Drawing Aids

by David Watson

Introduction

Drawing with AutoCAD is really just like drawing on a drawing board. Most newcommers to Computer Aided Design assume that they will need to learn how to draw all over again. In fact, many of the drawing aids that AutoCAD provides are analagous to traditional drafting tools. Just as you have a parallel motion and set squares to help you draw horizontal and vertical lines on a drawing board, AutoCAD has similar drawing aids which can help you to draw horizontal and vertical lines on a computer. This means that in many respects, the drawing techniques are very similar. If you ever get stuck, think how you would complete a task on a drawing board and then look for a similar way to do it with AutoCAD.

Ortho Mode

Status Bar   ORTHO

Pull-down   None

Keyboard   **ORTHO** or **F8**

Ortho is short for *orthogonal*, which means either vertical or horizontal. Like the other options on the status bar, Ortho is not really a command, it is a drawing *mode* which can either be turned on or off. Ortho mode can be *toggled*
on or off in one of three ways. The quickest way is just to click on the ORTHO button on the status bar. The appearance of the button tells you whether Ortho is currently turned on or turned off. When Ortho is turned on, the ORTHO button appears pressed in. You can see how this appears by looking at the status bar illustration below. In the illustration, Ortho is turned on but Grid and Snap are turned off.

```
Command: ortho
Enter mode [ON/OFF] <ON>:
Command:
490.4746, 161.1489, 0.0000          SNAP GRID ORTHO POLAR OSNAP OTRACK LWT MODEL
```

Ortho can also be toggled on and off using the **F8** Function key (see Function Keys below for more details). Finally, you can also type ORTHO at the command prompt as in the command sequence below. Using Ortho is the equivalent of using your parallel motion and set square on a drawing board. With Ortho mode turned on you can only draw lines which are either vertical or horizontal. Turn Ortho mode on now and draw some lines to get a feeling for how it works.

Command Sequence

Command: **ORTHO**

Enter mode [ON/OFF] <OFF>: (type **ON** or **OFF**)

Ortho mode is probably the simplest of all the drawing aids, and historically one of the oldest. It is either on or it is off and there are no special settings to make. Also, it does a very simple job; it constrains drawn lines to the horizontal or the vertical. You may not be surprised to learn, therefore, that its use has largely been superceded by more recent features, particularly Polar Tracking, described below.

The Drawing Grid

Status Bar   GRID (right-click for settings)

Pull-down   None

Keyboard   **GRID** or **F7**

The drawing grid is a regular pattern of dots displayed on the screen which acts as a visual aid, it is the equivalent of having a sheet of graph paper behind your drawing on a drawing board. You can control the grid spacing, so it can give you a general idea about the size of drawn objects. It can also be used to define the extent of your drawing. See, Setting Grid Limits, for more details.
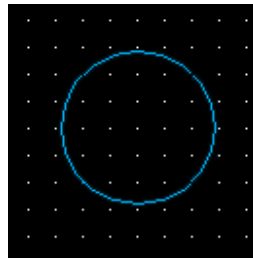
Command Sequence

Command: **GRID**

Specify grid spacing(X) or [ON/OFF/Snap/Aspect] <10.000>: (enter grid spacing)
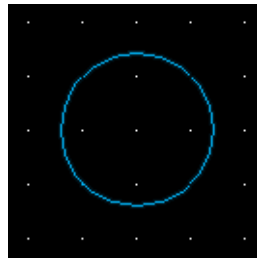

Grid spacing set to
10 drawing units

Grid spacing set to
5 drawing units

Although you can use the command line to control the visibility of the grid by using the "ON" and "OFF" options this is more easily achieved using the **F7**
key or, better still, by clicking the GRID button on the status bar. However, the command line does offer some additional options. The Snap option allows you to automatically set the grid spacing to the current snap spacing (see Snap Mode below). You can also change the aspect ration of the grid. By default, the X and Y spacing of the Grid are the same, resulting in a regular square matrix of grid points. But you can display a grid with different X and Y spacing by using the "Aspect" option.

Grid mode and X/Y spacing can also be set using the Drafting Settings dialogue box. You can access
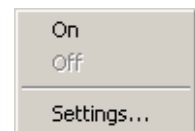
grid settings by right-clicking the Grid button on the status bar and selecting Settings… from the menu. You can also do this from the pull-down menu, **Tools ▶ Drafting Settings…** and click on the "Snap and Grid" tab.

You may have noticed that the grid does not extend infinitely in all directions. In fact, it is only displayed within a finite rectangle. You can control the extent of the visible grid using Drawing Limits.

Setting Grid Limits

Toolbar   None

Pull-down **Format ▶ Drawing Limits**

Keyboard **LIMITS**

Drawing Limits is used to define the extent of the grid display and to toggle Limits mode which can be used to define the extent of your drawing. The grid is displayed within a rectangle defined by two pick points or co-ordinates.

Command Sequence

Command: **LIMITS**

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>:

 (pick point, enter co-ordinates or ↵ to accept the default value)

Specify upper right corner <420.0000,297.0000>:

 (pick point, enter co-ordinates or ↵ to accept the default value)

Drawing Limits can also be used to turn Limits mode on or off. Limits mode can be used to control where objects can and cannot be drawn. Limits is turned off by default which means that there is no restriction as to where points can be picked and objects drawn. When Limits is on, AutoCAD will not allow points to be picked or co-ordinates entered at the command line which fall outside of the specified drawing limits. If you try to pick a point outside the drawing limits when Limits mode is turned on, AutoCAD reports to the command line:

**Outside limits

Limits mode is useful if you know the extent of your plotted drawing sheet and you want to prevent objects being drawn outside of this area. However, Drawing Limits is most commonly used simply to control the extent of the Grid.

Snap Mode

Status Bar SNAP (right-click for settings)

Pull-down None

Keyboard **SNAP** or **F9**

Snap mode takes AutoCAD one step further than the drawing board. With Snap mode turned on AutoCAD only allows you to pick points which lie on a regular grid. The Snap grid is completely independent of the display grid. However, the Grid spacing and Snap spacing are usually set to the same value to avoid confusion. You can force the display grid to conform with the snap grid by setting the display grid spacing to zero. The display grid will then automatically change each time the snap grid is changed. When Snap mode is turned on and the Grid is displayed, the Snap and Grid spacings are the same and the crosshairs will jump from one grid point to another as you move across the screen. This makes it very easy to draw objects which have a regular shape. The Snap command is used to set the snap spacing and to toggle Snap mode.

Command Sequence

Command: **SNAP**

Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <10.0000>: (enter the required snap spacing in drawing units)

Although you can use the Snap command to turn Snap mode on and off, it is much more efficient to use the **F9** function key on the keyboard or to click the SNAP button on the status bar.

The "Aspect" option can be used to vary the horizontal and vertical snap spacings independently.

"Rotate" is used to set the snap grid to any angle.

You can also set the snap style to either *Isometric* or *Standard* (the default) using the "Style" option. The Standard style is used for almost all drawing situations including detail drawings in *Orthographic Projection*. The Isometric style is specifically to aid the creation of drawings in *Isometric Projection* (see the illustrations on the right).

The "Type" option allows you to set the snap type to either Grid (the default) or to Polar. The Polar option can be used in conjunction with Polar Tracking so that Snap mode snaps along polar tracking angles rather than to the grid.
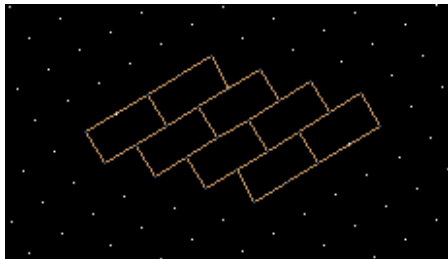
The grid snap is particularly useful if you need lots of modular objects such as bricks or paviors. In the illustration on the left, the Aspect option is used to set the X and Y snap spacings to the brick dimensions and the Rotate option is used to set the orientation of the bond. Once these settings are made, the bricks can be accurately drawn without any other drawing aids.

All of the Snap variables can also be set using the Drafting Settings dialogue box. Right-click on the SNAP button and choose Settings… from the menu.

Drafting Settings

Toolbar   None

Pull-down   **Tools ▶ Drafting Settings…**

Keyboard   **DDRMODES**

The Snap and Grid mode settings, can also be made from the Drafting Settings dialogue box, illustrated below .This dialogue can be invoked from the command line or from the pull-down menu but probably the simplest way is to right-click on either the GRID or SNAP buttons on the status bar and choose "Properties…" from the context menu. The advantage of the Drafting Settings dialogue box is that it gives you one-stop access to all the Grid and Snap settings.
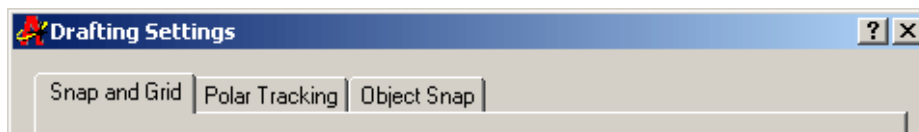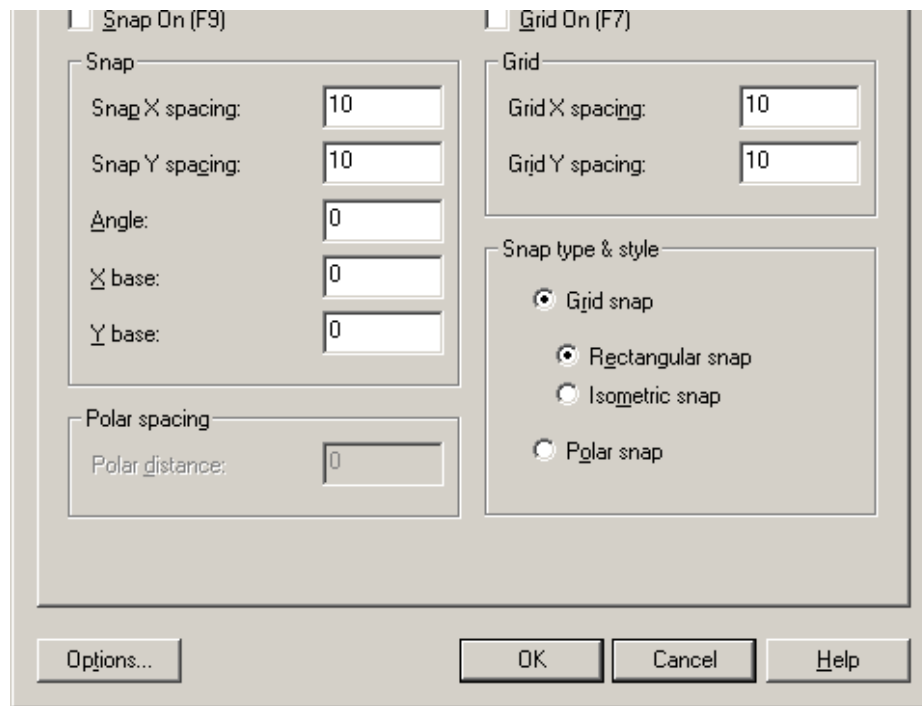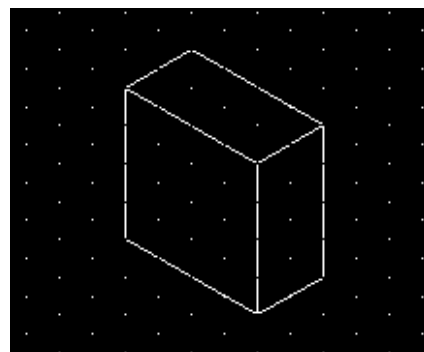
These options act in the same way as those in the respective commands described above, see The Snap Command and The Grid Command
for details. If you are creating drawings in isometric projection, use the "Isometric snap" option to change the grids from the standard *orthogonal* square grid to a 30 degree *isometric* grid.
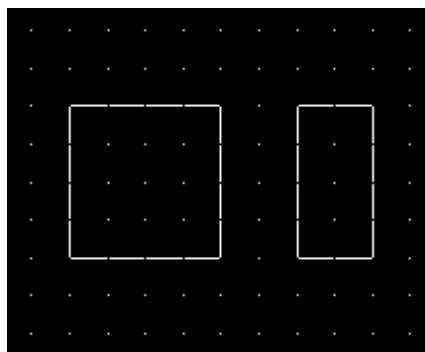
Standard orthogonal grid

Isometric grid

Polar Tracking
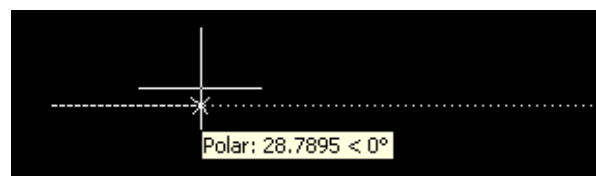
Status Bar   POLAR (right-click for settings)

Pull-down   None

Keyboard   **F10**

Polar Tracking is a bit like Ortho mode on steroids. Whereas Ortho constrains your lines to either the horizontal or the vertical, Polar Tracking allows you to snap into whatever angles you choose to configure.


By default, Polar Tracking snaps only to the horizontal and the

vertical. When you are drawing a line, pick the first point and then move the cursor close to a vertical or horizontal location and your cursor will snap into place. You will see a polar tracking vector and a tool tip which displays the angle and the distance from the pick point; in other words, a relative polar co-ordinate.
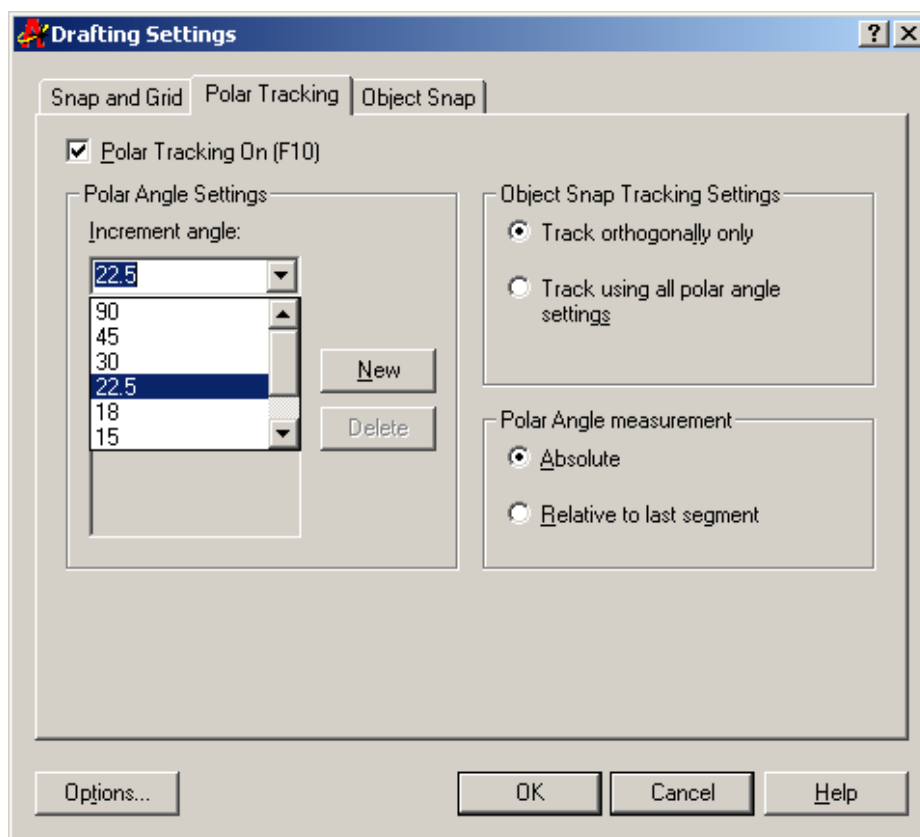
In this default setting, Polar Tracking works like a more flexible version of Ortho but if you look at the Polar Tracking tab on the Drafting Settings dialogue box, you will see just how versatile it can be. Right-click on the POLAR button on the status bar and choose Settings… from the menu.
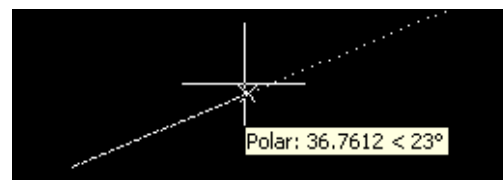
You can use the Increment angle drop-down list to select one of the preset angle increments. For example, if the increment angle is set to 22.5 degrees, Polar Tracking will snap at 22.5 degree increments starting with zero degrees.

Incidentally, you will notice that the reported angle on the tool tip shown in the illustration on the right is "23", whereas the actual snap angle is 22.5 degrees. This is because angular units are set to display only whole degrees and so, 22.5 is rounded up to 23. See the Units and Scales tutorial for more information on changing the precision with which angular measurements are reported.

You may sometimes need to snap to specific angles. Say you are working on a drawing of a site and the buildings are orientated in a particular way. If you know the angle, you can use the Additional angles option to add this specific angle so that Polar Tracking will snap to it.

To set additional angles, all you need to do is click on the New button and enter the value in the list. The Delete button can be used to remove unwanted angles. It is also possible to temporarily suspend Additional angles. You can do this by deselecting the Additional angles check box. When you do this, the angles list is greyed out and those angles won't be used for Polar Tracking until you check the box again.

One of the great benefits of Polar Tracking is that, when used in combination with direct distance entry, you can draw lines of a given length and at a preset angle without using any construction lines and without the need for entering relative co-ordinates. Drawing using this technique can be extremely efficient. See the Direct Distance Entry tutorial for more details.

The Function Keys

Many of the modes described above can be controlled quickly using the keyboard function keys. In most cases this is quicker than using a pull-down or the command line. The function keys are arranged along the top of your keyboard. AutoCAD uses function keys F1 to F11. Their use is described below.

The **F1**
key on your keyboard brings up the "AutoCAD Help: User Documentation" dialogue box. You can use this dialogue box to search for help on any AutoCAD command or topic. To find help on a command or topic, click on the Index tab and enter a keyword. You will usually be given a list of options in the topics list; select the most appropriate and click the "Display" button to see the item.
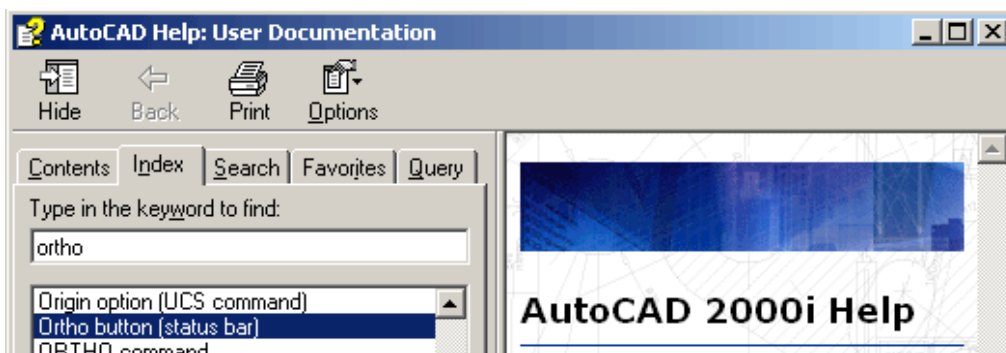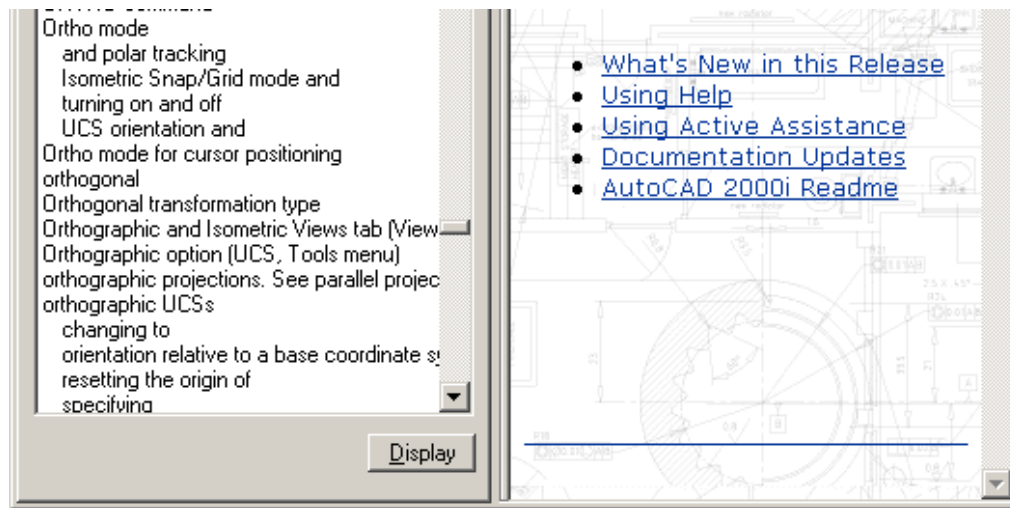
The **F2** key is used to *toggle*
(turn on and off) the AutoCAD text window. This is a floating version of the command window which can be resized to suit your requirements. The text window contains the whole command history from the beginning of the drawing session. If you wish, you can scroll back to see which commands you have used. The text window is also useful for viewing the results of commands like LIST which report to the command line on a number of lines which may scroll off the command window and make them difficult to view.
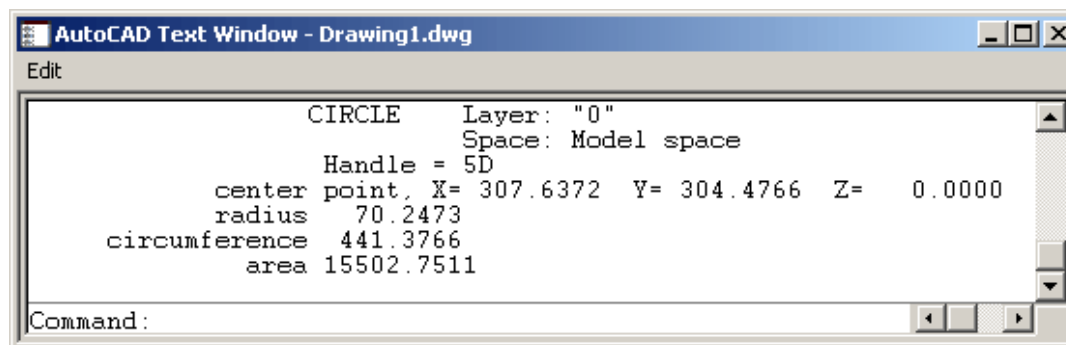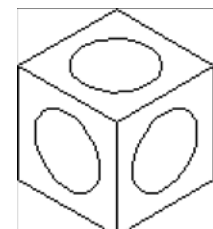

The **F3** toggles running Object Snaps on and off. See the Object Snap tutorial for details on the use of running Object Snaps.

The **F4**
key on your keyboard toggles tablet mode on and off. This only has an effect if a digitising tablet has been calibrated.

The **F5** key cycles through the *Isoplanes*, this only has an effect if "Isometric Snap/Grid" mode

is on. The options are Left, Top and Right. The different options describe the plane in which Ortho mode works. It also affects the orientation of *Isocircles*
drawn with the Ellipse command. The illustration on the right shows a cube with isocircles drawn on the top, left and right faces. Each isocircle was drawn using the corresponding isoplane. See Tips & Tricks for a worked example using isoplanes.

The **F6** key is a three way toggle which changes the co-ordinate reading in the status bar. By default the status bar shows co-ordinates using the Cartesian system. You can use the F6 key to turn the co-ordinate readout off and to change to the polar system when you are in pick mode. For a more detailed description of AutoCAD's co-ordinate systems see the "Using Co-ordinates" tutorial.

The **F7** key is used to toggle grid mode on and off. When grid mode is on a grid of dots is shown on the screen as a drawing aid. You can set the grid spacing by using "Drafting Settings" from the "Tools" pull-down. The grid points do not necessarily reflect the snap setting, they can be set independently, however, you can force the grid to reflect the snap setting by giving the grid setting a value of zero.

The **F8** key on your keyboard can be used to toggle Ortho (orthogonal) mode on and off. When Ortho mode is on AutoCAD will only allow you to draw either vertical or horizontal lines. You can think of it as being a computer version of the parallel motion on your drawing board. You can see if Ortho mode is on by looking at the status bar. The "ORTHO" button is shown "pushed in" when Ortho is turned on.

The **F9** key can be used to toggle Snap mode on and off. Snap makes the *crosshairs* jump to points on a defined grid. The snap spacing can be set using the "Drafting Settings" dialogue box from the "Tools" pull-down menu. You can also see if *Snap* mode is on by looking at the status bar.

The **F10** key is used to switch polar tracking off and on. Polar Tracking allows you to snap to specific angles and these are user definable. See Polar Tracking for more details.

The **F11** key toggles object snap tracking on and off. See the Object Snap tutorial for more information on object snap tracking.

Tips & Tricks
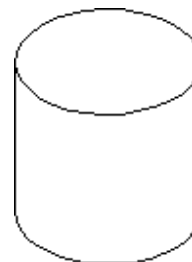
One of the most difficult aspects of drawing in isometric projection is the correct representation of circles. Obviously a circle in isometric projection looks like an ellipse (see illustration) but knowing exactly what aspect ratio to draw the ellipse at is difficult. Fortunately AutoCAD makes the whole process very simple. When "Isometric Snap/Grid" mode is turned on, the ellipse command gains an extra option, the "Isocircle" option. Follow the exercise below to draw a cylinder in isometric projection.

Note: During this exercise, you will be using the Quadrant Object Snap. See the Object Snap tutorial for more information on the use of Object Snaps.

Step 1 - Start a new drawing
Start AutoCAD and use the "Start from Scratch" option from the "Start Up" dialogue box. If you are already using

AutoCAD, create a new drawing by clicking on the [ ] button and use "Start form Scratch" from the "Create New Drawing" dialogue box.

Step 2 - Make the Drafting Settings
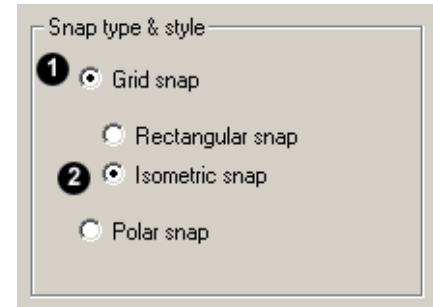
Display the Drafting Settings dialogue box by selecting Drafting "Settings..." from the "Tools" pull-down menu. Click on the "Snap and Grid" tab. In the Snap type & style section of the dialogue, set the type to "Grid snap" and the style to "Isometric snap", as shown in the illustration. Now, check the two boxes at the top of the dialogue, once for "Snap On" and once for "Grid On". Click on the "OK" button to confirm these mode changes. The graphic window now displays a grid of dots arranged at an angle of 30 degrees and the crosshairs will jump from one dot to another. Notice also, that the crosshairs are oriented in the left hand isoplane.

Step 3 - Setting the correct isoplane

In this exercise, we will draw a cylinder which stands vertically. The circles which we draw must, therefore be drawn in the "Top" isoplane. Use the **F5** key on the keyboard to change the isoplane to "Top". AutoCAD reports to the command line:

```
Command: <Isoplane Top>
```

Step 4 - Drawing the base circle

Circles in isometric projection are drawn using the Ellipse command. Start the Ellipse command by clicking on the [ ] button or by selecting **Draw ▶ Ellipse ▶ Axis, End** from the pull-down menu. Now look at the command line:

Command: **_ellipse**
Specify axis endpoint of ellipse or [Arc/Center/Isocircle]: **I** (Isocircle)
Specify center of isocircle: (pick a point in the lower half of the graphics window)
Specify radius of isocircle or [Diameter]: **30** (enter a radius of 30)

Your drawing should look like the one in the illustration above.

Step 5 - Copying the base circle

Start the Copy command by clicking on the [ ] button or selecting **Modify ▶ Copy** from the pull-down menu. Now look at the command line:

Command: **_copy**
Select objects: (pick the circle)

Tip: If you find picking the circle difficult, use the F9 key to turn off Snap.

Select objects: ↵

Specify base point or displacement, or [Multiple]: (pick the grid point in the centre of the circle)

Note: Use **F9** to turn Snap back on if you turned it off.

Specify second point of displacement or <use first point as displacement>: (move the crosshairs vertically by 6 grid points and pick)

You should now have two isometric circles, one above the other.

Step 6 - Drawing the sides

Start the Line command by clicking the [ ] button or selecting **Draw ▸ Line** from the pull-down menu. Now look at the command line.

Command: **_line**

Specify first point: (use the Quadrant Osnap [ ] to pick the left-hand quadrant of the lower isocircle)

Tip:

There are a number of ways to invoke osnaps, they are available from the Osnap toolbar and from the keyboard. However, in this case it may be simplest to select Quadrant from the cursor menu. To do this, hold down the Shift key on the keyboard and click on the right hand mouse button. A menu will appear at the crosshair position. Simply select Quadrant from the menu.

Now, move the crosshairs near to the left hand quadrant point on the lower isocircle. You will see a yellow diamond appear at the quadrant point (see illustration). Pick the point.

Specify next point or [Undo]: (use the Quadrant Osnap again to pick the left-hand quadrant point on the upper isocircle)

Specify next point or [Undo]: ↵ (to end the Line command)

Now repeat this process to draw the right hand line or use the Copy command to copy the left hand line to the right. Remember to use the Quadrant Osnap to pick points whichever method you use. This will ensure that the line is drawn or copied in exactly the right place. When you have completed this step, your drawing should look similar to the illustration on the right.

Step 7 - Trimming the circle

To complete the drawing we will remove the upper half of the lower isocircle to give the impression of a solid cylinder. To

do this we will use the Trim command. Start the Trim command by clicking the [icon] button or selecting "Trim" from the "Modify" pull-down. Now look at the command line:

```
Command: _trim
Current settings: Projection=UCS Edge=None
Select cutting edges ...
Select objects: (pick the two vertical lines)
```

Tip: You may need to turn Snap off (F9).

```
Select objects: ↵
Select object to trim or [Project/Edge/Undo]: (pick the upper arc of the lower isocircle)
Select object to trim or [Project/Edge/Undo]: ↵ (to end the Trim command)
```

The isometric cylinder is now complete. Use **F7**
to turn the grid off and your drawing should look similar to the one in the illustration at the beginning of this exercise.