Figure 25-7 illustrates the 2D nature of an isometric drawing created in AutoCAD. Isometric lines are created on the XY plane. The Isometric SNAP and GRID are also on the 2D plane. (The Vpoint command was used to give other than a Plan view of the drawing in this figure.)

Although pictorial drawings are based on the theory of projecting 3D objects onto 2D planes, it is physically possible to achieve an axonometric (isometric, dimetric, or trimetric) viewpoint of a 3D object using a 3D CAD system. In AutoCAD, the Vpoint command is used to specify the observer’s position in 3D space with respect to a 3D model. Chapter 35 discusses the specific commands and values needed to attain axonometric viewpoints of a 3D model.

**ISOMETRIC DRAWING IN AutoCAD**

AutoCAD provides the capability to construct isometric drawings. An isometric SNAP and GRID are available, as well as a utility for creation of isometrically correct ellipses. Isometric lines are created with the Line command. There are no special options of Line for isometric drawing, but isometric SNAP and GRID can be used to force Lines to an isometric orientation. Begin creating an isometric drawing in AutoCAD by activating the Isometric Style option of the Snap command. This action can be done using any of the options listed in the following command table.

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND (TYPE)</th>
<th>ALIAS (TYPE)</th>
<th>Shortcut</th>
<th>Screen (side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tools</td>
<td>SNAP</td>
<td>SN</td>
<td>F9 or Ctrl+B</td>
<td>TOOLS 2 Grid</td>
<td>W,10</td>
</tr>
<tr>
<td>Drafting Settings...</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Snap and Grid</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Figure 25-8**

Alternately, toggling the indicated checkbox in the lower-right corner of the Drafting Settings dialog box activates the Isometric Snap and Grid Snap (Fig. 25-8).
Figure 25-9 illustrates the effect of setting the Isometric SNAP and GRID. Notice the new position of the cursor.

![Figure 25-9](image)

Using Ctrl+E (pressing the Ctrl key and the letter “E” simultaneously) toggles the cursor to one of three possible Isoplanes (AutoCAD’s term for the three faces of the isometric pictorial). If ORTHO is ON, only isometric lines are drawn; that is, you can only draw Lines aligned with the isometric axes. Lines can be drawn on only two axes for each isoplane. Ctrl+E allows drawing on the two axes aligned with another face of the object. ORTHO is OFF in order to draw inclined or oblique lines (not on the isometric axis). The functions of GRID (F7) and SNAP (F9) remain unchanged.

With SNAP ON, toggle Coords (F6) several times and examine the read-out as you move the cursor. The absolute coordinate format is of no particular assistance while drawing in isometric because of the configuration of the GRID. The relative polar format, however, is very helpful. Use relative polar format for Coords while drawing in isometric (Fig. 25-10).

The effects of changing the Isoplane are shown in the following figures. Press Ctrl+E to change Isoplane.

![Figure 25-10](image)

With ORTHO ON, drawing a Line is limited to the two axes of the current Isoplane. Only one side of a cube, for example, can be drawn on the current Isoplane. Watch Coords (in a polar format) to give the length of the current Line as you draw (lower left corner of the screen).
Toggling Ctrl+E switches the cursor and the effect of ORTHO to another Isoplane. One other side of a cube can be constructed on this Isoplane (Fig. 25-11).

Direct Distance Entry can be of great help when drawing isometric lines. Use Ctrl+E and ORTHO to force the Line to the correct orientation, then enter the desired distance value at the Command line.

**Isometric Ellipses**

Isometric ellipses are easily drawn in AutoCAD by using the *Isocircle* option of the *Ellipse* command. This option appears only when the isometric SNAP is ON.

**ELLIPSE**

<table>
<thead>
<tr>
<th>Pull-down Menu</th>
<th>COMMAND (TYPE)</th>
<th>ALIAS (TYPE)</th>
<th>Shortcut</th>
<th>Screen(side) Menu</th>
<th>Tablet Menu</th>
</tr>
</thead>
<tbody>
<tr>
<td>Draw Ellipse</td>
<td>ELLIPSE</td>
<td>EL</td>
<td>...</td>
<td>DRAW 1</td>
<td>M,9</td>
</tr>
</tbody>
</table>

Although the *Isocircle* option does not appear in the pull-down or digitizing tablet menus, it can be invoked as an option of the *Ellipse* command. You **must** type "I" to use the *Isocircle* option. The command syntax is as follows:

Command: *ellipse*
Specify axis endpoint of ellipse or [Arc/Center/Isocircle]: i
Specify center of isocircle: PICK or (coordinates)
Specify radius of isocircle or [Diameter]: PICK or (coordinates)

Command:

After selecting the center point of the *Isocircle*, the isometrically correct ellipse appears on the screen on the current Isoplane. Use Ctrl+E to toggle the ellipse to the correct orientation. When defining the radius interactively, use ORTHO to force the rubberband line to an isometric axis (Fig. 25-12, next page).
ince isometric angles are equal, all isometric ellipses have the same proportion (major to minor axis). The only differences in isometric ellipses are the size and the orientation (Isoplane).

Figure 25-12 shows three ellipses correctly oriented on their respective faces. Use Ctrl+E to toggle the correct Isoplane orientation: Isoplane Top, Isoplane Left, or Isoplane Right.

Figure 25-13

When defining the radius or diameter of an ellipse, it should always be measured in an isometric direction. In other words, an isometric ellipse is always measured on the two isometric axes (or center lines) parallel with the plane of the ellipse.

If you define the radius or diameter interactively, use ORTHO ON. If you enter a value, AutoCAD automatically applies the value to the correct isometric axes.