Assignment 12

CAD Mechanical – Part 2

Hidden Lines & Isometric Functions

Objectives

In this assignment you will learn to apply the **hidden lines, isometric snap, and ellipses** commands along with commands previously learned..

General

1. When AutoCAD's menu appears, scroll down and select the **Otto 2016.dwt** template file.

2. From the **popup dialog box** type the information for the title block. You will name the nineteenth drawing assignment the same as previously drawn assignments.

   Anvil   C19
3. Remember to set the **Scale** to **½" = 1".**
4. Set the dimscale factor to 2 and set the ltscale to .70. Refer to Assignment 11 steps 1-3 for assistance if needed.
5. Create **drawing C19** using the instruction manual.
6. **Dimension** the **drawing** and **save** the **file** into the **correct location** and **file name**.

Multiview Drawings

1. The Anvil drawing is a multiview drawing. Multiview drawings are twodimensional representations of a three-dimensional object. The object can best be fully described with three views. The most commonly drawn views are the front, top, and right side views.
2. The first view that you will draw is the front view. This is the drawing in the lower left corner of the title block.
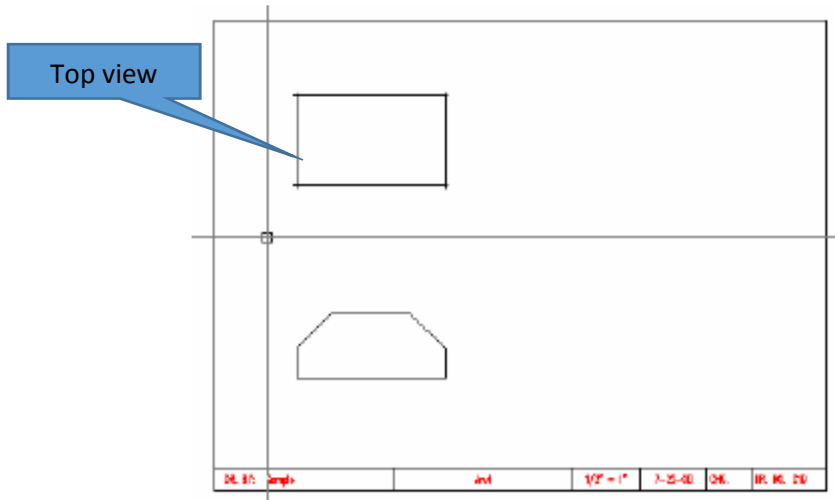
3. Draw the object lines in the object layer as illustrated below:
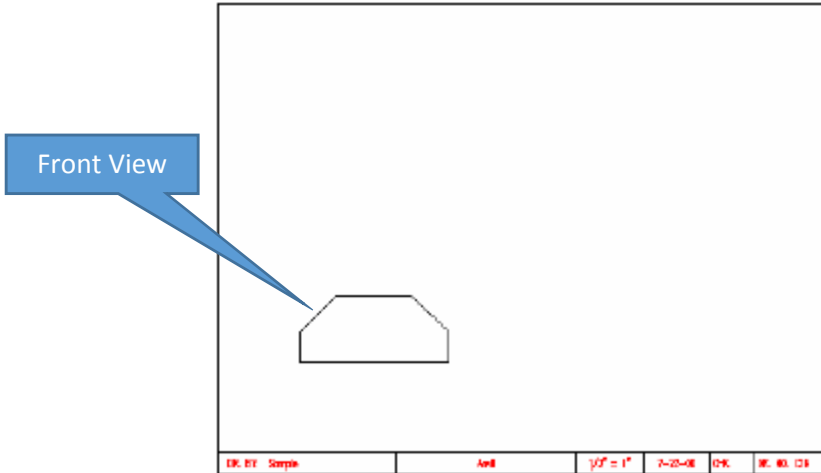
Front View

4. Move the cross hair directly above the front view. The vertical line on the crosshair should align with the left side of the front view.

Align vertical cross hair here.
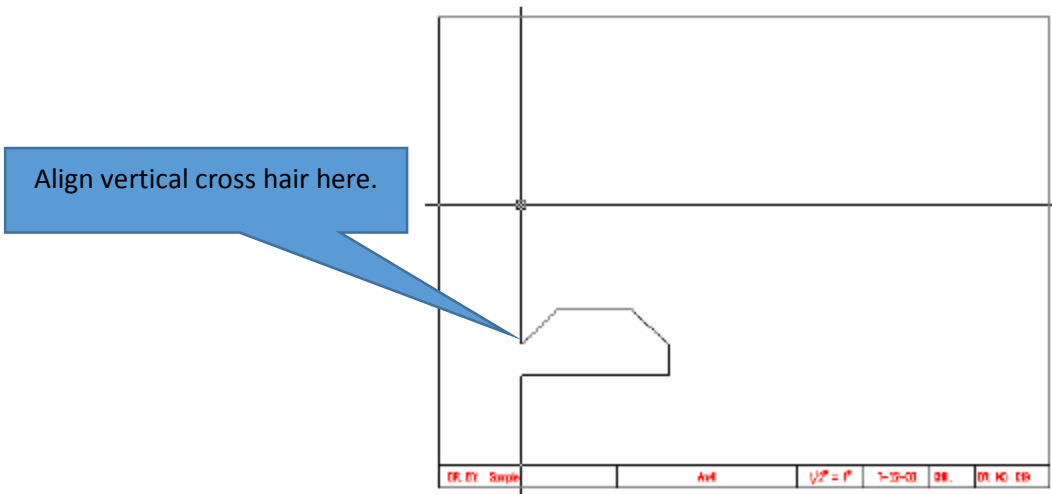
5. When the alignment is correct, draw the perimeter of the top view.

Top view

6. Position the cross hair to align with the bottom of the front view and to the right as illustrated:
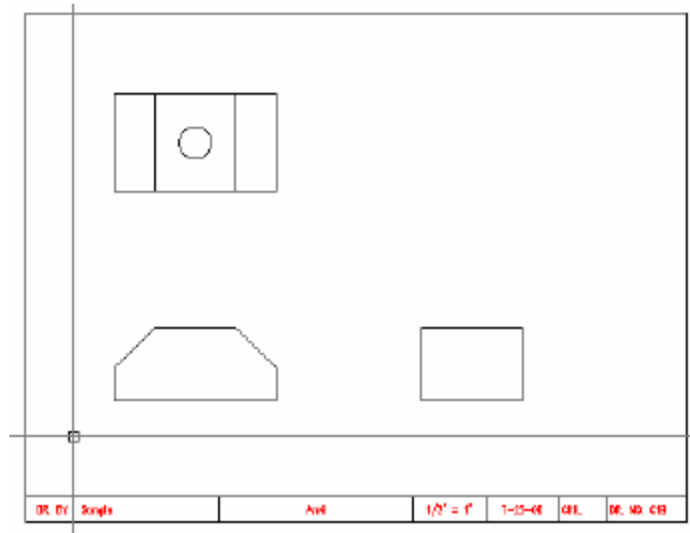

Align horizontal cross hair here.
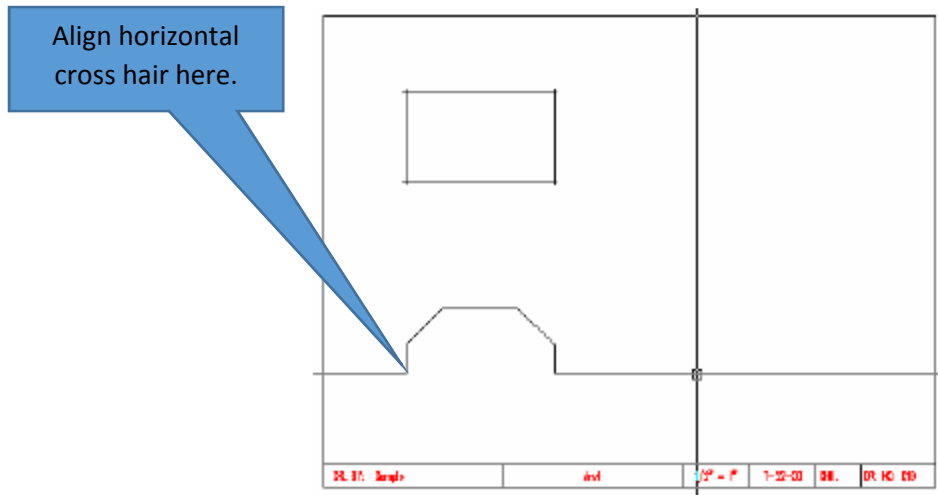
7. Draw the perimeter of the right side view here. This method of aligning views across and above each other is called orthographic projection.
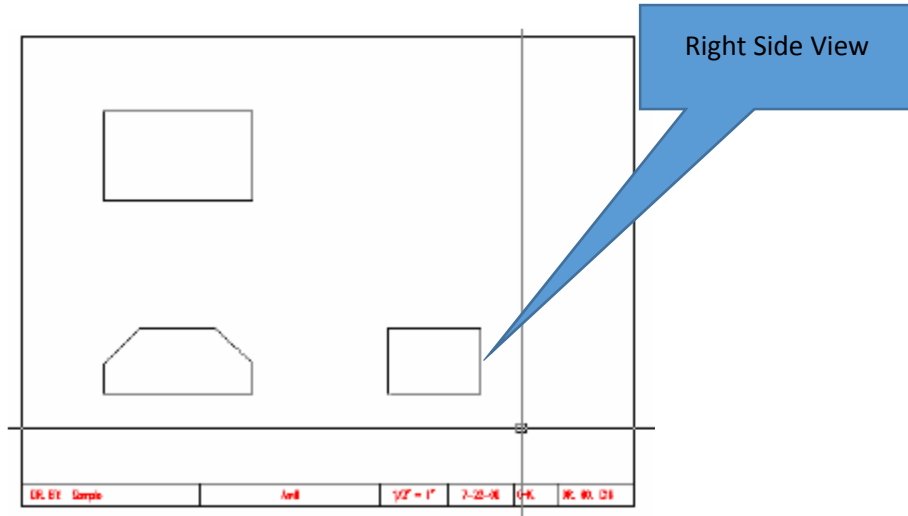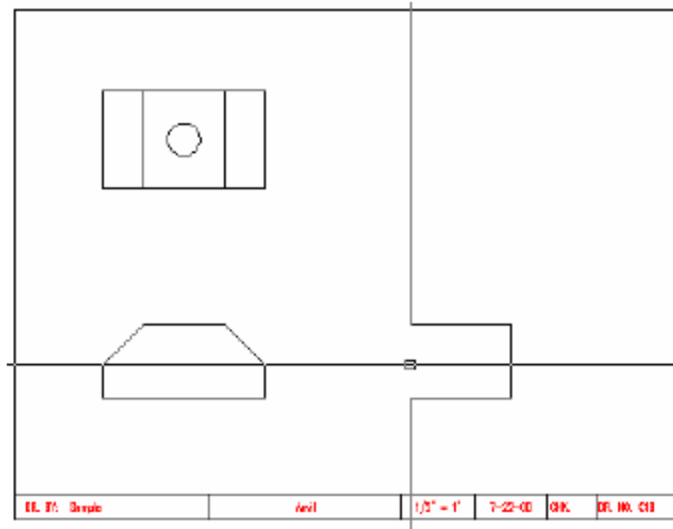
Right Side View

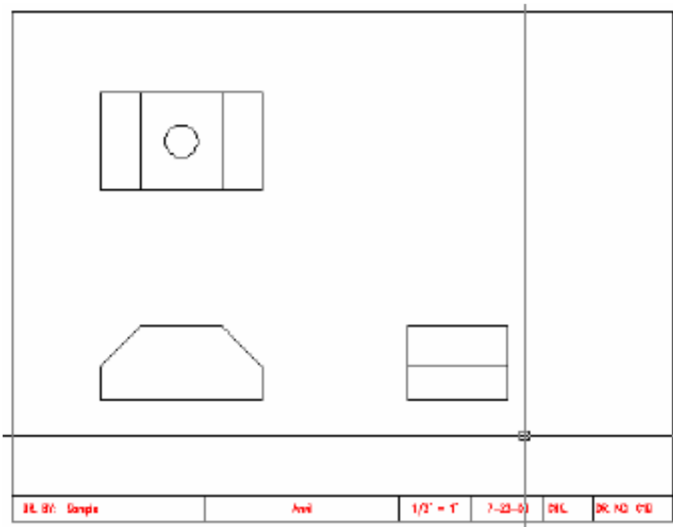# *Drawing the Inside Object Lines*

1. Draw the vertical lines onto the top view and the 1" diameter hole.

2. Align the cross hair with the angle on the front view and the vertical line on the left of the right side view.

3. Click the start point of the line and draw the horizontal line on the right side view.
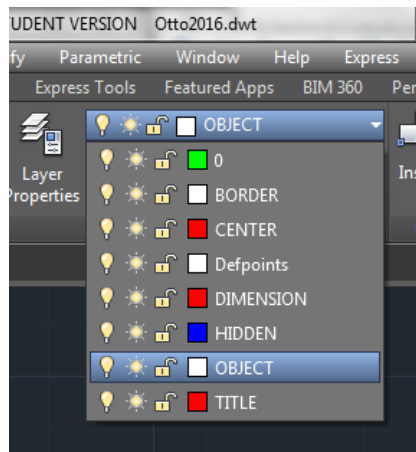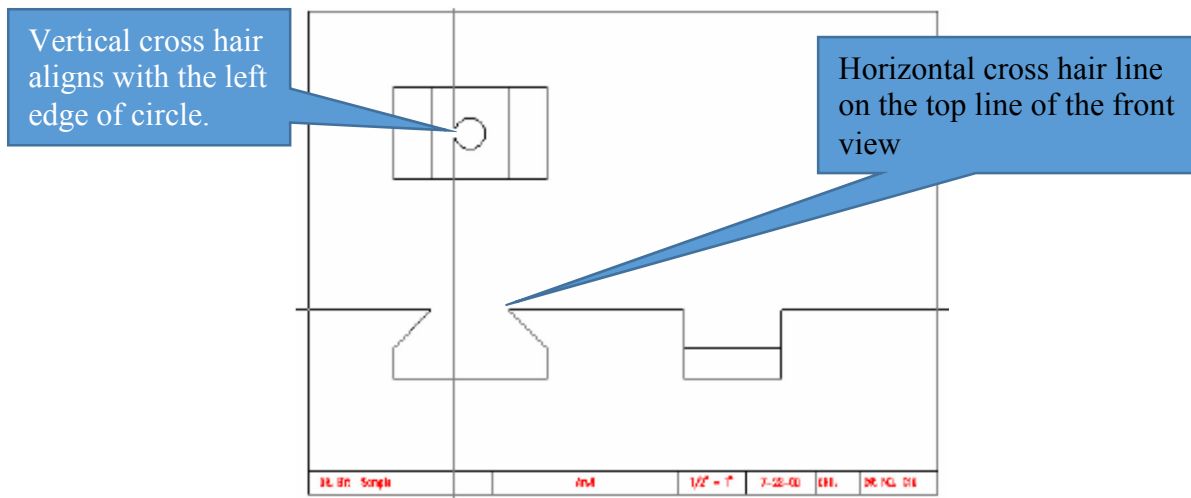
Drawing the Hidden Lines

1. The 1" diameter hole that you drew onto the top view must also be represented on the front and the right side views. You cannot use the object lines because they are solid lines and would be confusing. Hidden lines are drawn as a series of dashed lines. They are used to represent interior lines on an object that you cannot see, because the shape of the object blocks their full view. The color that we will use for hidden lines is blue.
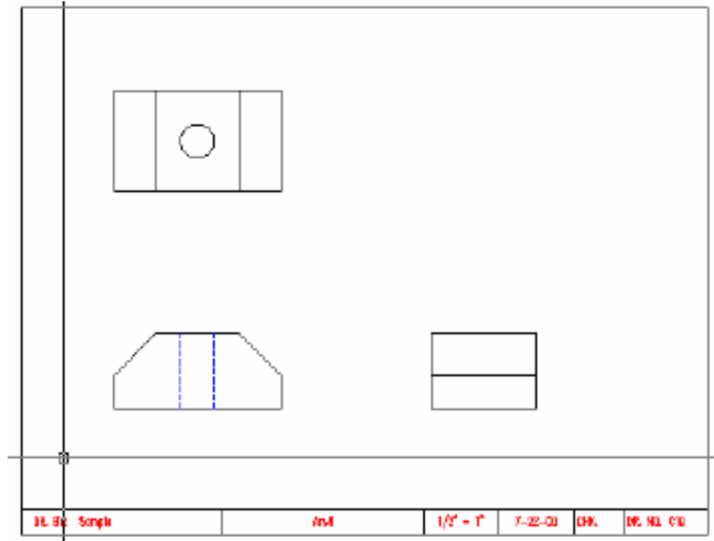

Hidden Line Sample

2. Select the Hidden line layer from the Home Ribbon menu toolbar.

3. Select the Line command and align the cross hair with the left edge of the circle on the top view and the top line of the front view. When positioned correctly, click that point for the first point of the line to be drawn.

Vertical cross hair aligns with the left edge of circle.

Horizontal cross hair line on the top line of the front view

4. Draw the hidden line onto the front view when alignment is attained. Draw the other hidden line by aligning with the right edge of the circle.

5. The hidden lines for the right side view can be drawn using the distance command to locate the location. Draw the two vertical hidden lines on the right side view.
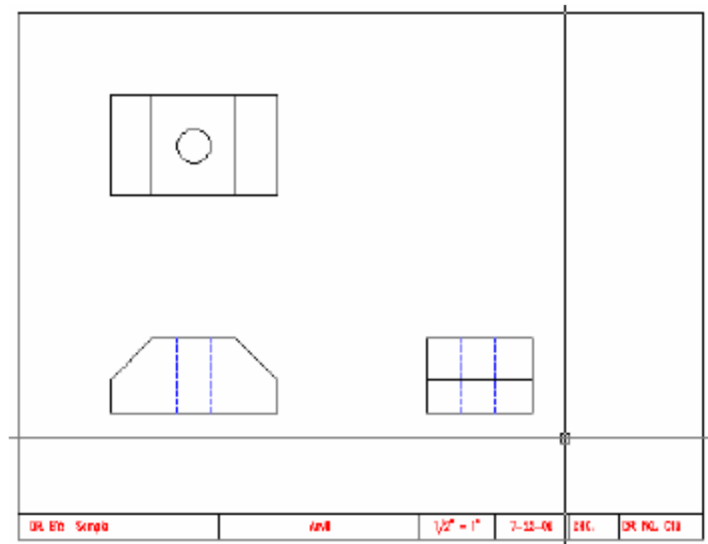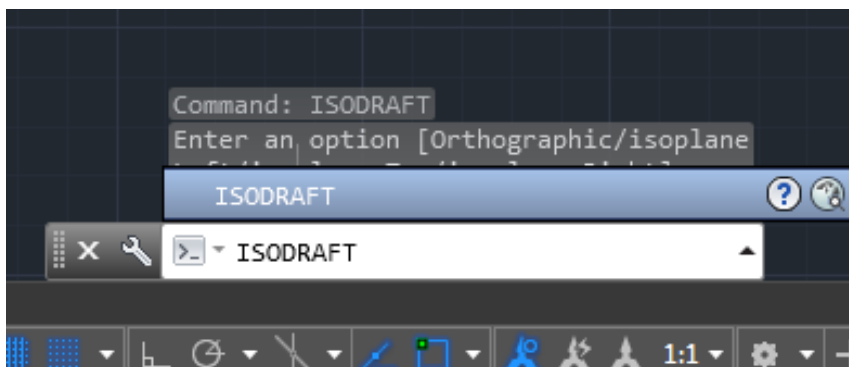
6. Complete the drawing to the three views by adding the center lines and th dimensions onto the views.
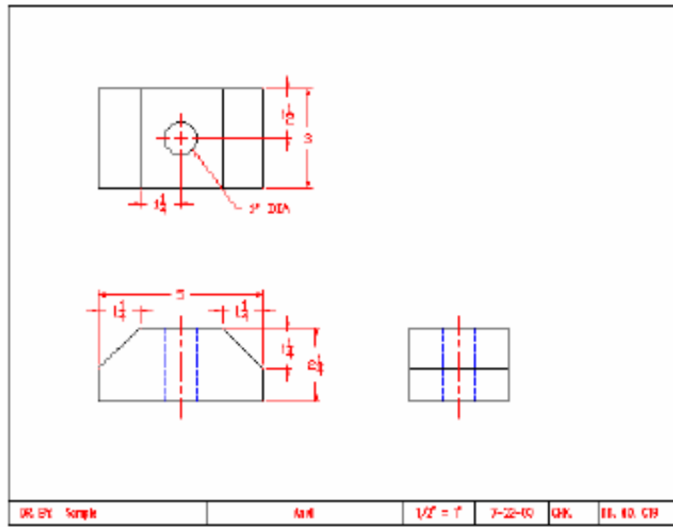

Drawing the Isometric View

1. On the sample drawing you will notice a drawing that is a three dimensional representation of the anvil. The axes for the drawing are 120 degrees apart on an isometric drawing. Type isodraft
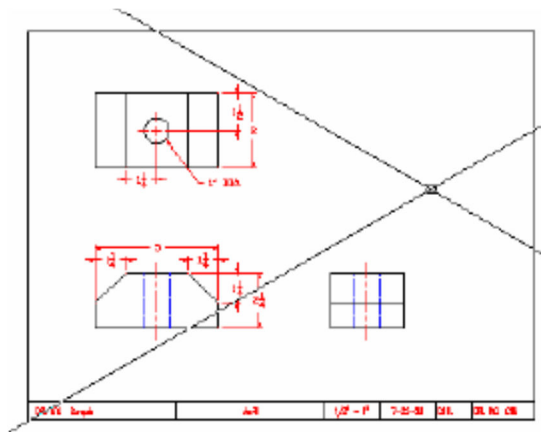
2. When the drawing reappears the cross hair is no longer positioned in a vertical / horizontal plane. The horizontal axis is on a 30 degree angle as represented below:
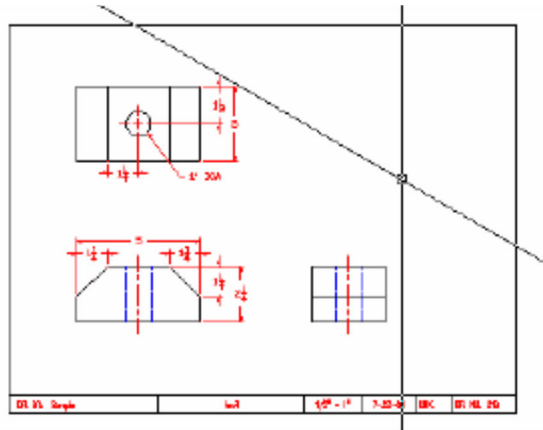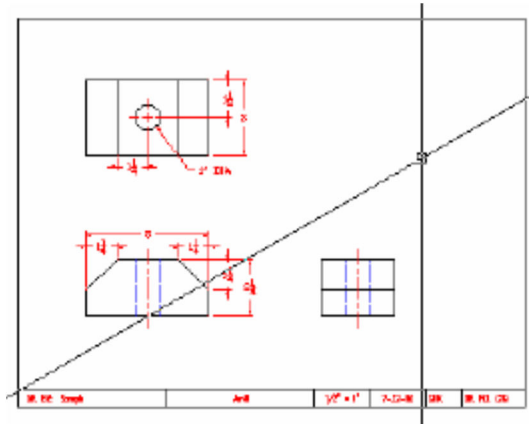

Note: The current cross hair setting is the Isoplane Left Setting.

3. Press the <CTRL> key and while pressing the <CTRL> press the <E> key. You will see the cross hair switched to Isoplane Top Setting.

4. Press <CTRL> <E> again and you will see the cross hair switch to the Isoplane Right Setting.

5. Move the cross hair over the drawing and observe the readings on the Status Line. If the units are reading in 1/32 or 1/64, you will need to set the snap.


Status line reading is /64.

The snap will require resetting.
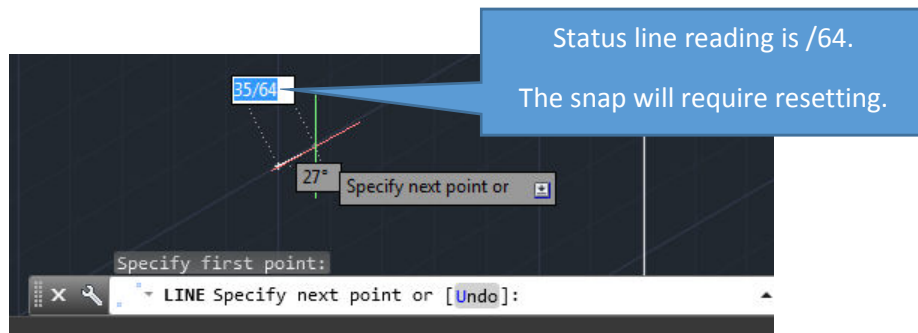
6. Change the Snap to ¼ or .25".

7. Be sure that you are in the object layer. Select the line command and press <CTRL> <E> to return the cross hair to Isoplane Left. Turn ortho on and draw the front side of the isometric drawing as illustrated:


Use Isoplane left, snap ¼", and ortho on.
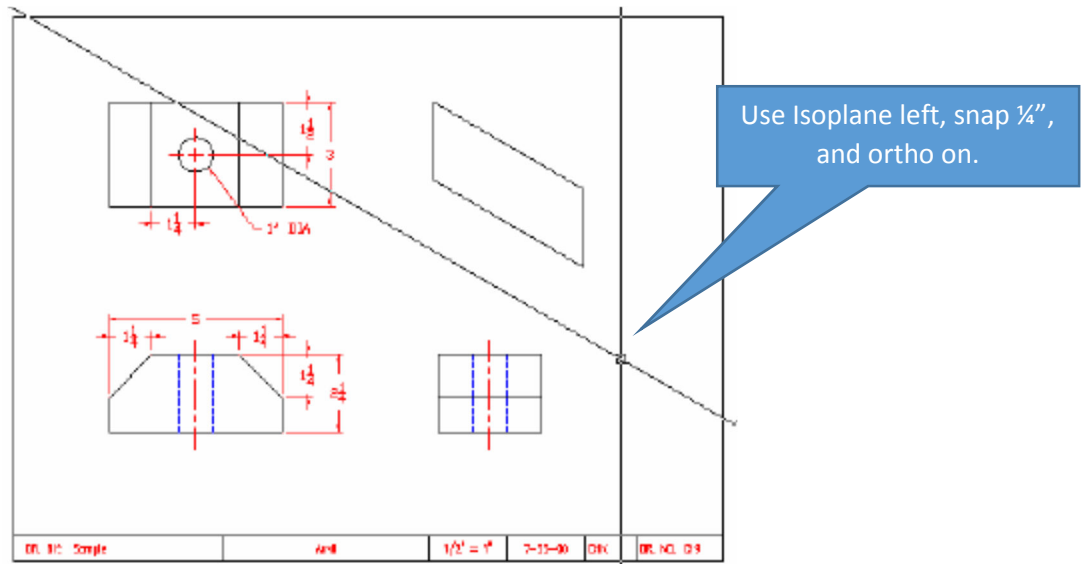
8. Use the chamfer command to construct the angle on the corners.
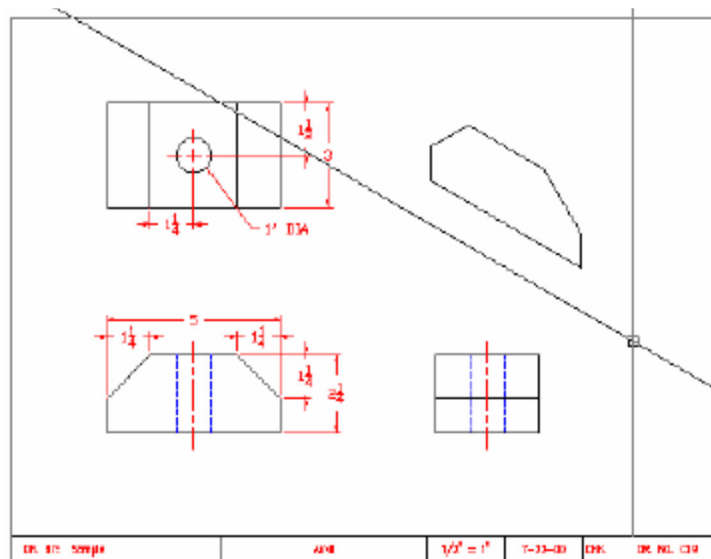
9. Press <CTRL> <E> to change the cross hair to Isoplane Top. Choose the copy command and select the three lines in the illustration below. Pick the lower right corner of the drawing for the base point and move the cross hair 3" to the right.
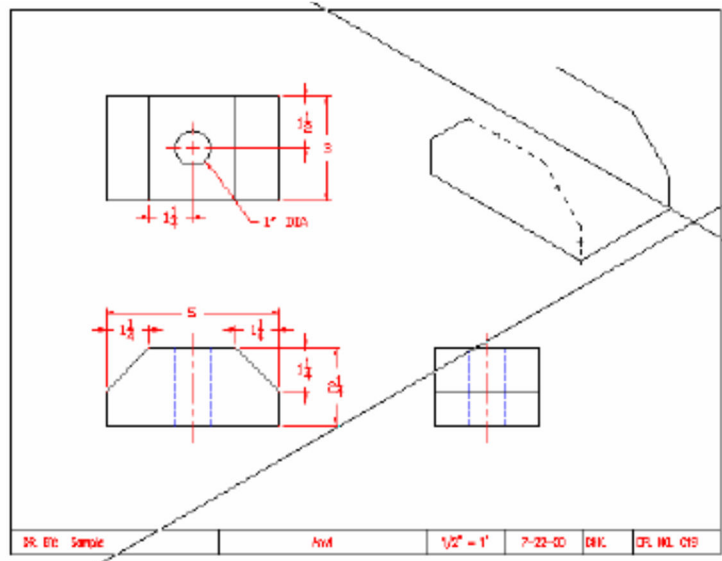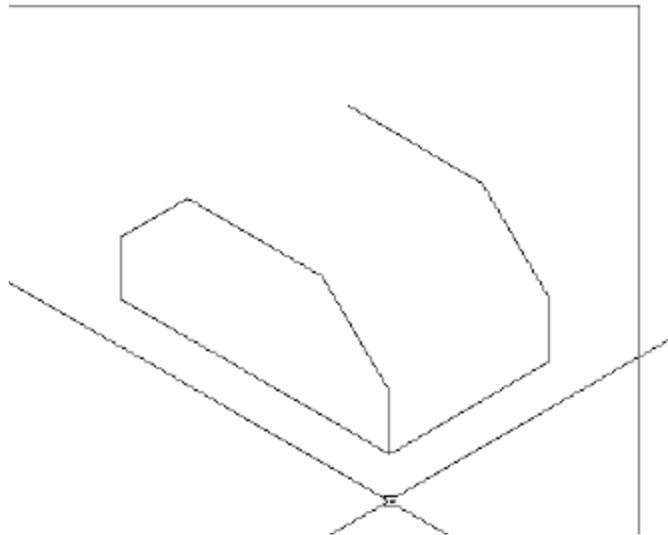

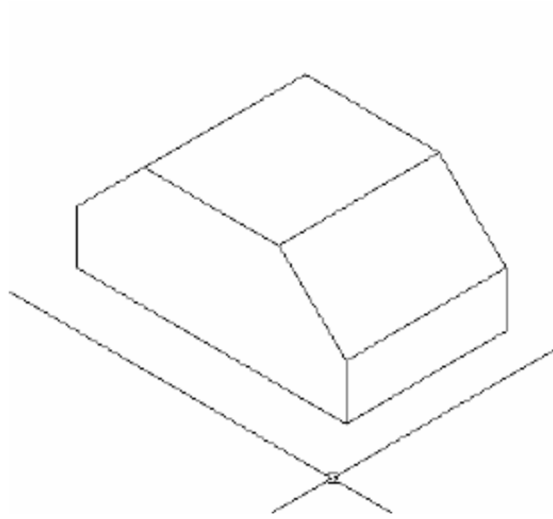10. Pick the point at 3" and the copied lines will appear as shown.
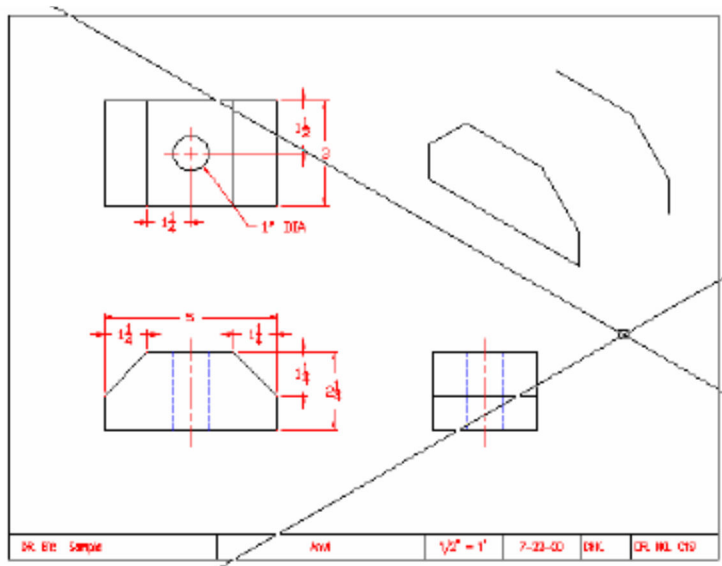
11. Select the Zoom Window and drag a window around the Isometric drawing.
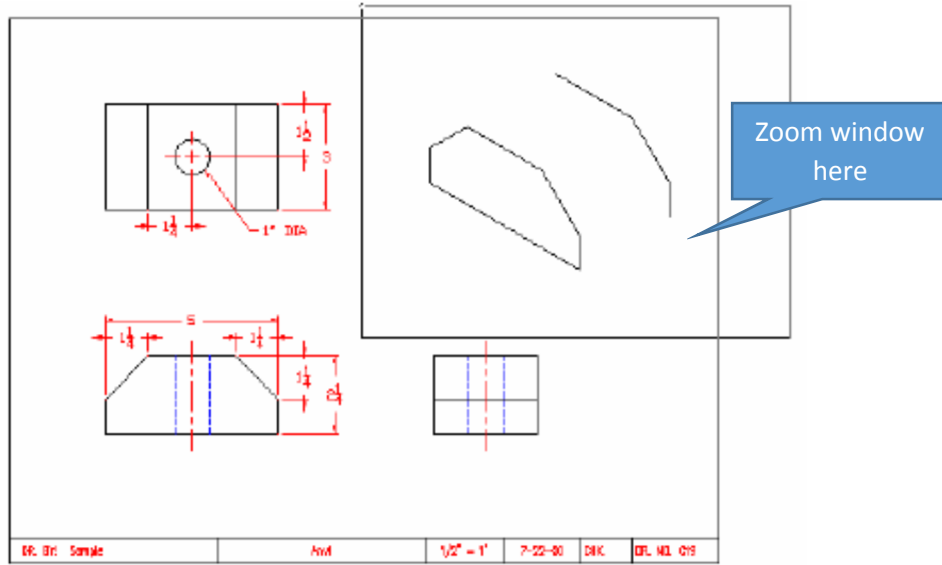
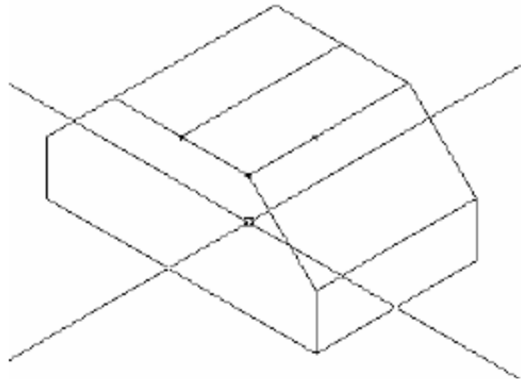12. Draw the base line that connects the view the drawing together.

13. Continue drawing the other three lines that connect the isometric drawing together.


Drawing the Isometric Circle

1. Next copy one of the top lines and move the copy 1 ¼" in the middle.

2. You are now going to draw the hole onto the top of the anvil. The line that you copied in step 1 will assist you in locating the center point of the hole. The drawing is drawn on an Isometric plane. You will not use the ellipse command to draw the hole. Select Ellipse from the Draw menu and choose Axis, End. Enter I (Isocircle) for isometric circle and press Enter.

3. The command prompt alerts you to select the center point for the circle. Use the midpoint tool to locate the midpoint of the middle line on the top view. Pick the mouse on the center point.

4. Drag the mouse and observe the size of the isocircle. If the ellipse appears to be shaped wrong, press <CTRL> <E> until the Isoplane top appears. Drag the mouse until the status line reads ½" pick the button. The Status line reads in radius settings. The 1"diameter isometric circle should be drawn.
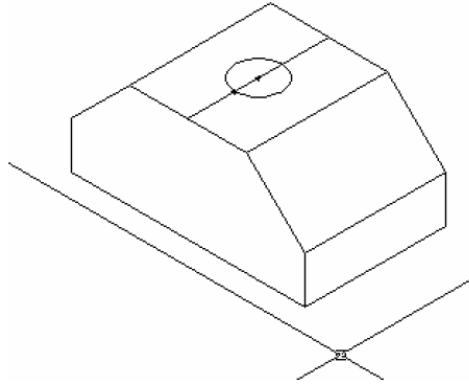
5. Erase the line that crosses the middle of the top view. Type z (zoom) and press the Enter key and type a (all) and press the Enter key.
6. Check the drawing to see if all of the dimensions and views are correct. If you have errors or need to position the drawing better into the title block, you will need to restore the original setting. Select the Tools menu and Drafting Settings. Click the Rectangular Snap bull'eye and click OK.
7. The final drawing should appear as illustrated.
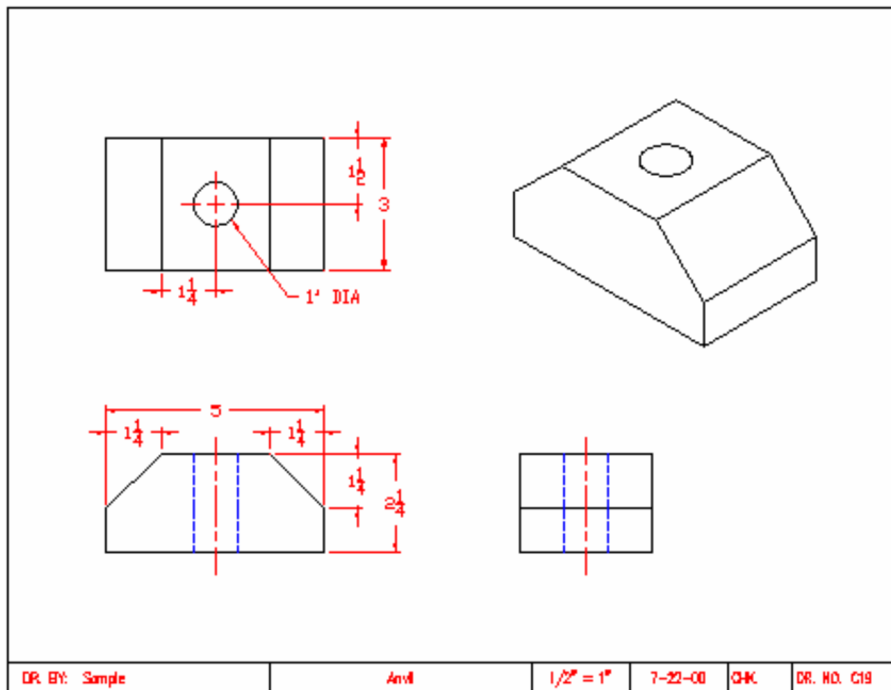
8. Save the drawing and if time permits begin on the second drawing in this activity.

General Instructions

1. When AutoCAD's menu appears, scroll down and select the **Otto 2016.dwt** template file.

2. You should notice that there is no Title Block. You will learn how to insert the **8X11** file and set it up.

3. From the **popup dialog box** type the information for the title block. You will name the nineteenth drawing assignment the same as previously drawn assignments.

    Guide C20
4. Remember to set the **Scale** to **½" = 1".**
5. Set the dimscale factor to 2 and set the ltscale to .70. Refer to Assignment 11 steps 1-3 for assistance if needed.
6. Create **drawing C20** using the instruction manual.
7. **Dimension** the **drawing** and **save** the **file** into the **correct location** and **file name**.

Application

1. Draw the three orthographic views of C20 to start the drawing and complete all of the dimensions. If the text or the line scaling is wrong, check the ltscale or the dimscale.
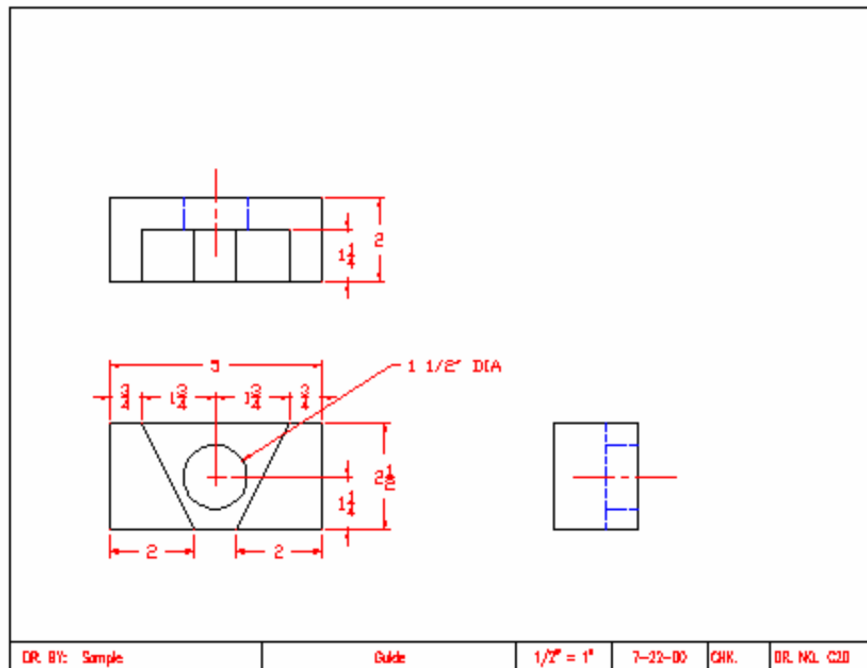

2. Select the isometric drawing settings (Tools menu, Drafting Settings, and Isometric Snap).

3. Set the snap to ¼" and turn ortho on make the object layer active.
4. Draw the top of the isometric view while using the Isoplane Top as illustrated. Use the dimensions from the multiview drawing to locate the points for the isometric view.
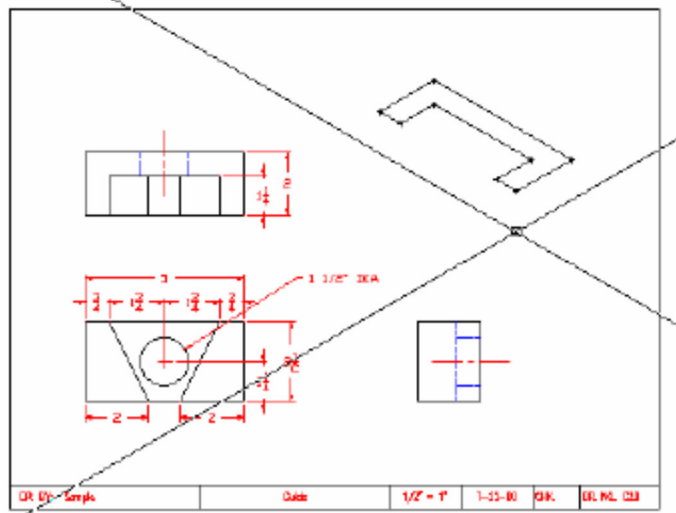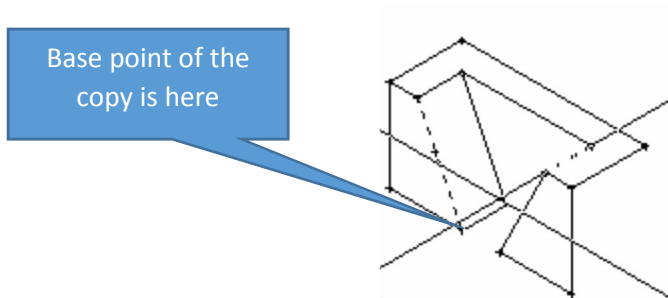

5. Use the Isoplane Left and draw the lines of the front as shown.
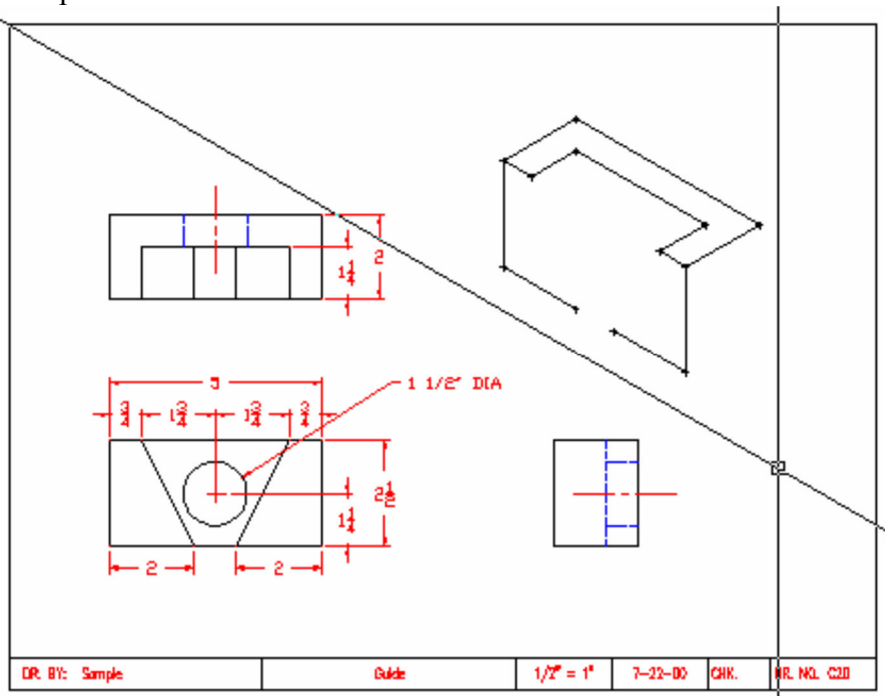
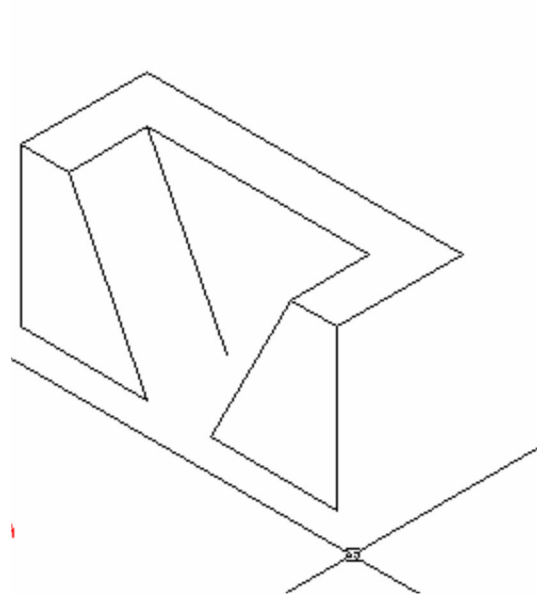6. Turn ortho off and draw the two angular lines on the front.
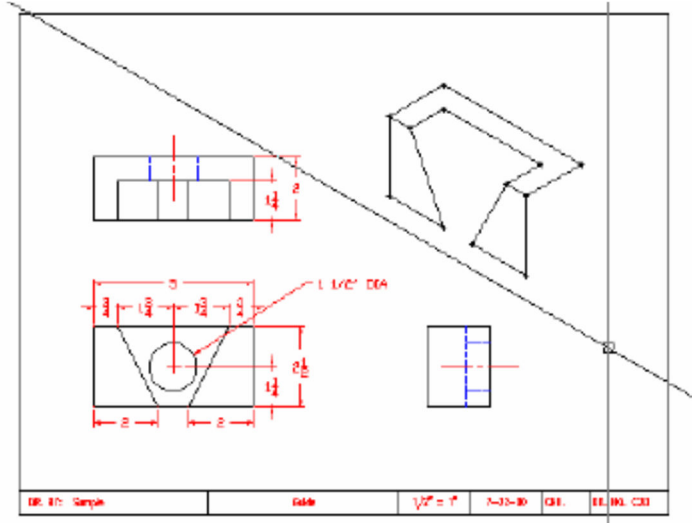
7. Turn ortho on and copy the angle line from the front toward the back. Position the copy 1 ¼" in from the original.
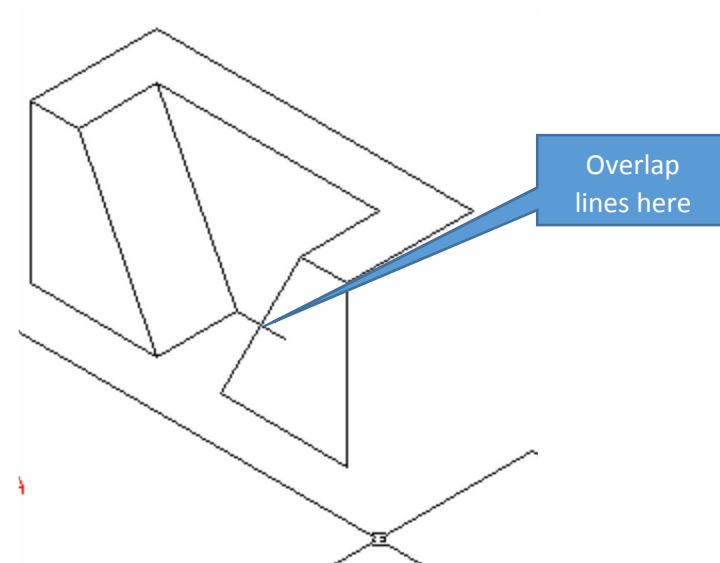

Note: Select the line, change to Isoplane top, and move inside the drawing 1 ¼"
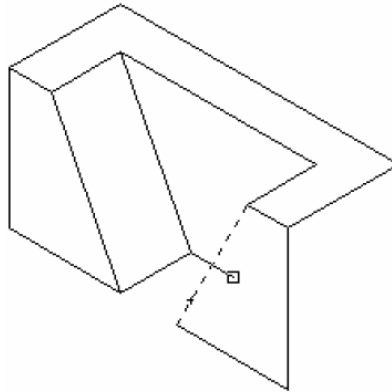
8. The drawing should appear as illustrated.

9. Draw the next two lines using the Isoplane Top as illustrated. Draw the inside line slightly past the angled line on the right.

10. Trim the over lapping lines. Select the angle line for the cutting edge and place the pick box the excess side of the line to trim.


11. Copy Line A to the Line B position. Change to Isoplane Right to position the copy down 1 1¼" from Line A.
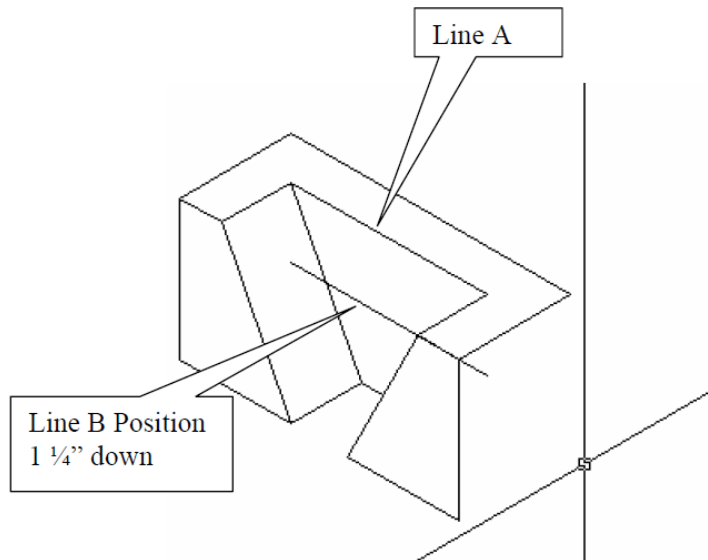
12. From the Draw pull down menu select the Ellipse command. Choose the Axis, End. Click the Command window and type i (isocircle). Choose the midpoint snap and position the cross hair onto Line B to locate the center for the isometric circle.
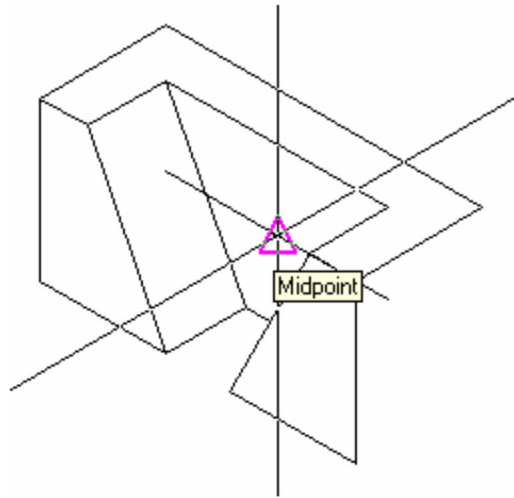
13. Pick the point and drag the mouse to the correct diameter of the circle. If the Status line reads in radius, drag the window until it reads ¾" for the radius.
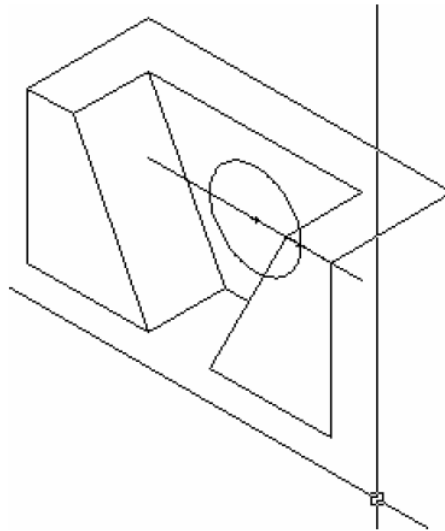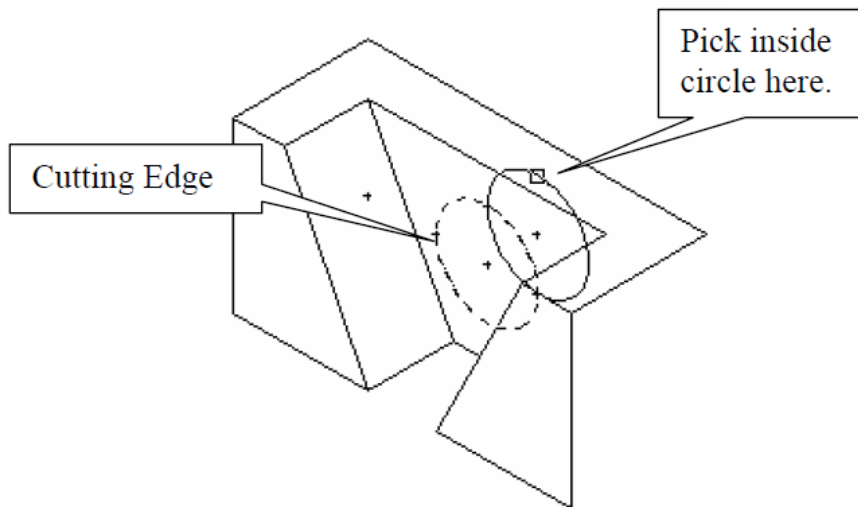
14. Copy the circle and drag the copy back ¾" in from the original.
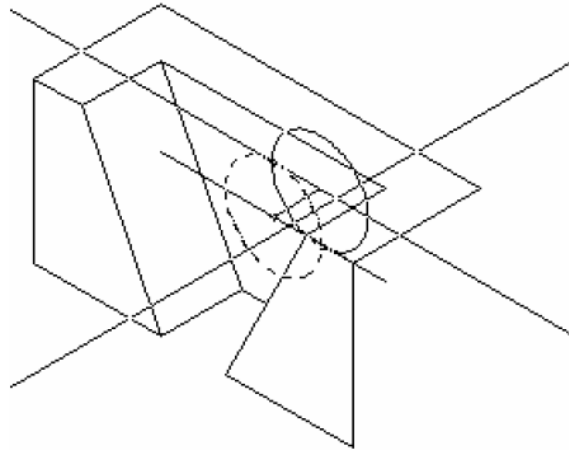
15. Erase the Line B (the line used to locate the center point of the isometric circle.
16. Select the Trim command, pick the front isometric circle for the cutting edge, and press Enter. Pick the top of the inside isometric circle to trim.

17. Continue the Trim command by selecting the next two lines for cutting edges.
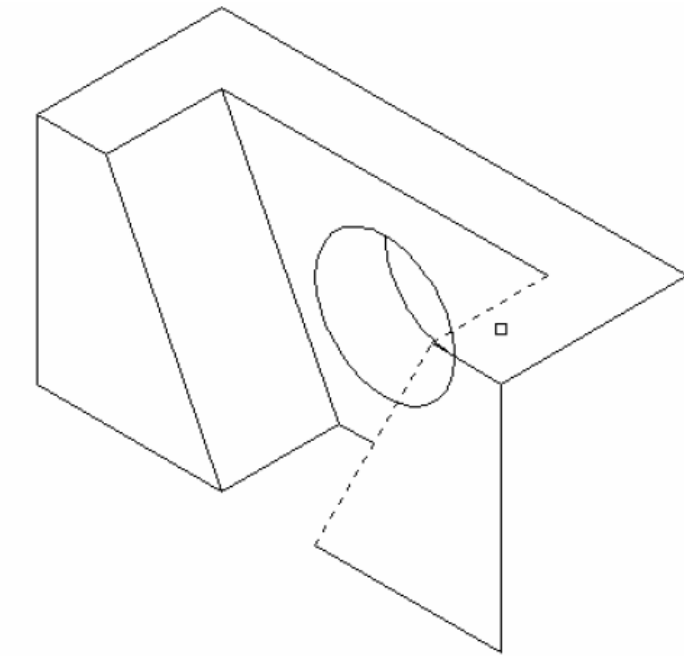

18. Press the Enter key after selecting the two lines. Pick the part of the front circle that over laps the cutting edge with the pick box. The line circle should be trimmed.
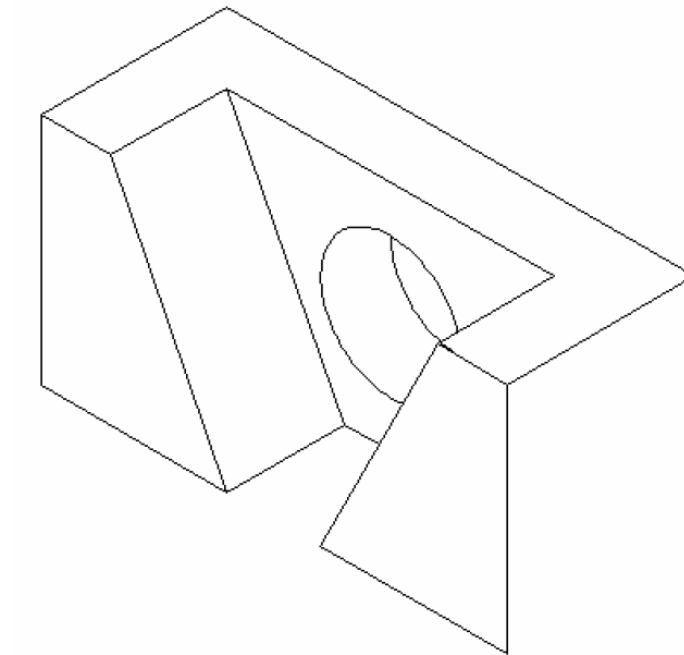

19. Draw the final two missing lines onto the right side of the drawing.
20. Zoom out of the window using the zoom all feature.

21. Change the drawing back to Rectangular Snap and check the entire drawing for missing lines or dimensions.
22. The completed drawing should appear as illustrated.
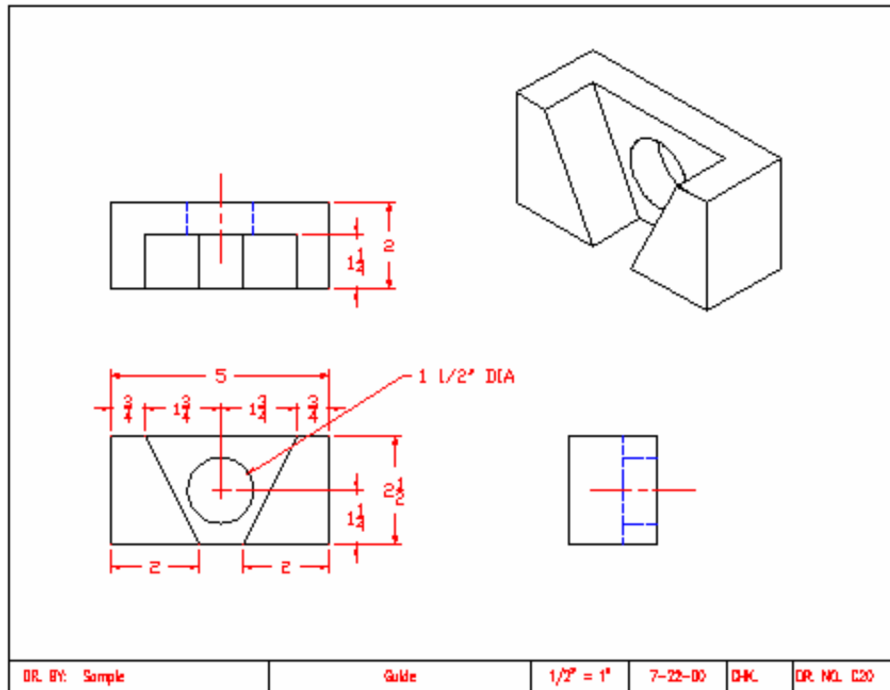

Terms to Know

Front View - From multiview drawing, one of the three most commonly drawn views to represent a 3-D object on a two dimensional page. These views are normally arranged as shown at the right.
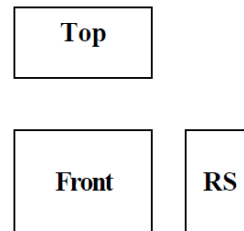

Hidden line - Represents a surface that cannot be seen in a view because it is hidden by other features.

Iso ellipses - To draw an isometric circle, use the Isocircle option of the ELLIPSE command. The Isocircle option is available for ELLIPSE only when the Style option of Snap mode is set to Isometric (see DSETTINGS).

Isometric drawing - simulates a 3D object from a particular viewpoint by aligning along three major axes, as shown on the right.
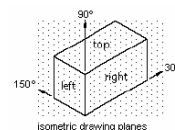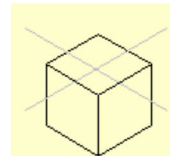
Isometric snap - A drafting option that aligns the cursor with two of three isometric axes and displays grid points, making isometric drawings easier to create.
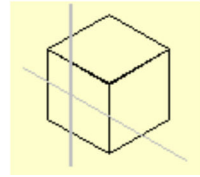
Isoplane left - Selects the left-hand plane, defined by the 90-degree and 150-degree axis pair.
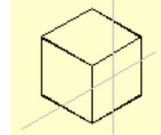
Isoplane right - Selects the right-hand plane, defined by the 90- degree and 30-degree axis pair.

Isoplane top - Selects the top face of the cube, called the top plane, defined by the 30-degree and 150-degree axis pair.

Multiview Drawing - A two-dimensional (2-D) representation of a three-dimensional (3-D) object; the object can be described fully through the use of more than one view.

Orthographic projection - An object is seen as a series of several single views that show each face of the object in its true size and shape.

Right Side View - From multiview drawing, one of the three most commonly drawn views to represent a 3-D object on a two dimensional page. These views are normally arranged as shown above.

Top View - From multiview drawing, one of the three most commonly drawn views to represent a 3-D object on a two dimensional page. These views are normally arranged as shown above.