Tutorial 2: Setting up the Drawing Environment

Drawing size

With AutoCAD all drawings are done to FULL SCALE. The drawing limits will depend on the size of the items being drawn. For example if our drawing is the plan of a floor 23.8m X 15m then we make the limits of the drawing big enough to contain the whole floor plus other items such as margins, title boxes etc. The preferred drawing unit used in AutoCAD is millimetres, so a good size for this case would be 30000 X 20000 units or millimetres.

Drawing this way makes it possible to plot the drawing to any scale and on any size of paper as long as the plotter allows it. For the example above we need to set the limits to 30000 mm X 20000 mm. To do this type LIMITS at the command prompt:

Command: LIMITS
Reset Model space limits:
Specify lower left corner or [ON/OFF] <0.0000,0.0000>: (to accept the default)
Specify upper right corner <420.0000,297.0000>: 30000,20000

At this point the screen is still displaying the old 420 X 297 size and we need to magnify the screen to the new size. This is done by typing:

Command: ZOOM
Specify corner of window, enter a scale factor (nX or nXP), or [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: A
Regenerating model.

Electronic set squares

The ORTHO command provides a tee square allowing you to draw horizontal and vertical lines with 100 % accuracy. When ORTHO is switched on by pressing the function key F8 the screen cursor movement is restricted to horizontal and vertical only no matter where you position the mouse. The word "ORTHO" should appear on the top line of the screen beside the layer name. The ORTHO mode can be switched on or off even during commands by pressing the ORTHO function key at the top of the keyboard (F8). It can also be switched on by typing the command line e.g.

Command: ORTHO
Enter mode [ON/OFF] <OFF>: ON
**Rulers, grids and snapping**

To make the display appear somewhat like graph paper one can use the GRID command. This produces a grid of dots to help your positioning of the crosshairs. To have a grid to match the axes created above do the following:

- Click on Tools Menu
- Click on Drafting Settings
- In the GRID section enter 5 in the X spacing box
  (The Y spacing changes automatically to 5 by highlighting the Y box)
- Check the ON box
- Click on OK

Since you are set the limits to (0,0) and (30000,20000) and zoomed all, grids of 5 millimeters will be too dense to display on the screen. But if you zoom to a very small window on your screens, you will be able to see the grids.

The grid can be toggled on or off in the same manner as the ortho mode by using the special function key on the keyboard (F7).

The grid is a good guide but does not guarantee accuracy in positioning the crosshairs. If all the items of your drawing can be measured, say, to 2.5mm (ie 1/4 cm) then it makes sense to make that the smallest size to construct your drawing. The SNAP command makes the crosshairs lock into alignment with an imaginary grid which may or may not have the same value as the GRID of dots (though it makes sense to keep them similar).

- Click on Tools Menu
- Click on Drafting Settings
- In the SNAP section enter 5 in the X spacing box
  (The Y spacing changes automatically to 5 by highlighting the Y box)
- Check the ON box
- Click on OK

The crosshairs will now 'jump' in steps of 5mm and so a point (1275,825) can be located with 100% accuracy using the mouse. The SNAP can be toggled on/off by using the function key F9 or pressing the SNAP button in the status area at the bottom of the screen.

*Typing in a co-ordinate location at the keyboard overrides both ORTHO and SNAP.*

The subcommands of SNAP, Value/Aspect/Rotate/Style allow you to choose the size of the imaginary grid spacing, to change the Y spacing, to rotate the whole grid and finally to have either a normal grid or an isometric grid to snap onto.
**Layers**

Having decided on the size it is now worth considering how your drawing is to be organised. Using layers is the most efficient way of doing this.

Most CAD drawings depict a number of different kinds of objects in the same drawing. For example, an architectural drawing might show walls, doors, windows, and dimensions.

To organize your drawing, and to reduce the visual complexity of your drawing as you work on it, you can assign objects to different layers. You can imagine layers as sheets of plastic that contain the components of your drawing. You can choose to display or to hide objects on a layer. You can also lock a layer to prevent making changes to objects on that layer.

Objects that are created on a layer assume the color, linetype, and linewidth properties of the layer.

It is very important to establish layer organization before beginning a drawing. Most firms standardize the layer names and property assignments in their CAD drawings.

When you start a new drawing there is only one layer present, the default layer 0. Extra layers can be set up at any stage during the drawing session but it is advisable to decide on the structure of the layers before commencing the drawing. Each line type e.g. dash, continuous etc. requires a separate layer. As an example our drawing DRAW will contain different layers as we try out some drawing commands, some text, and some dashed construction lines. The following sequence creates and gives names to three new layers.

Click on **LAYERS** icon (Top left corner of the screen)
(There is no image of the LAYERS dialogue box available for this text.)
Click on the NEW LAYER button
Type in **LINE** <ret>
Click on the NEW LAYER button
Type in **TEXT** in the box <ret>
Click on the NEW LAYER button
Type in **CONSTRUCTION DASH** in the box <ret>
Click on the LINETYPE tab, click on continuous
Click on LOAD button
Click on DASHED
In the Select Linetype Dialogue box select DASHED
Click on OK in Select Linetype Dialogue box
Click on OK in the Layer Properties Manager window

Layers can have alphanumeric names up to 255 characters long and these names should give full description of what is contained on the layer.
To draw on a particular layer it must be 'On' and 'Set' to the current layer. Click in the LAYER dialogue box scroll down and pick the layer you wish to work on then click Current and OK eg the CONSTRUCTION-DASHED layer.

All layers are initially 'On' and of line type 'Continuous'.
Exercise

Change the limits to 8000, 6000 and Zoom all
Set the Grid and Snap to 300
Draw a box 4000X4000 starting at point 2000, 1000

Exploding Entities

The command Explode breaks a compound object into its component objects. A compound object comprises more than one AutoCAD object. For example, a rectangle is a compound object comprising four lines.

Click on Modify then Explode or click on or type X in the command line.
Select objects: click any point on the square and press

Offsetting

Click on Modify then Offset or click on or type O in the command line.
Input offset distance: 1000 and press return
Click on the base of the square (Point 1) and then inside the box
Click on the top line of the square (Point 2) and then inside the box again then press return.
This will offset the two sides inside the box, as seen in the following figure

Click on Modify then Offset
Input offset distance: 1400 and press return
Press return or click on the right hand button on the mouse
Click on the left line of the square (Point 3) and then inside the box
Click on the right line (Point 4) and then inside the box again then press return

Trimming

Click on Modify then Trim or click on or type TR in the command line.
Using the mouse draw a window around the box by clicking on two points outside the box, bottom left and top right
Press return or click on the right hand button on the mouse
Trim the lines on the shape one by one until you end up with an I shape as seen in
the following figure

Hatching
Click on Draw and then Hatch
Click on Pattern on the Hatching dialogue box
Pick the Angle Hatching Pattern
Click on Add: Pick Points in the Boundaries window.
Click on any point inside the shape
Press return or click on the right hand button on the mouse
Change the Scale to 20 and the Angle to 45 then Click on OK.

Dimensioning
The DIMension command in AutoCAD allows you to add annotations to drawings
showing the lengths of objects or the distances or angles between objects. You simply
specify what you want dimensioned and the program draws the lines, arrows and text. In
the following example use the mouse to pick the commands and co-ordinate location.

1. Click on the TOOLS menu on top of the screen
2. Click on Drafting Settings
3. Click on Object Snap
4. Check the Endpoint and Midpoint boxes
5. Click on OK
6. Click on Dimension, Dimension Style, Modify
7. Click on Symbols and Arrows
8. In Arrowheads window, change Arrow size to 250
9. Click on Text
10. In Text Appearance window, change Text height to 300
11. Press OK then close
12. Click on the DIMENSION menu on the top of the screen
13. Then click on LINEAR from the menu. The command line will then display:
   Specify first extension line origin or <select object>:
14. Using the mouse move close to the starting point of a horizontal line in the rectangle you drew. A box will appear on the point and the text Endpoint will be displayed. Click (LB) on the point.
15. Pick the end of the horizontal line. Again a box will appear at the end of the line. Click on the point. The command line will display:
   Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:
16. Pick a point outside the rectangle and next to the horizontal line. Command line will display:
17. Dimension text= 4000 Repeat the same process to dimension a vertical line on the rectangle, clicking on the DIMENSION menu and LINEAR, again.