Creating an Isometric Drawing

In this exercise, the object in Figure 25-14 is drawn in isometric.

The initial steps to create an isometric drawing begin with the typical setup (see Chapter 6, Drawing Setup):

1. Set the desired Units.
2. Set appropriate Limits.
3. Set the Isometric Style of Snap and specify an appropriate value for spacing.

4. The next step involves creating an isometric framework of the desired object. In other words, draw an isometric box equal to the overall dimensions of the object. Using the dimensions given in Figure 25-14, create the encompassing isometric box with the Line command (Fig. 25-15).

Use ORTHO to force isometric Lines. Watch the Coords display (in a relative polar format) to give the current lengths as you draw or use direct distance entry.

5. Add the lines defining the lower surface. Define the needed edge of the upper isometric surface as shown.
The inclined edges of the inclined surface can be drawn (with Line) only when ORTHO is OFF. Inclined lines in isometric cannot be drawn by transferring the lengths of the lines, but only by defining the ends of the inclined lines on isometric lines, then connecting the endpoints. Next, Trim or Erase the necessary Lines.

7. Draw the slot by constructing an Ellipse with the Isocircle option. Draw the two Lines connecting the circle to the right edge. Trim the unwanted part of the Ellipse (highlighted) using the Lines as cutting edges.

8. Copy the far Line and the Ellipse down to the bottom surface. Add two vertical Lines at the end of the slot.

9. Use Trim to remove the part of the Ellipse that would normally be hidden from view. Trim the Lines along the right edge at the opening of the slot.
10. Add the two holes on the top with *Ellipse, Isocircle* option. Use *ORTHO ON* when defining the radius. *COPY* can also be used to create the second *Ellipse* from the first.

**Dimensioning Isometric Drawings in AutoCAD**

Refer to Chapter 28, Dimensioning, for details on how to dimension isometric drawings.

**OBLIQUE DRAWING IN AutoCAD**

Oblique drawings are characterized by having two axes at a 90 degree orientation. Typically, you should locate the front face of the object along these two axes. Since the object’s characteristic shape is seen in the front view, an oblique drawing allows you to create all shapes parallel to the front face true size and shape as you would in a multi-view drawing. Circles on or parallel to the front face can be drawn as circles. The third axis, the receding axis, can be drawn at a choice of angles, 30, 45, or 60 degrees, depending on whether you want to show more of the top or the side of the object.

Figure 25-22 illustrates the axes orientation of an oblique drawing, including the choice of angles for the receding axis.

Another option allowed with oblique drawings is the measurement used for the receding axis. Using the full depth of the object along the receding axis is called *Cavalier* oblique drawing. This method depicts the object (a cube with a hole in this case) as having an elongated depth (Fig. 25-23).

Using 1/2 or 3/4 of the true depth along the receding axis gives a more realistic pictorial representation of the object. This is called a *Cabinet* oblique (Fig. 25-24).

No functions or commands in AutoCAD are intended specifically for oblique drawing. However, *Polar Snap* and *Polar Tracking* can simplify the process of drawing lines on the front face of the object and along the receding axis. The steps for creating a typical oblique drawing are given next.