Chapter 6

Lessons in AutoCAD

Most of these lessons are filtered from http://www.we-r-here.com/cad/tutorials. However, the following lessons are edited and in few cases there are additional information, figures and other things.

6.1 Lesson 1

The X,Y co-ordinate system

Everything that you draw in AutoCAD is exact. It will be more accurate than you will ever need it to be. All objects drawn on the screen are placed there based on a simple X,Y co-ordinate system. In AutoCAD this is known as the World Co-ordinate System (WCS).

Angular Measurement

When drawing lines at an angle, you have to begin measuring the angle from 0 degrees, that is the positive X-axis (Isn't it 3 o'clock position of the hour-needle in the watch?). The rotation is counter clockwise.

6.1.1 Entering Points in AutoCAD

1. *ABSOLUTE CO-ORDINATES* - Using this method, you enter the points as they relate to the origin of the WCS. To enter a point just enter in the exact point as X,Y.
2. *RELATIVE CO-ORDINATES* - This allows you to enter points in relation to the first point you have entered. After you've entered one point, the next would be entered as @X,Y. This means that AutoCAD will draw a line from the first point to another point X units over and Y units up relative to the previous point.
3. *POLAR CO-ORDINATES* - You would use this system if you know that you want to draw a line a certain distance at a particular angle. You would enter this as @D<A. In this case, D is the distance and A is the angle. Example: @10<90 will draw a line 10 units straight up from the first point.

The three ways of entering co-ordinates shown above are the ONLY way AutoCAD accepts input. First decide which style you need to use, and then enter as shown. Remember that X is always before Y (alphabetical). Don't forget the '@' symbol when you are entering relative points. Any typing error or omission will give you results you don't want. If you make a mistake and need to see what you typed, press F2 to bring up the text screen and check your typing. (Press F2 to get back to your drawing.)

6.1.2 Icons, Keystrokes and Menus

Example: If you want to draw a line, you can do it a few ways:

1. At the command line type: LINE (or) L and press the ENTER key.

2. Select the line icon from the DRAW toolbar.
3. Select Draw > Line from the pull-down menu

Tips: *Easy way of drawing straight lines. Make the 'Ortho' tab 'on' (it's a tab at the bottom with others, click to turn it on or off). Then start drawing lines, select first point and then point the mouse anywhere in the desired direction, then give the distance (e.g. 4) and then press Enter.*

6.2 Lesson 2

6.2.1 Introduction to Drawing and Modify Commands

Start the LINE command and draw a line from **1,2** to **3,2** to **3,4** to **1,4** Press enter after each point. (*Remember to watch the command line as you do this.) For the last line, you can either type in **1,2** or **C** to close the line back to the first point you entered. You have just drawn a 2" square using **absolute co-ordinates.** Next draw a similar box using **relative co-ordinates.** Start the **LINE** command and begin at point 4.5,2. From there draw a line two units to the right by typing @2,0 (this means 2 units in the X direction, 0 units in the Y direction). Next type @0,2 then @-2,0 then @0,-2 to finish the box. (Remember to press enter after each point.) Draw the next box using **polar co-ordinate** input. Start the LINE command and begin at point 8,2 then enter. Type @1<45 to draw the first line. Next enter @1<135 then @1<225 then @1<315 (or C to close). What you have just done is drawn a line 1 unit long at 45 degree, then another at 135 degree and so on. Start the CIRCLE command and add a circle that has a center point at 7,6 with a radius of .75 (Watch the command line for instructions).

6.2.2 Basic Drawing Skills

Command	Keystroke	Icon	Menu	Result
Rectangle	RECTANGLE / REC		Draw > Rectangle	Draws a rectangle after you enter one corner and then the second.
Multi Lines	MLINE / ML	No Icon	Draw > Multiline	Draw parallel lines based on the parameters you define.
Trim	TRIM / TR		Modify > Trim	Trims objects to a selected cutting edge.
Extend	EXTEND / EX		Modify > Extend	Extends objects to a selected boundary edge.
Offset	OFFSET / O		Modify > Offset	Offsets an object (parallel) by a set distance.
Object Snaps	OSNAP / OS / F3	CLICK 	Tools > Object Snap Settings	Brings up the OSNAP dialog box.

Example:

6.2.3 TRIM command

Draw a line from **2,5** to **2,6.5** Draw another line from **1,6** to **3,6** You should now have two perpendicular lines. What you want to do is trim off the top of the vertical line and create a T.

Start the TRIM command. It will first ask for a cutting edge. Select the horizontal line and press <ENTER>. It will now ask for the object to be trimmed. Select the vertical line anywhere above the horizontal (cutting) line and press <ENTER> to finish the command.

This is what you saw on the command line:

Command: **TR** <enter> TRIM

Current settings: Projection=UCS, Edge=None

Select cutting edges ...

Select objects: <Select the Horizontal line>1 found

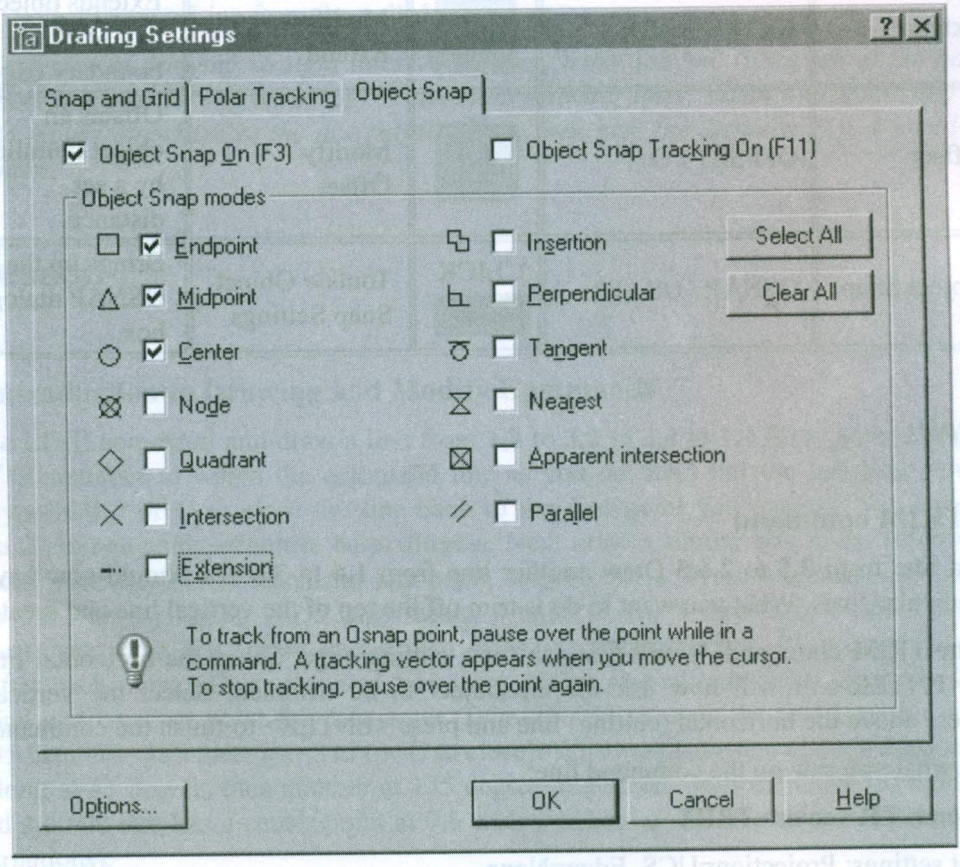
Select objects: <enter>

Select object to trim or shift-select to extend or [Project/Edge/Undo]: <Select the vertical line> Select object to trim or shift-select to extend or [Project/Edge/Undo]: <enter>

Once again, it is important to keep your eye on the command line as it will guide through most commands.

6.2.4 Object Snaps

Suppose you want to draw a line from the center of the circle to the middle of the vertical line you extended earlier. AutoCAD has a feature that makes this very easy. These are the Object Snaps (or Osnaps "Oh-Snaps"). Type OS <ENTER> . You will see this dialog box appear.



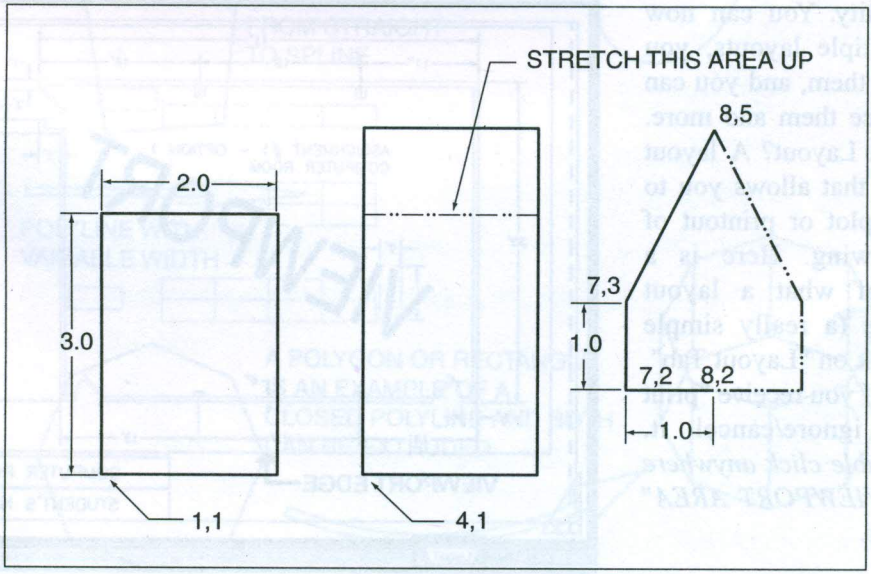
6.2.5 More Modifying Commands

All of these commands are ones that you will use on a regular basis.

Command	Keystroke	Icon	Menu	Result
Move	Move /M		Modify > Move	Moves an object or objects
Copy	Copy /CP		Modify > Copy	Copies object(s) once or multiple times
Stretch	Stretch /S		Modify > Stretch	Stretches an object after you have selected a portion of it
Mirror	Mirror /MI		Modify > Mirror	Creates a mirror image of an object or selection set

Exercise:

Create a box as shown on the left, then move and stretch, and then on the right side, create a mirror image using the left half segment.

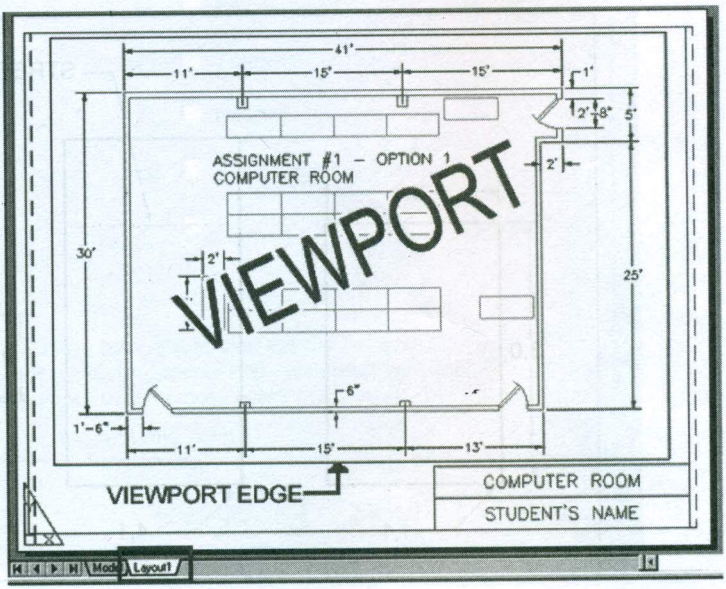


More Modifying Commands

Command	Keystroke	Icon	Menu	Result
Rotate	Rotate /RO		Modify > Rotate	Rotates objects to a certain angle
Fillet	Fillet /F		Modify > Fillet	Creates a round corner between two lines
Chamfer	Chamfer /CHA		Modify > Chamfer	Creates an angled corner between two lines
Array	Array /AR		Modify > Array	Creates a repeating pattern of the selected objects

6.2.6 Layout Tabs - Paper Space & Model Space

In AutoCAD there are two different workspaces: **model and Layout**. For now think of model space where you make your model, or draw. Think of the **Layout Tabs** as where you print your drawing from, or copy and output it to paper. Look at the images below to see a visual explanation of the concept. After AutoCAD R14, the term "Paper Space" was replaced by "Layout". These terms are interchangeable. The release of AutoCAD 2000 brought other enhancements to this functionality. You can now have multiple layouts, you can name them, and you can re-sequence them and more. What is a Layout? A layout is a page that allows you to set up a plot or printout of your drawing. Here is a sample of what a layout looks like (a really simple one). Click on "Layout Tab", and then if you receive "print option", ignore/cancel it. Then *Double click anywhere on the "VIEWPORT AREA" to activate.*



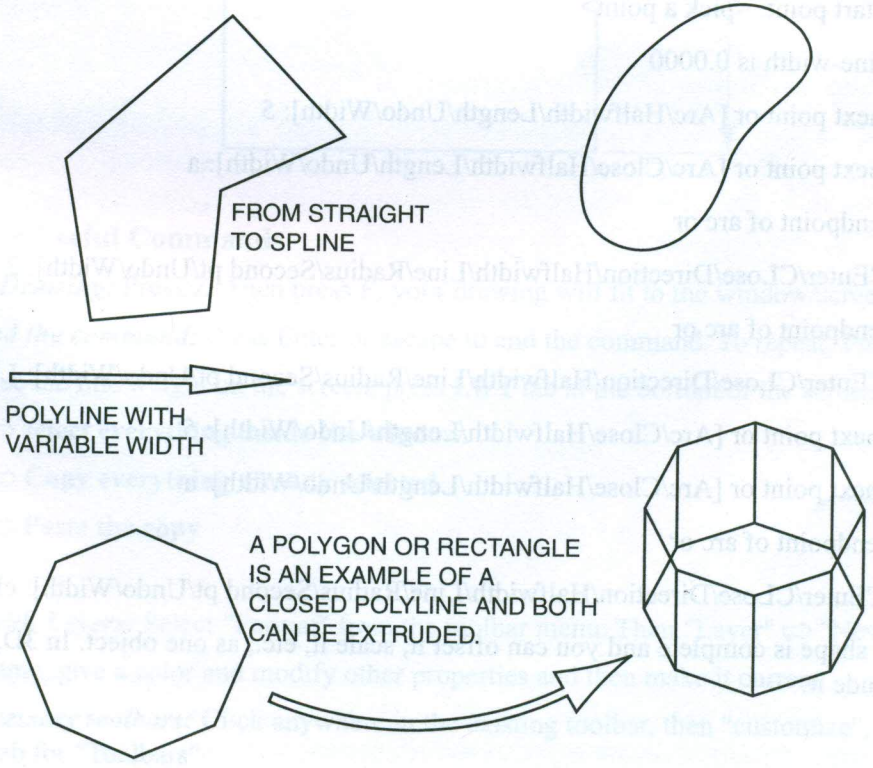
6.2.7 Polylines

A polyline is an object in AutoCAD that consists of one or more line (or arc) segments. A rectangle is an example of a polyline that you are already familiar with. As you've seen, it is one object that can be modified and worked with easier than four separate lines.

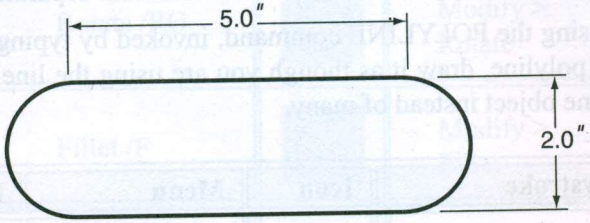
Polylines are created using the POLYLINE command, invoked by typing PL at the command line. To draw a simple polyline, draw it as though you are using the line command. The only difference is that it is one object instead of many.

Command	Keystroke	Icon	Menu	Result
Polyline	Pline / PL		Draw > Polyline	Creates a polyline of arcs and/or lines.
Polyline Edit	Pedit / PE		Modify > Polyline	Edits polyline objects

Examples of Polyline



Below is a sample drawing you can reproduce using a polyline. First, think about how you would draw it using lines, arcs and circles.



Here's how you draw it using one polyline:

Command: <ortho on>
(click the ort ho tab)

Command: pl
(PLINE)

Specify start point: <pick a point>

Current line-width is 0.0000

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 5

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: a

Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: 2

Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: Line

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 5

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: a

Specify endpoint of arc or
[Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]: cl

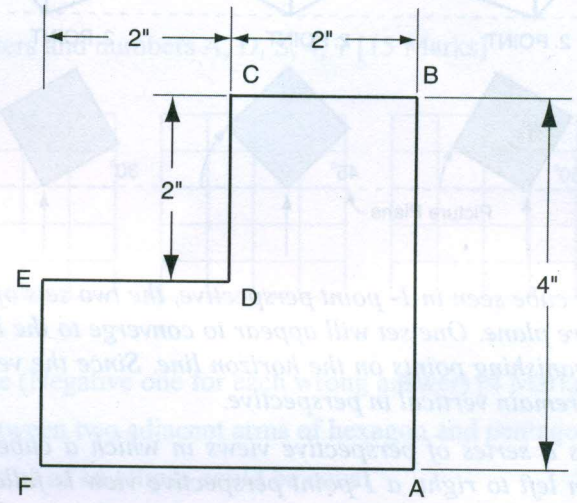
Now the shape is complete and you can offset it, scale it, etc., as one object. In 3D, you could also extrude it.

6.3 Lesson 3

6.3.1 Finding Area of Any Shape

This is a neat feature. Draw the figure below. Now, select **'Tools' ⇨ 'inquiry' ⇨ 'Area'**

Then select the points A, B, C, D, E, F and at the end A. Then Press Enter. You should see the answer for the area and the perimeter.



6.3.2 Some Useful Commands

Zoom the Drawing: Press Z. Then press E. your drawing will fit to the window/screen.

Repeat/End the command: Press Enter or escape to end the command. To repeat, Press Enter

LWT: To see the line weight on the screen, press LWT tab at the bottom of the screen.

Ctrl + A ⇨ select everything inside the window.

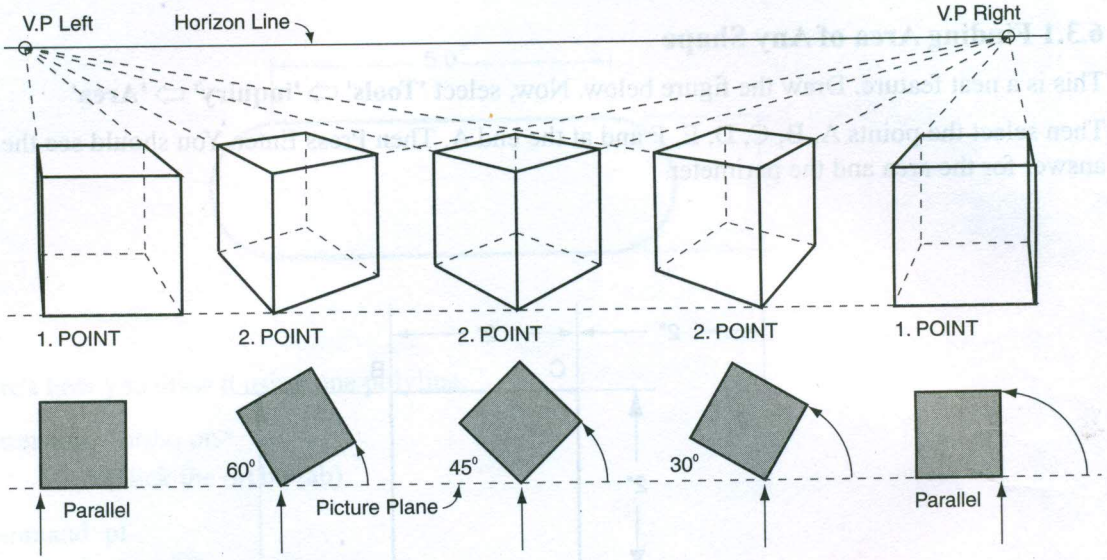
Ctrl + C ⇨ Copy everything already selected

Ctrl + V ⇨ Paste the copy

Working with Layers: Select "Format" from the toolbar menu. Then "Layer" ⇨ "New" Select a Name, give a color and modify other properties and then make it current

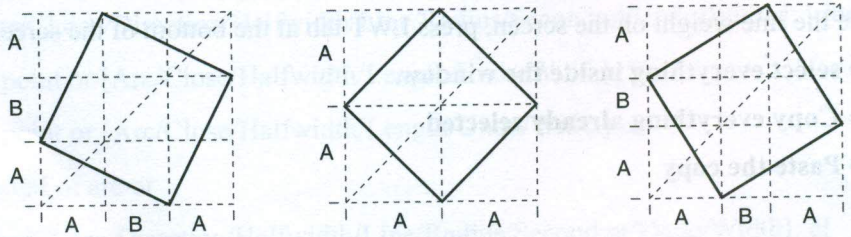
To add necessary toolbars: Click anywhere in the existing toolbar, then "customize", then select the tab for "Toolbars"

Drawing, a Creative Process[56] (Francis D.K. Ching, 1990)



When we rotate the cube seen in 1- point perspective, the two sets of horizontal lines are both obique to the picture plane. One set will appear to converge to the left, the other to the right, thus we have two vanishing points on the horizon line. Since the verticals are parallel to the picture plane, they remain vertical in perspective.

Illustrated above is a series of perspective views in which a cube rotates about a forward vertical edge. From left to right, a 1-point perspective view is followed by a sequence of 2-point perspective views. The cube finally rotates back to where its front face is parallel to the picture plane. It is thus seen in 1-point perspective.



For the purpose of sketching a 2-point perspective view, we can build on our familiarity with a 1-perspective cube. We must first understand that a square can be rotated at any angle and be inscribed within a larger square, as shown above. In each case, note that the inscribed square touches the larger square at points equally distant from each corner.