and not shown in an isometric drawing. Use **ERASE** to delete any leftover elliptical arc segments.

Command: **TR** (For **TRIM**)

Current settings: Projection=UCS

Edge=None

Select cutting edges ...

Select objects: (Select the dashed lines "A" and "B")

Select objects: (Press **ENTER** to continue)

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the line at "C")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the elliptical arc at "D")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the ellipse at "E")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Select the elliptical arc at "F")

Select object to trim or shift-select to extend or [Project/Edge/Undo]: (Press **ENTER** to exit this command)

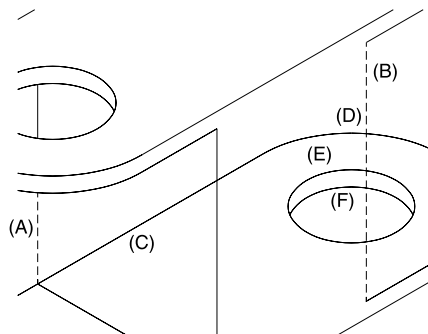


Figure A-76

STEP 25

The completed isometric appears in Figure A-77. This drawing may be dimensioned with the **Oblique** option of the **DIMEDIT** command. Consult your instructor if this next step is necessary.

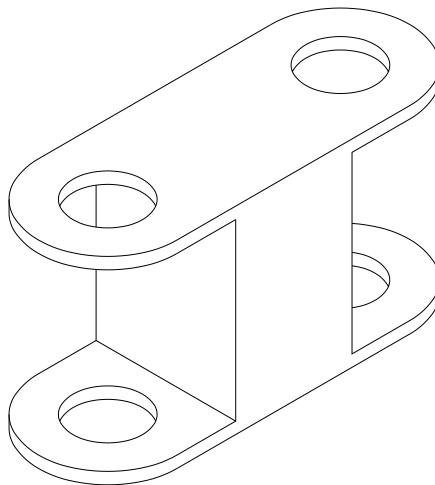


Figure A-77

STEP 26

As an extra step, convert the completed hanger to an object with a rectangular hole through its center in Figure A-78. Begin by copying lines “A,” “B,” “C,” and “D” a distance of 0.25 units to form the inside rectangle using polar coordinates. Since the lines will overlap at the corners, use the **FILLET** command set to a radius of 0 to create corners of the hole. Use the **LINE** command to draw the inclined line “E” at a 150° angle. Use Polar intersection to connect the “E” to “C” and complete the new version of the hanger.

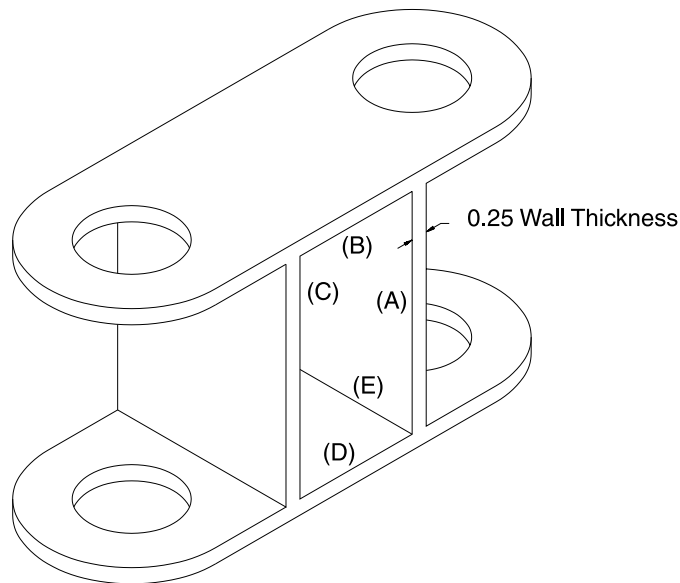


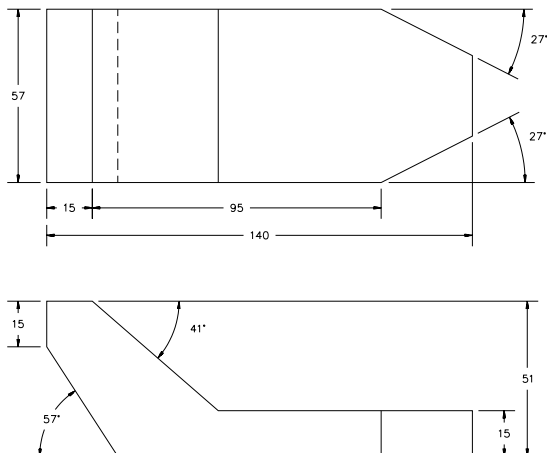
Figure A-78

PROBLEMS FOR CHAPTER A

Directions for Problem A-1:

Construct an isometric drawing of the object.

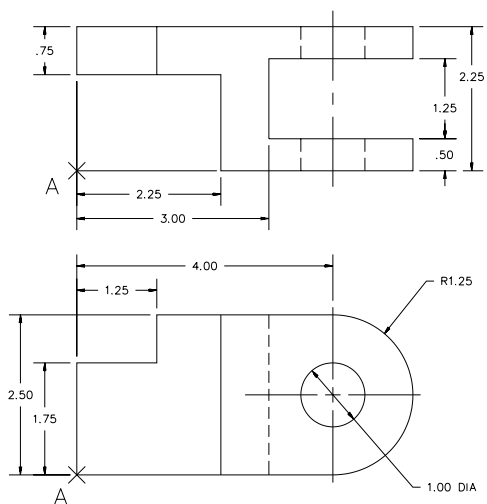
PROBLEM A-1



Directions for Problem A-2:

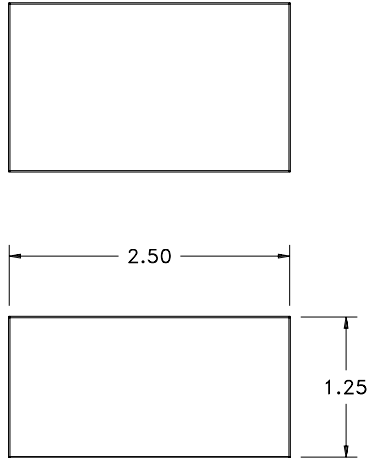
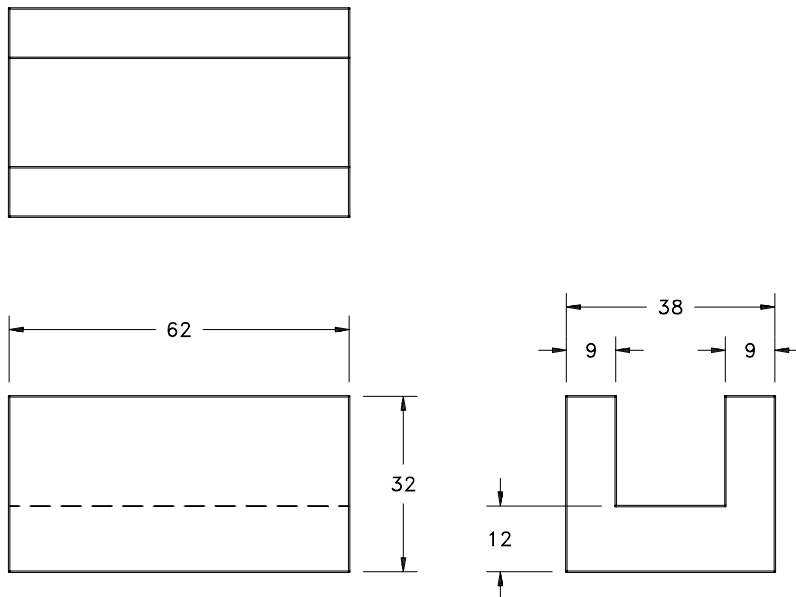
Construct an isometric drawing of the object. Begin the corner of the isometric at "A."

PROBLEM A-2

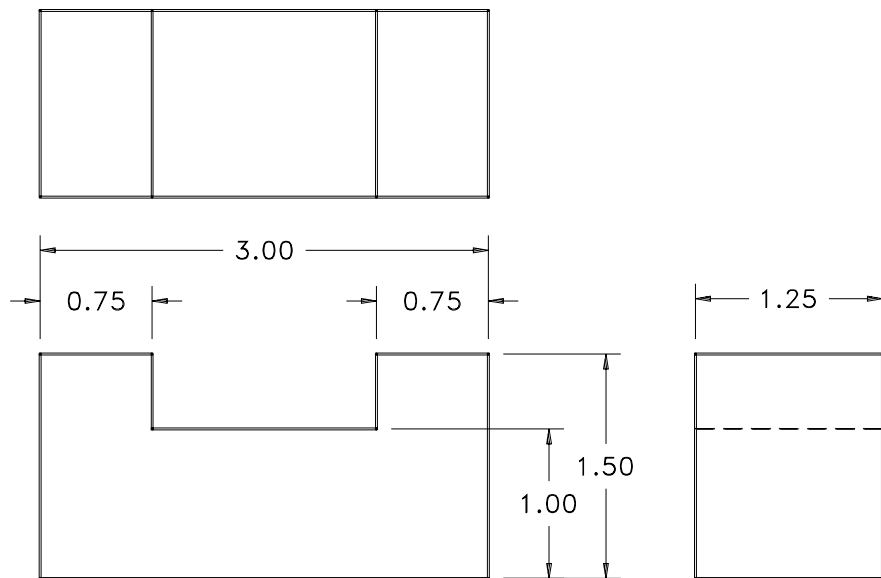


Directions for Problems A-3 through A-17:

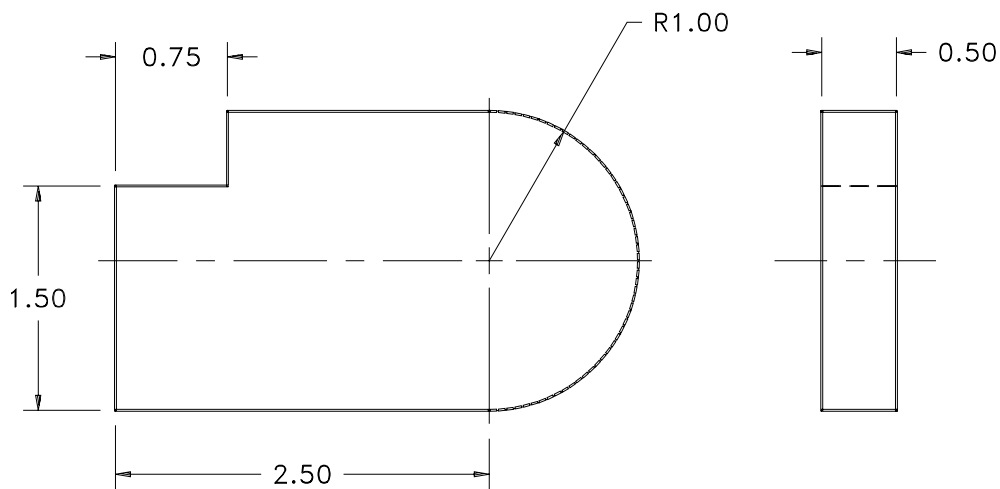
Construct an isometric drawing of the object.

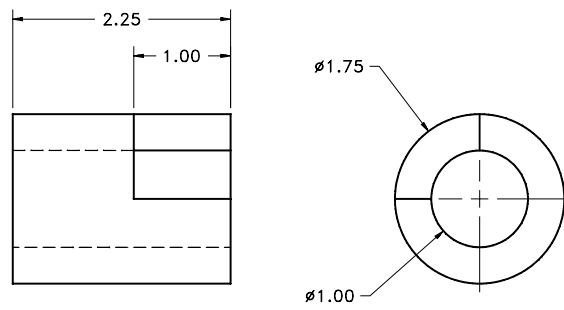
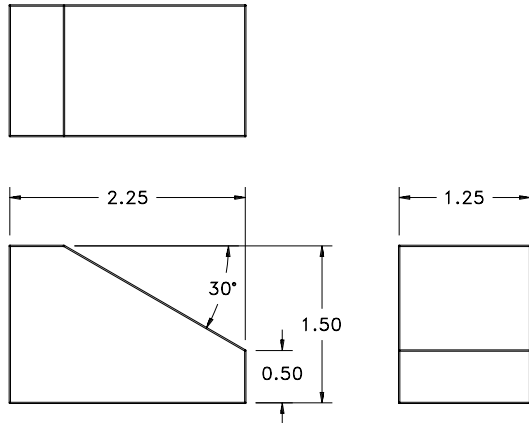
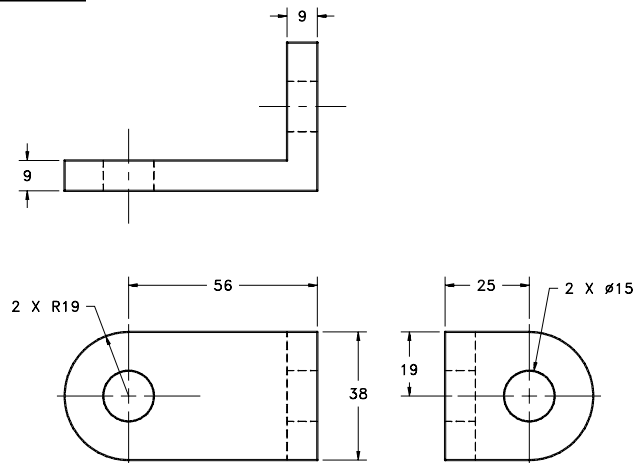
PROBLEM A-3**PROBLEM A-4**

PROBLEM A-5

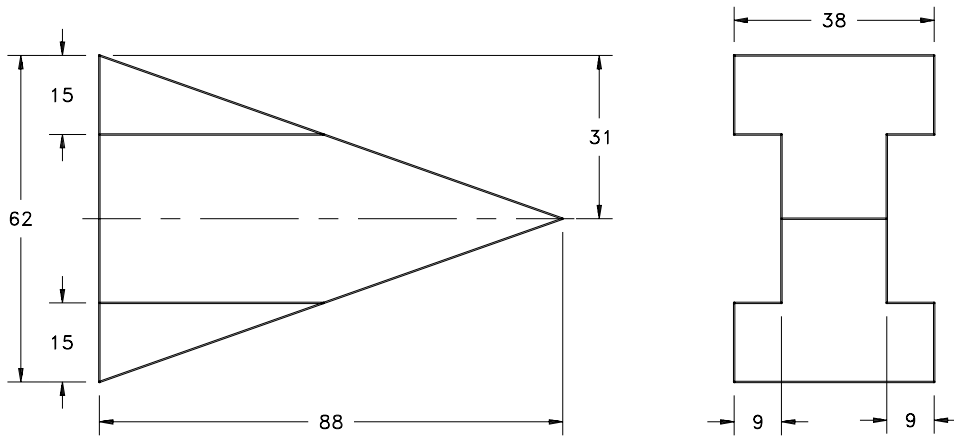


PROBLEM A-6

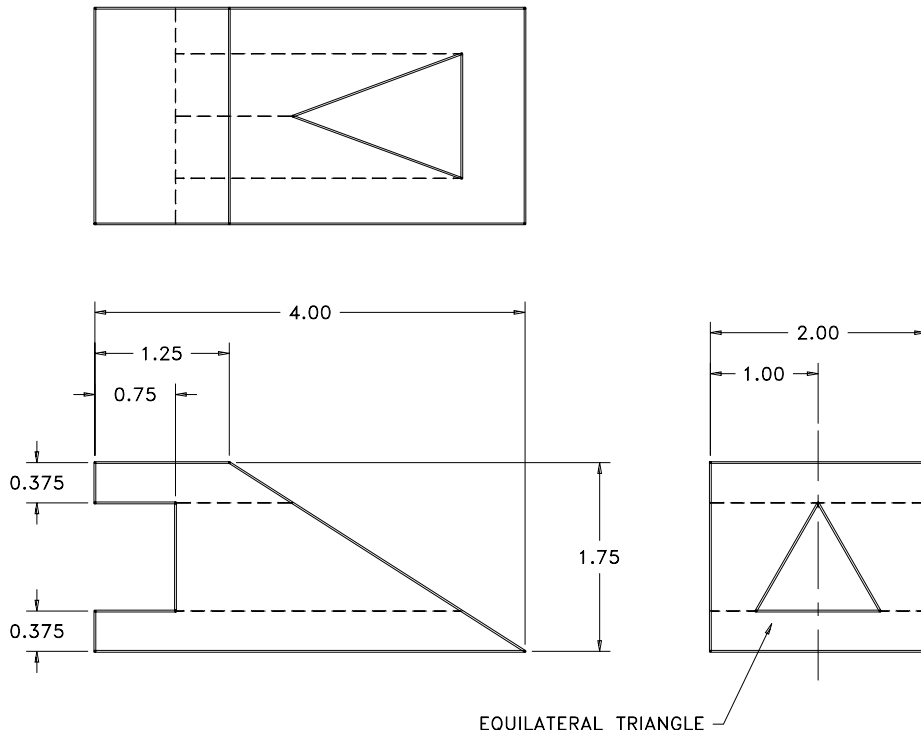


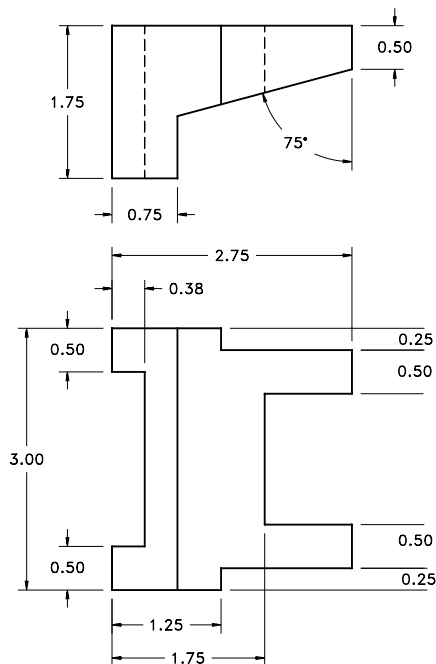
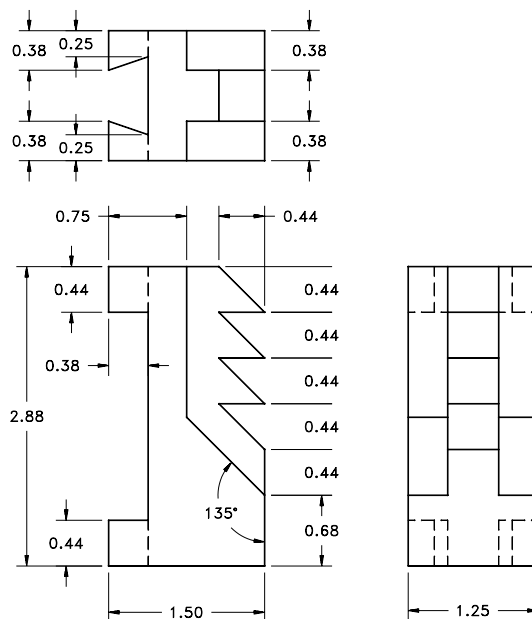
PROBLEM A-7**PROBLEM A-8****PROBLEM A-9**

PROBLEM A-10

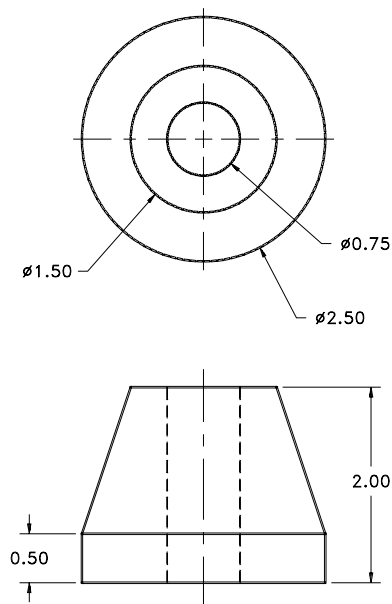


PROBLEM A-11

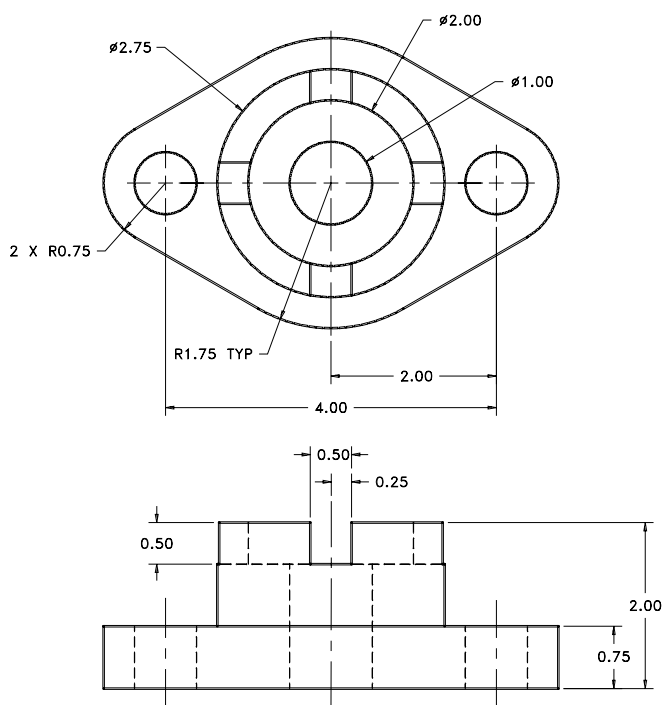


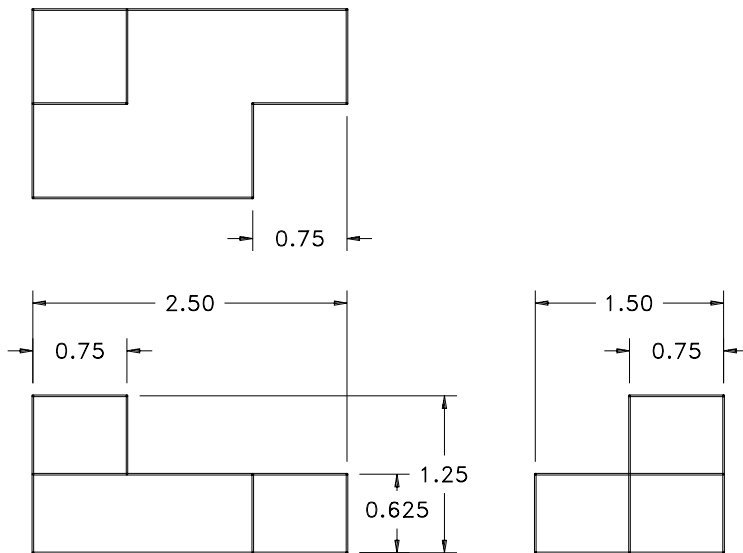
PROBLEM A-12**PROBLEM A-13**

PROBLEM A-14



PROBLEM A-15



PROBLEM A-16**PROBLEM A-17**