Topics covered in this Lesson:

Welcome to the world of CAD - In this tutorial you will be learning the basics of CAD. The course is designed so that the commands and instructions should work on almost any version of AutoCAD, although this version is designed specifically for AutoCAD 2010 and will work great for 2012. By the end of this level you will have the skills to develop basic 2D drawings and print them out to scale. For an introduction to what CAD is all about, check out my Blog post.

Let's start at the beginning, these things you need to know, or the rest of it won't make any sense at all. Make sure you have a very good understanding of this lesson before continuing. This lesson is longer than most, but will cover important topics. Learn it, live it.

The X,Y coordinate system

Everything that you draw in AutoCAD is **exact**. It will be more accurate than you will ever need it to be. We're talking 14 decimal points accurate. All objects drawn on the screen are placed there based on a simple X,Y coordinate system. In AutoCAD this is known as the World Coordinate System (WCS). You must understand this to know how to put things where you want them. *(3-D work has an added axis, the Z-axis, but this is not covered in this lesson.)* Below is a diagram showing you how this system works (place your mouse on the diagram for more info).

In order to work effectively with AutoCAD, you **have** to work with this system. Until you are comfortable and familiar with it, learning AutoCAD will be more of a chore. My experience in teaching is that the better a student is with coordinates, the better CAD draftsperson they will become.

Here is how it works:

AutoCAD uses points to determine where an object is located. There is an origin where it begins counting from. This point is (0,0). Every object is located in relation to the origin. If you were to draw a line straight out to the right from the origin, this would be considered the positive X-axis. If you were to draw a line straight up, this would be the positive Y-axis. The picture above shows a point located at (9,6). This means that the point is 9 units over in the X-axis and 6 units up in the Y-axis. When you are working with points, X **always** comes first. The other point shown is (-10,-4). This means that the point is 10 units in the negative X-axis (left) and 4 units in the negative Y-axis (down).

A line has two points, a start point and an end point. AutoCAD works with the points to display the line on the screen. Move your cursor over the picture above and you will see line drawn from the absolute points of (-10,-4) to (9,6).

Most of the time you will not have an indication of where the origin is. You may need to draw a line from the endpoint of an existing line. To do this you use relative points. These work the same way, but you have to add the @ symbol (shift+2) to tell AutoCAD that this next point is relative from the last point entered.

To review:
ABSOLUTE POINTS are exact points on the drawing drawing space.
RELATIVE POINTS are relative to an OBJECT on the drawing space.

Its a simple system, but mastering it is the key to working with AutoCAD and is explained in more detail further below.

Angular Measurement

AutoCAD measures angles in a particular way also. Look at the diagram below and then place your mouse on it to see how this is done.

When drawing lines at an angle, you have to begin measuring the angle from 0 degrees, which is at the 3 o'clock position. If you drew a line at 90 degrees, it would go straight up. The example above (when you move your mouse over it) shows a line drawn at +300 degrees (270+30), or -60 degrees.

You might not always have an obvious reference point for 0 degrees. Look at the example below and place your mouse on the image to find out the angle in question.

In this example, you are given information about the lines, but not the angle AutoCAD needs to draw the line from the start point. What you are given though, is (a) the knowledge that 0° is at the 3 o'clock position (b) the knowledge that 180° is at the 9 o'clock position and (c) the angle between 180° and the line you want to draw is 150°. With this information, you can figure out what angle you need. Here is a fool-proof way of getting the angle you need:

1.) Start at the 0° position and measure counter-clockwise (+) to 180°.

2.) From 180°, measure clockwise 150° (-)

3.) Consider that you just went +180-150 and use that as an equation: +180-150=30

4.) Now you can draw your line using polar coordinates (discussed below)

Entering Points in AutoCAD

You can enter points directly on the command line using three different systems. The one you use will depend on which is more applicable for the situation. The first assignment will get you used to this. The three systems are as follows:

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.

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.

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 coordinates 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.)

More AutoCAD Basics

Subjects covered in this section:
The AutoCAD screen - Workspaces - Starting Commands - Terminology

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The AutoCAD Screen

Move your cursor around the image above to find the names of various areas of the screen.

- Application Button - This button displays commands for printing, saving, drawing utilities and other non-drawing tool.
- Quick Access Toolbar - This is for quick access to common commands like New, Open, Save, Plot
- Filename - The name of the current file you are working on.
- Search Bar - Search for text in your drawing or search the help files.
- Ribbon - The Ribbon has most of the commands/tools that you will use while you are working.
- Tabs - A series of Tabs make up the Ribbon (Home, Insert, Manage, etc) and organize the Tools into common groups.
- Panels - Contain a group of tools
- Tools - These are the icon that start the commands you use to draw, modify, etc.
- Tool Tip - If you hover your mouse over a tool, a tool tip will appear to give your more information. Hold it longer for more info.
- Drawing Space - These is where you draw your designs.
- Command line - When you type a command, you will see it here. AutoCAD uses this space to 'prompt' you for information. It will give you a lot of information and tell you where you are in the command. Watch this line while learning.
- Status bar - This allows to see and change different modes of drawing such as Ortho, Osnaps, Grid, Otrack, etc. You can right click this area to toggle between icons and text for this area.

Workspaces

With the introduction of AutoCAD 2009, a new screen layout was added. The program now allows you to work in different workspaces depending upon what you are working on. For example, the screen will look different if you are working on 2D than it will with 3D work. There is also an option for AutoCAD Classic (which is how the screen looked from Versions 2000-2008). This set of tutorials will deal with the new new workspace. Since you are a new user, you may as well learn the new interface. If you want to use the Classic interface, use the tutorials for AutoCAD 2008.

For the first 2 levels of tutorials, you will want to be in the 2D Drafting & Annotation workspace. Set this by clicking in the bottom right of the AutoCAD screen on the 'gear' icon as shown in the image below. In AutoCAD 2012, this is at the top of the screen.

Icons, Keystrokes and Menus

There are many ways to do things in most Windows programs. AutoCAD is no exception.

Everyone will develop a way that works best for him or her. In this course, we will primarily be working with the keystroke commands. The reason for this is because they will work in most AutoCAD versions (including DOS versions), and in some other CAD programs. The icons work well, but as you will see, icons can be placed anywhere on the screen and can be difficult to find quickly. You may be working on another employee's computer that is set up differently than than what you're used to. The pull-down menus will access almost all commands, but are a slower way of doing things. Icons in AutoCAD 2010 are found on the ribbon, divided into panels - just click on the appropriate tab to open the panel you need..

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

- At the command line type: LINE (or) L and press the ENTER key.
- Select the line icon from the DRAW Panel..

All three approaches will do the same thing: prepare AutoCAD to draw a line where you tell it.

AutoCAD is a popular program because it can be customized to suit an individual's needs. The toolbars are a good example of this. You can have the toolbars you use most often on the screen all the time. You can easily make them go away so that you have more drawing space. You can also customize them so you have the most common commands on one toolbar. For example, the dimensioning toolbar is one that you will not want taking up space on your screen while drawing, but is very handy when you're dimensioning your drawing.

To remove the ribbon and have the most drawing space available, click on the "Clean Screen" icon in the bottom right corner of the screen. To go back the to the standard display, click again on the same icon.

Basic AutoCAD Terminology

Here are some basic terms that you will want to review before using AutoCAD. Some terms have links to give you more information (but it is not necessary to memorize them all now).

Absolute coordinates	A way of inputting points based on AutoCAD's origin.
Acad.dwt	This is the default template that automatically loads whenever you start a drawing session. It can be customized to suit your needs.

Associated Dimensioning	Dimensions that are associated with specific points will update as that point is moved.
Backup file	AutoCAD can be set to automatically backup your drawing and save it. This is a safeguard in case your file gets corrupted. It is saved with a .BAK extension
Block	A pre-drawn image you can insert in your drawing to save time and make your file size smaller.
Clean Screen	A display setting that gives you maximum drawing space.
Crosshairs	This is your cursor when it is in the drawing space.
Cursor	Your cursor will change depending on what function it is performing in the program.
Database	An AutoCAD drawing file is actually one large database containing all the information needed to reproduce the objects when the file is opened. Info for layers and linetypes, etcare stored in this manner.
Dialog box	AutoCAD uses a large number of dialog boxes to get information from you. You must know how input the information that it asks for.
Drawing template file	This is a file that contains preset values for frequently used settings. AKA a prototype drawing. The file extension is DWT.
Extents	The outer boundaries of the objects you have drawn.
Grid	This is pattern of dots displayed on the screen to guide you. It can be toggled on and off by pressing the F7 key.
Grips	Small 'handles' on objects that allow for quick editing.
Layer	All objects are drawn on a layer. You can group objects (such as electrical) on a single layer and organize your drawing.
Layout Tabs	A space used for plotting your drawings (formerly called Paper Space).
Limits (Grid)	A setting to impose an 'artificial' boundary on your drawing that sets the area of the grid, and when turned on, limits you to drawing in the grid area.
Linetype	All objects are drawn with a particular linetype. Examples would be solid, center, dashed, etc.
Model space	The drawing space where you 'model' the objects.
Modify	A generic term used for changing your objects
Object	Any item that is in the AutoCAD database. Also known as an entity.
Origin	The (0,0) point of your current coordinate system.
Ortho mode	This is a drawing mode that allows you to draw only perpendicular lines. It is toggled on and off by pressing the F8 key.

Orthographic Projection	A standard drawing method that shows 2 or more views of the same part.
Osnap - Object Snap	This is a method of 'snapping' to certain, precise points on an object.
Pan	To move around drawing by dragging the drawing area around your screen.
Panel	A grouping of commands on the ribbon
Path	The specific folder where AutoCAD looks for, or saves files.
Pick	To select an object by 'left-clicking' on it.
Plot	Also known as print. To make a hard copy of your drawing.
Polar coordinates	A way of inputting points based on distance and angle.
Property	Any specific characteristic of an object such as layer, scale, linetype, start point, etc.
Ribbon	The Ribbon runs across the top of the drawing space and contains panel - each panel has a group of associated tool. Switch to different panels by clicking on the tabs at the top of the ribbon.
Relative coordinates	A way of inputting points based on a starting point.
Section View	A drawing that represents a cross section of a part or assembly.
Selection set	The current group of objects selected for modifying.
Snap	This is a drawing mode that allows you to snap your cursor to precise points laid out in a grid pattern. Toggle with the F9 key.
Styles	Formatting that defines the look of text, dimensions, etc.
Units	The basic drawing unit set for you drawing. For example, you can use inches or millimeters depending on your needs. You can also set the precision you want displayed, such nearest 1/4", 1/2" 1/64", etc.
User coordinate system (UCS)	Modifications made to the World Coordinate System (WCS) results in a User Coordinate System (UCS)
View	A particular area of your drawing.
Viewport	A separate 'window' on your drawing. You may have more than one viewport visible to see different areas of your drawing at the same time.
Wizard	An easy step-by-step instruction set to help you set-up certain aspects of your drawing.
World	This is the common X-Y coordinate system that is the default. If it is

Coordinate System (WCS)	modified, it becomes a User coordinate System (UCS)
Zoom	To view either a smaller section of your drawing (zoom in) or a larger section (zoom out)

Topics covered in this Lesson:

AutoCAD allows you to have access to a large number of commands. A general rule is that you will use 20% of the commands 80% of the time. I will start by introducing you to the most common drawing commands. When you combine these with the basic modify commands, you will be able to make elaborate drawings quite quickly. In other words, most of the commands you will use while using AutoCAD are taught in Level 1.

The important thing to remember is that AutoCAD will expect you give it information in a very particular order. The most frustrating thing when you begin using this program is that you will try to do something, but AutoCAD will 'not work'. In most cases, it means that you are trying to input information at the wrong time. This is why it is **veryimportant** to be in the habit of looking at the command line.

The command line tells you what information AutoCAD requires to continue.

Your first drawing assignment will be to use the drawing commands in conjunction with the co-ordinate system defined in Lesson 1-1. This is a basic assignment, but it is very important to understand how to give the program accurate information. You will use the following commands:

Command	Keystroke	Icon	Location	Result
Line	Line / L		Home >Line	Draw a straight line segment from one point to the next
Circle	Circle / C		Home>Circle > Center, Radius	Draws a circle based on a center point and radius.
Erase	Erase / E		Modify >Erase	Erases an object.
Print	Print / Plot CTRL+P		Quick Access Toolbar > Print	Enables the Print/Plot Configuration Dialog Box
Undo	U / CTRL+Z		Quick Access	Undoes the last command.

Toolbar>Undo

Assignment #1 - Drawing lines to exact points

Duplicate the drawing called Assign #1.

You will not have to worry about the title block or text, or dimensioning.

Make sure you are comfortable with the co-ordinate system as explained in Lesson 1-1. When you are finished this assignment, check the printed drawing with a scale ruler. All lines should measure up exactly if all went well.

Steps:

Start AutoCAD and a new drawing by pressing the Application Button (top left corner) and pressing the new button to reveal the flyout. Once you see the flyout, click on Drawing.

You will see a dialog box open that asks you to select a template drawing to use (as shown below):

Select the "acad.dwt" template file and press the **Open** button to continue to the drawing screen.

Once there, type in Z <ENTER> E <ENTER> this will zoom into to the extents of the drawing area and make it easier to see what you are drawing (NOTE: nothing will appear to happen).

For all lessons on this level, make sure that you **do not have Dynamic Input turned on**.

You can check this on the status bar. Make sure (the DYN button) isn't depressed. Your status bar buttons (bottom of the screen) should like the image above with 3 icon pressed (in blue).

Start the **LINE** command (as explained in the table above) 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**. Your command history (F2 key) should look like this:

Command: L LINE Specify first point: 1,2

Specify next point or [Undo]: 3,2

Specify next point or [Undo]: 3,4

Specify next point or [Close/Undo]: 1,4

Specify next point or [Close/Undo]: 1,2

Specify next point or [Close/Undo]:<ENTER>

If you make a mistake, you can use the **undo** icon, press **U** or press CTRL+Z.

You can also use the **ERASE** command to get rid of lines you don't want.

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 based on the last point you entered). Next type **@0,2** then **@-2,0** then **@0,-2** to finish the box. (Remember to press enter after each point.)

Now erase the last box you just drew. Start the **ERASE** command and then select the lines you want to erase. Then press <enter>. Now redraw the box for more practice!

> BREAK TIME : Take a moment to think about what you just did. You first drew lines based on ABSOLUTE points on the screen. Then you drew lines based on points RELATIVE to other points and objects. the difference is critical.

Draw a third 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^{o}, then another at 135^{o} and so on. Do the angles you entered make sense to you? If not, review it.

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).

Topics covered in this Lesson:

Rectangle | Multi Line | Extend | Offset | Trim | Introduction to Object Snaps

The previous lesson dealt with drawing commands. This lesson will introduce some common modifying commands. In AutoCAD, you may actually use modifying commands more often than drawing commands. Now that you know the basics, here's some more commands to add to your collection. Three commands, Trim, Extend and Offset are used standard AutoCAD work.

Command	Keystroke	Icon	Location	Result
Rectangle	**RECTANGLE / REC**		H ome> Draw > Rectangle	Draws a rectangle after you enter one corner and then the second.
Trim	**TRIM / TR**		Home > Modify > Trim	Trims objects to a selected cutting edge.
Extend	**EXTEND / EX**		Home > Modify > Extend	Extends objects to a selected boundary edge.
Offset	**OFFSET / O**		Home > 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.

Assignment #2 - Modifying Commands

The purpose of this assignment is to use the commands learned in the previous lesson and learn some new ones.

Once again, do not worry about title blocks, text or dimensions, draw only what is in yellow.

Start AutoCAD and begin the the drawing by opening up the template file like you did in Lesson 1-2.

Draw a **LINE** from **1,2** to **3,2** to **3,4** to **1,4** (***Remember to watch the command line as you do this.)** For the last line's endpoint , you can either type in **1,2** or **C** to close the line back to the first point you entered. These are **absolute coordinates.** Make sure you understand what the points your just entered represent.

Draw the next square using the **RECTANGLE** command. A rectangle is created by

specifying 2 points to represent the opposite corners. Enter the first point as 4.5,2 and then make the opposite corner 2 inches over and 2 inches up @2,2 using **relative coordinates.** This is much faster and also makes the square **one object** and not 4 separate lines.

ERASE the rectangle. You will see that all of it is gone with **one pick**. Redraw it and continue.

For the 3rd square, draw a 1.5 x 1.5 unit square using any of the methods you know. The bottom left corner must be a 8,2.

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 [Fence/Crossing/Project/Edge/eRase/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 you through most commands.

Draw a LINE from **4,6.5** to **6,6.5** Draw another line from **5,5** to **5,6** What you want to do now is **extend** the vertical line up to the meet horizontal line. Start the **EXTEND** command. AutoCAD asks for a boundary edge; select the horizontal line press <ENTER>. It then asks for an object to extend; select somewhere in the top half of the vertical line. Press <ENTER> to end the command. Your command line history should match what is shown below.

Command: EX **<enter>** EXTEND
Current settings: Projection=UCS, Edge=None
Select boundary edges ...
Select objects: **<Select the horizontal line>** 1 found
Select objects: **<enter>**
Select object to extend or shift-select to trim or

[Fence/Crossing/Project/Edge/Undo]:**<Select the top half of the vertical line>**
Select object to extend or shift-select to trim or [Project/Edge/Undo]: **<enter>**

Draw a CIRCLE with a center point of **7.5,5.5** with a radius of **.5** Now you will use to offset command to make another circle 1/4" larger. Start the **OFFSET** command (watch the command line) and enter **.125** as the offset distance (1/2 of 1/4"). Now select the circle and pick anywhere **outside** the circle. Press <ENTER> to end the command.

BREAK TIME : You won't believe me now, but I use the Offset A LOT when I am drafting. As you watch the videos in the next level, you'll see what I mean. I would be lost without my Offset. You will be too, so learn how to use it and be quick with it. You'll get more practice.

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.

ICON	SETTING		ICON	SETTING
	Endpoint			Insertion Point
	Midpoint			Perpendicular
	Center			Tangent
	Node			Nearest
	Quadrant			Apparent Intersection
	Intersection			Parallel
	Extension		M2P	Midpoint between 2 points

You may select whichever points you want to 'snap' on an object. Here is a list of your options. Followed by the command entry to invoke the needed Osnap.

Endpoint - snaps to either the beginning or the end of an object such as a line - **END**

Midpoint - snaps to the exact middle of a line or an arc - **MID**

Center - snaps to the center-point of a circle or arc - **CEN**

Node - snaps to 'nodes' (not covered in this course) - **NOD**

Quadrant - snaps to any of the four quadrants of a circle - **QUA**

Intersection - snaps to the point where two object cross - **INT**

Extension - Snaps to the phantom extension of an arc or line - **EXT**

Insertion - snaps to the insertion point of an object (such as a block or text) - **INS**

Perpendicular - will snap so that the result is perpendicular to line selected - **PER**

Tangent - snaps to create a line tangent to a circle or arc - **TAN**

Nearest - will find the closest point an object and snap to that point - **NEA**

Parallel -Snaps parallel to a specified line - **PAR**

Topics covered in this Lesson:

Creating Selection Sets | Changing your options

By now you have probably seen a colored rectangle appear on your screen when you left -click and the move the crosshairs around. You'll learn all about these in this lesson. AutoCAD uses what's called a **selection set** to allow you to group objects together and then modify them. For example, if you want to erase several lines, you could press E <ENTER>, pick on the line, press <ENTER> again and repeat until you're done. Another way would be to press E <ENTER>, and then pick the lines one by one until they are all selected, and press <ENTER>. By selecting a group of objects, you have created a selection set. Whenever you want to modify an object, and are asked by AutoCAD to "**select objects**" you can create a selection set and then apply the command.

REMEMBER TO PRESS **ENTER** WHEN YOU ARE FINISHED SELECTING THE OBJECTS

There are also a few other ways to create a selection set. The most common way is to use a window.

There are two very different types of windows you can use.

One is a *'crossing window'* and the other is a *box*. If you create the window from **right to left**, you make a crossing window. This means that any object that crosses the border or is surrounded by the window is added to the selection set. This is shown as a green rectangle with a dotted outline on the screen. If you create the window from **left to right**,

you create a box. Using this method you'll add only the items that are completely within the box. This is shown as a blue box on the screen. **The difference is very important.**

To summarize:

DIRECTION	CALLED	DISPLAY	EFFECT
RIGHT TO LEFT	CROSSING SELECTION	GREEN (with a dotted outline)	SECLECTS ANY OBJECT THAT EITHER CROSSES THE BOUNDARY OR IS INSIDE IT
LEFT TO RIGHT	WINDOW SELCTION	BLUE	SELECTS ON OBJECTS THAT ARE COMPLETELY WITHIN THE BOX

There are other ways to select objects and here a few of the more common ways. These can also be used in combination.

REMOVE - by typing **R** when asked to select objects, AutoCAD will select change to allow you to Remove objects from the selection set. You can also remove individual objects by pressing the **Shift Key** when selecting (known as a 'shift-select')

LAST - by typing **L** when asked to select objects, AutoCAD will select the last object that you created. This is handy if you create something and want to move or modify it right away.

PREVIOUS - by typing **P** when asked to select objects, AutoCAD will select the previous selection set. This is used if you select a few objects, modify them and then want to modify them again.

FENCE - by typing **F** when asked to select objects, AutoCAD allows you to draw a series of lines (called a fence) to select objects. This is convenient if you want to select a group of lines to be trimmed to a single cutting edge.

CROSSING POLYGON - typing **CP** when you are selecting objects give the ability to create a crossing polygon for object selection. This is similar to a crossing box (drawn like a Fence), but you can pick points on the screen to create a polygon. By default, this is a crossing polygon, therefore any objects that cross the polygon will be added to the selection set.

CROSSING WINDOW - typing **WP** (window polygon) when you are selecting objects give the ability to create a crossing window for object selection. This is similar to a crossing box, but you can pick points on the screen to create a polygon. This is a crossing window, therefore any objects that are completely within the polygon will be added to the selection set. Newer versions of AutoCAD allow you to use your cursor to create a crossing window without typing in the 'CW'.

CYCLING - When are ready to select objects, hold down the Shift key on your keyboard and then press the Space Bar when your mouse is on top of overlapping objects. Repeatedly press the Space Bar until the one object you want to modify is highlighted. Then pick with the mouse. This is a good approach if you have many objects in a small area.

To deselect everything you have selected, you just press the **escape** key on your keyboard or right-click and choose "Deselect All" from the menu.

TIP: Zoom in when selecting in tight spaces. Make sure you are 100% sure you are selecting the correct object. Imagine the difference of offsetting an interior wall instead of the exterior wall - the resulting object would be off by over 6 inches! Get used to zooming in, then zooming out quickly.

You need to quickly select objects in AutoCAD to be a good operator.

For practice, open one of your previous drawings and try selecting objects using all of the methods described above. Notice that as you move your cursor over an object the object will highlight.

View the video about selecting objects in AutoCAD (Part 1).

View the video about selecting objects in AutoCAD (Part 2).

Changing your selection options

Many people are happy with the default AutoCAD settings. Sometimes you might want to change your settings for things like your grip or pickbox size. This is totally personal and it's your call. I'll show how to change these, but remember where the settings are in case you want to change them back.

Type in OP for Options and look for the Selection tab. You'll see a dialog box that looks like the one below.

As you can see, you can customize a few things. Think about having a drawing that has a lot of blue lines in it. If you want, you can change your grip color so that they are clearer. Some people like a larger pickbox size, but I prefer a smaller box that is more precise. Another option that is used a lot is "Enable grips within blocks" - you'll learn about grips and blocks in later lessons.

Topics covered in this Lesson:

Move, Copy, Stretch, Mirror

In this assignment you will be adding some more common commands to your

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

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

This is a short assignment to show you how these commands are used. You will recreate the drawing called Assignment #4.

Start up AutoCAD and set up your drawing as you have in the previous assignments.

Turn on your Endpoint Osnap.

This time draw the border first. Draw a 10" wide by 7" high rectangular border using any method. The bottom left corner must be at **0,0**

Draw a 2" wide by 3" high rectangle using the
RECTANGLE command. The bottom left corner must be at **0,0**

Notice that the small rectangle and the border are overlapping each other at the bottom left of your drawing. What you want to do is move the small rectangle over 1" and up 1" so that it is away from the border.

To do this, start the MOVE command by typing in either **M** or **MOVE**<ENTER>. Select all the lines of the rectangle using one of the selection methods described earlier. Press <ENTER>. Now AutoCAD asks for a "**base point or displacement**". What it is needing is a reference point. Click on the bottom left corner of the rectangle. AutoCAD now asks for a '**second point of displacement**'. What it needs to know now is how far you want to move it. This is a great time to use relative coordinates. In this case, you want to move it 1" over and 1" up. So type **@1,1**<ENTER> to achieve this. The rectangle will automatically move to its new location.

Now you want to **copy** this rectangle 3" over to the right. The copy command is very

similar to the move command. (The only difference is that the copy command leaves an original behind.)

Start the **COPY** command. You will be asked to select objects. Select the rectangle you just moved. AutoCAD now needs the "**base point or displacement**" just like in the move command. Once again, select the bottom left corner of the rectangle. Once you've done this, you need to tell AutoCAD what the second point of displacement is. Since you want to move the rectangle over 3" to the right, type in **@3,0**<ENTER> The rectangle has now been copied 3" over.

> BREAK TIME : Copy and move are 2 commands that you use a lot with AutoCAD. Both commands work the same way in terms of the steps you take. Learn to use these commands well - you'll need them. Honestly. Being quick with these commands makes you a much better drafter.

But the rectangle is not as tall as the one in the sample drawing, the sample drawing's rectangle is 1" taller. To modify this, you'll use the **stretch** command.

Start the **STRETCH** command by typing **S**<ENTER>. AutoCAD now makes you select objects by using a crossing window or crossing polygon. You're going to use a crossing selection. Remember from the Lesson 1-5 that you make a **crossing selection** by creating it from the right to left. Left-click just a bit above and to the right of the top right hand corner of the new rectangle (P1). Move your crosshairs down and to the left until your (dotted) crossing window covers the top half of the rectangle completely and then left click again (P2). You'll see that the objects are highlighted now. Press <ENTER> to accept this. Next you're asked for that now familiar base point. Pick on the top left corner of the rectangle. Now give AutoCAD the second point of displacement. In this case, you want to stretch the rectangle 1" up, so type **@0,1**<ENTER> to do this. The rectangle is now 1" taller.

The goal when selecting objects to stretch is to draw the window over the vertices or points that you want to stretch. If you miss a corner, you will change the shape of the rectangle. So always be aware of which points need to be stretched. In this example, it was pretty obvious, in more complex drawings - it won't be.

Next you want to draw the polygon on the right side. To do this, you will draw the three lines on the left side first and then mirror those lines over to the right side. Draw the 3 lines any way you like (hint: use absolute coordinates.).

Once they are drawn, begin the **MIRROR** command. Select the three lines (press <ENTER>) Now you are asked for the first point of the mirror line. With your endpoint Osnap turned on, pick the end of the line at **8,2**. Now you are asked for the second point. Select the point on the line at **8,5**. Once you've done this, AutoCAD wants to know if you want to delete the old objects. In this case you don't, so accept the default by pressing <ENTER>. The mirror line will be half-way between the object the you are mirroring and where you want it to be. Figuring out where the mirror line is the toughest part of this command.

The assignment is now complete. Review what you have done and practice on these commands. Save and print your assignment. Copy and Move are very common commands. For example, you might create a ceiling light, then have to move it into place, and then copy it to other rooms. Fortunately, these commands work very similar, and once you master one, you have mastered both.

View the video for this assignment.

Extra Practice: Copy this drawing using the commands you have learned in lessons up to this point - **extra_007.gif**

View the video for Extra_007.

Topics covered in this Lesson:

Rotate, Fillet, Chamfer, Array

Now it's time to learn a few more commands. Like all of the commands learnt so far, these too will be ones that you will use regularly.

Here are the commands that you will be learning in this lesson.

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

Once again you will recreate a drawing. This one is called Assignment #5.

Click here to see the GIF format file.
Click here for the DWG file.

Follow the steps shown carefully. As these commands require a little more input, make

sure that you **keep an eye on the command line**. You will be asked to provide information throughout the commands.

Start up AutoCAD and load the acad.dwt template like you have for the other lessons.

Start by drawing a horizontal 10" X 7" border with the bottom left corner at 0,0

Draw a rectangle 1" wide by 3" tall with the bottom left corner at .75,.75

You are now going to rotate this rectangle 90° clockwise.

Start the ROTATE command. AutoCAD asks you to select objects. Select all parts of the rectangle and press <ENTER>. Now you must indicate a '**base point**'. Think of this as a pivot point around which the rectangle will rotate. In this example, you want to select the bottom right corner (*remember to use your Osnap*). Once you've selected the base point, the command line shows rotation angle or [Reference]: This means that 'Rotation angle' is the default, so type in the angle you want to rotate the object. Think about how AutoCAD measures angles. Looking at your rectangle and the one on the assignment sheet, you'll see that you want to rotate the rectangle clockwise or: -90 degrees. Enter that number and press <ENTER>.

Command: RO**<ENTER>**
Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0
Select objects: <Select the Rectangle>1 found
Select objects: <ENTER>
Specify base point:<PICK BOTTOM RIGHT CORNER OF THE RECTANGLE>
Specify rotation angle or [Reference]:-90 <ENTER>

The rectangle is now been rotated -90 degrees from its original position. Picking different base points will give you different results. **Undo** the last command. Try a few different combinations of base points and angles to see what results you get. When you are done practicing, get the rectangle back to the position that it needs to be in.

Make a **COPY** of the rectangle 2" above the first one (remember your relative coordinates).

Now you're going to modify the second rectangle so that it has rounded corners. Start the FILLET command. Look at the command line. It will look something like this:

Command: F <ENTER> FILLET
Current settings: Mode = TRIM, Radius = 0.0000
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:

AutoCAD first shows you what the current fillet radius is (0.0000). This will be the last value that was used. Once it's changed, it will keep the new value in memory. The next line shows you what options you have in this command. Remember that the **C**apitol of

each option selects that particular option. What you want to do is change the fillet radius to 3/8" (or .375). To do this you have to type R <ENTER>. When you type this AutoCAD will give the chance to enter a new fillet radius. At this point enter .375 and press <ENTER>.

The fillet radius is now .375 (which is what you want). The default option is **Select first object.** Select the left side of the top rectangle (yes, the whole rectangle will highlight if you drew it as a rectangle). AutoCAD now asks you to select second object. Select the top line and AutoCAD will make a smooth round corner with a radius of .375. AutoCAD automatically ends the command at this point.

Restart the FILLET command and do this to the remaining corners so that you have an object similar to the example.

BREAK TIME : The Fillet command is commonly used with a Zero radius. This can sometimes be much quicker than trimming two lines that meet at endpoints. To practice this, fillet this lines you made with round corners using a Zero radius.

Copy the first rectangle to a point 4-1/2" above. Now you will use the chamfer command to give this rectangle sharp, angled corners.

Start the CHAMFER command.

This is very similar to the fillet command. You have several options available. Want you want is an even 45 degree angle 3/8" in from the corner. Like the fillet command, you first have to tell AutoCAD what distance you want. To do this, type D to select the Distance option. The command line now looks like this:

Specify first chamfer distance <0.5000>: .375<ENTER>as your first distance.) The command line now asks for the second distance. AutoCAD will automatically change the default of the second distance to match the distance you entered for the first.

Specify second chamfer distance <0.3750>: (Press <ENTER> to accept this)

You will then be asked to Select first line. The chamfer command works just like the fillet command. Select the line on the left of the top rectangle. (Don't worry if the entire rectangle highlights.) When prompted to Select second line: select the top line. You will now have a perfect sharp corner at a 45 degree angle 3/8" in from the corner. Do this to the rest of the corners.

Look at the command line. It should look like this:

Command: CHA<ENTER>CHAMFER
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Select first line or [Polyline/Distance/Angle/Trim/Method]: D <ENTER>
Specify first chamfer distance <0.5000>: .375 <ENTER>

<0.3750>: <ENTER>
Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: <select one side of the rectangle>

Now look at the assignment sheet and notice the group of six rectangles on the bottom right. You could draw each one individually, but AutoCAD has a command that will allow you to draw one, and it will make the others.

Create a rectangle that is 1/2" square with the bottom left corner at 6,1.5 (absolute points).

Start the ARRAY command. Look at the dialog box shown below (if you are using AutoCAD 2012, please check these videos for the updated command):

When confronting a new dialog box, I recommend that you look for what is needed from the TOP DOWN to the bottom. This is a great example.

1. Choose the radio button for "Rectangular Array". This will array the object in a row/column arrangement.
2. Next select the object you want to array, by picking on the button in the top right corner. (Press enter when done)
3. Enter the number of rows (going across the page) and column (running up and down the page).
4. Enter the Row offset. This is this from the bottom left of the original rectangle, to the bottom left of where the first copy will go.
5. Enter the Column offset
6. Pick the Preview button to see the array before committing.

   Pick or press Esc to return to dialog or <Right-click to accept array>:

If the array is correct (check the sample drawing), press right click. If you need to change anything, press the ESC button, make your changes in the dialog box and preview again.

Now you are going to use the ARRAY (polar) command to create the shape in the top right corner of the assignment.

Start by making a CIRCLE with a center point of 7.5,5.5 and a diameter of 1.5 Next make a LINE from the center of the circle going 1" to the right (remember your relative input and Osnaps).

Start the ARRAY command. When asked to select objects, pick the line you just drew.

Examine the dialog box above. Remember to start from the TOP. In this case, you have to select your objects and select a Center Point for the array. (Select the center of the circle.)

NOTE: Sometimes the Array command can be quicker than the offset command. Think of creating lines for a ceiling grid. You 'could' offset 30 lines one at a time, or you 'should' use the array command to create all 30 lines at once.

Save and print your drawing.

View the video for this assignment .

So far in seven lessons, you have learned many of the common commands in AutoCAD. It may not seem like a lot, but the idea is to become fluent in them. It should be second nature to run these commands, as these are the ones you will be using most often. With practice, you won't need the command line to help you along. Think of the first time you drove a car with a manual transmission. After time, it got smoother, didn't it. I can't stress enough how important practice is during these early lessons.

Extra Practice: Copy this drawing - **extra_009.gif**

View the video for extra_009.

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Extra Practice: Copy this drawing using 2 methods. First create the lines using the offset command, then create the lines using the Array command. Compare which method should be easier, and when you might use one over the other - extra_012.gif

View the video for extra_012.

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CONTINUE TO THE NEXT LESSON - 1-8 >

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View the video for extra_012.

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Topics covered in this Lesson:

Direct Distance Entry | Polar Tracking | Object Snap Tracking | Dynamic Input

Direct Distance Entry

As mentioned earlier in the lessons, there are many ways to do things in AutoCAD. To enter distances, you have been shown Absolute, Relative and Polar Coordinates. I've spent the previous 8 lessons showing you foolproof ways to enter data (the hard way). Now you will be shown four more ways to tell AutoCAD where the locate the point you are drawing to (the easy way).

DO NOT FORGET THE BASICS OF MANUAL ENTRY

Direct Distance Entry (or DDE) is a way of bypassing the usual ways of entering in coordinates. and just entering the distance.

The method is quite easy.

Make sure that you have set Ortho (locking your input to vertical and horizontal) by pressing the F8 button and checking to see that the "Ortho" button is depressed (blue)

on the status bar like this:
Your **F8** key will toggle Ortho on and off.

Lets say you want to draw a line from one point directly to the right 10 units.

Start the Line command and click on the screen anywhere, then more your cursor to the right of that point. If Ortho is turned on, the line should only point directly to the right and not at an angle. Now type in 10<ENTER>. Press <ENTER> again to exit the command. You now have line that is 10 units long.

That is a very easy way to draw line segments. If Ortho is not set, your drawing will get messed up very quickly and will not be easy to fix. I have seen too many students take this easy route and destroy their projects, fail the class and become homeless. So don't do this.

Polar Tracking

Now wouldn't it be cool to draw angled lines (like the short ones in the above image)? Well you can, but first you have to make change in your settings.

Type in DSETTINGS and you get the Drafting Settings dialog box, go to the "Polar Tracking" tab. :

Then make sure that Polar Tracking is On (press your F10 key to toggle it on and off) and then select the increment angle. I recommend using polar settings with 45° increments unless you specifically need something else.

In the example above, I made 4 lines 1 unit long using Direct Distance Entry (DDE). See if you can duplicate this on the left end of the 10 unit line you just drew. The process is the same as you did for the DDE of the previous line. Make sure your increment angle is 30° and draw a line 1 unit long using DDE..

You can not have Ortho and Polar Tracking on at the same time. As you start to draw more, you will see that these two features are great time savers. There will be times, though, when you have to use absolute and relative coordinate entry (especially in 3D).

Refer back to the assignment in Lesson 1-2 and draw using the methods shown in this lesson.

Now I'm going to repeat this one last time:

Make sure you have either Ortho or Polar tracking on and don't forget how to manually enter points!

View the video for a sample of DDE.

Object Snap Tracking

Now you've just seen how you can find distances from points and measure accurately, but what if you want to find specific points based on previously drawn objects? You can use Osnaps, but they don't always find the point you need.

Consider that you want to draw a circle in the middle of a rectangle like the example below:

In the olden days, you would have drawn a line from opposite corners of the rectangle to find the middle, then snapped to the midpoint of that line for the circle's center point, drawn the circle and then erased the reference line.

Now you can use **object snap tracking**. This is a way of finding reference points and drawing from them. To do this exercise, make sure that your Midpoint Osnap is on.

Draw a RECTANGLE from 0,0 to 4,3

Make sure that the Object Snap Tracking and Osnap button are depressed.

Start the CIRCLE command.

Move your cursor over the midpoint of the bottom line of the rectangle. You should see a light dotted line project vertically through your cursor. Now more your cursor to the midpoint of the right vertical line of the rectangle. As you move your cursor towards the center of the rectangle, you should see the 2 dotted lines cross. It should look like this:

Once you see both lines, you can pick and the center point will be exact center of the rectangle. Give your circle a radius of 1".

This is a very simple example, but as you start drawing more complex shapes, this will be a handy to tool to master, and a great time saver. Object tracking will work with any Osnap that is invoked. But always check to make sure that you are picking the point you want.

Now the fun begins! You can combine Object Snap Tracking with Direct Distance Entry. What if you want to draw a line that starts 10 units above the top right corner of a rectangle? You can just use these techniques in tandem to move up from the corner and and type in 10 to get the starting point.

This technique is a great time saver. In older versions, you have needed to offset the rectangle, draw the line and then erase the rectangle. Try this out on your own.

View the video to see object tracking in action.

Dynamic Input

Now it's time to put all these concepts together and combine them with on-screen help that's called 'Dynamic Input'. In older versions of AutoCAD, users relied solely on the command for information. Later versions of AutoCAD added Dynamic Input to put a user's keystrokes in the drawing area near their cursor.

Turn on Dynamic Input by clicking on the status bar (toggle with F12):

Now when you draw, you will see all kinds of information on the screen.

In this example, I have drawn a line starting at the midpoint of the bottom line going up 20.5 units (which I manually entered) at 135°. Once I enter the distance, I can press the Tab key to switch over to the Degrees and enter a value there. I could also have typed @20.5<135 using relative coordinates. By default you are in the first field (length) - press the TAB button on your keyboard to be able to enter the angle.

If you are new to AutoCAD, I recommend turning this feature off as it takes your eyes off the command line (which will prompt you for information).

To review, AutoCAD provides some nice tools to enhance your productivity. Use them, master them, but don't forget the basic of inputting points. Especially when you do 3D work, manual entry becomes valuable.

Topics covered in this Lesson:

Object Properties

Now that you have learned how to draw objects and modify them using special commands, you need to know more ways to change the properties of an object. For instance, you may have made a spelling mistake in your text. You could erase it and re-create it, or you could double-click on it and just correct the error. Maybe you drew something on the wrong layer? This process is an extremely important tool to help you with your CAD work.

DON'T REDRAW AN OBJECT - CHANGE ITS PROPERTIES

There are a few ways to change, or view an objects properties. First is by using the PROPERTIES command.

Command	Keystroke	Icon	Location	Result
Properties	**PROPERTIES** /	No	Home	Displays the

	PR / CTRL+1	Icon	>Properties	properties of the object in the Properties Palette
Match Properties	MATCHPROP / MA PAINTER		Home > Clipboard >Match Properties	Copies the properties from one object to another

The method is quite easy and also the most complete way of viewing properties.

Draw a line from (any point) to (any point) and press <ENTER>. This is only time I will not tell you to use specific coordinates. **:)**

Now select the object by clicking on it and pick the properties icon. Once you do this, you should see the properties palette appear showing everything that makes that line what it is.

What you see here is the complete palette. Before you look at the properties, take a moment to examine the palette itself. These are new to later versions of AutoCAD. Since it takes up a large amount of the screen, you have the option to either close it completely ('X' at the top) or collapse or hide the palette (below the X) for future reference. You can also dock a palette to the side of the screen. Try the buttons out and then continue reading.

Now look at the properties as displayed in the palette. At the top are the 'General' properties - these are common to all AutoCAD objects (they all have them). You will see that one endpoint of the line is shown as the "Start X" and "Start Y" points (X,Y point). If you want to change the location of the X coordinate for the Start point, pick in the text area, type a number and press (ENTER). You also have the option of picking new X coordinate with the small icon shown above.

Have good look at the palette. Erase the first line, draw another line and then display the new line's properties. You should see different numbers.

Now erase the line and draw a circle. Display the properties and you will see different fields. You will see that the the Start X point has been changed to "Center X". The properties displayed are specific to the object you have selected.

Now draw a line next to the circle. Select both by clicking on one, and then the other. Now display the properties by selecting the icon. You will now see a much short list of properties. This short list will include properties that are common to both objects, such as layer, color, etc.

Now try something different. Erase everything and close the Properties palette. Draw a circle and then select it. Right click on the screen and choose the Properties option at

the bottom of the list as shown below:

You'll see the same palette appear - in the same space where you closed it. You also have the choice of selecting Quick Properties the same way. If you do, this is the smaller palette that you will see (handy on smaller monitors).

OTHER OBJECTS

By now you see how you can view and edit properties of objects you have drawn. Those the methods will work on any object. But what about other objects? Could there be easier ways to edit their properties? You bet!

Enter a line of text like you did in the Lesson 1-8. View the properties like you did in the previous examples. Note again, the various fields that in this case are unique to Text. Now try "Double-Clicking" on the text.

Now you can change the text and press <ENTER> (then <ENTER> to end the command). This is a much easier method to change the text's content, but none of the other properties. You could also have changed the text in the Properties palette, but this method is much quicker.

Changing Layers

Many times you will draw an object and find that is on the wrong layer. To change an object's layer, you can open the Properties palette (as shown above) or use this easy tip.

Create a new LAYER (as you did in Lesson 1-8) and call it **LINES**. Do not make it the current layer. Draw a line on the 0 layer.

Now select the object and then move your cursor up to the layer droplist. You'll see a list of all of your layers - pick on the **LINES** layer - then press **ESC**.

Now when you check the properties of the line, you'll see that it is on the **LINES** layer. This is one method you will be using a lot, so learn it and master it.

Match Properties

One of the easiest ways to set properties is to use the MATCH PROPERTIES command. This is used by starting the command (MA) and then selecting the source object (the one that has the desired properties) and then selecting the target object(s). Use can also select the source object first, then the icon and then the target object. The icon for

this command is in the clipboard tool panel. And no, I don't know why it's not in the Properties panel.

Conclusion

You now have the skills to change to properties of any object you draw. There are many ways of doing it, but you'll see that some methods are easier. So remember, once something is drawn, you can modify it as shown in the previous lessons in this level, or modify the properties. In only rare instances, is it easier to erase and redraw the object. As you learn more commands in the next lessons, try these methods to see what can be changed - and how.

Topics covered in this Lesson:

Zoom | Pan

So far the tutorials have dealt with drawing and modifying objects. This lesson will be a primer on how to move around in your drawing. With simple drawings like the assignments in Level 1, you didn't have much need for moving around or zooming in your drawing. The more complex your drawing is, the more you will need to master the power of zooming and panning. Mastering these techniques will enable you to be more productive in your drawings and life will be better.

EFFECTIVE ZOOMING CAN DRAMATICALLY INCREASE YOUR SPEED

One single command will give you the versatility to move around your drawing. This is the **ZOOM** command. Another useful command is **PAN**. These are both quicker than using the scroll bars on the side of the drawing area, unless you have a very short distance to move your drawing (and can make your scroll bars obsolete and thereby create more drawing space)..

Start the Zoom command by typing Z <ENTER>. When you do this, you will see the following options on the command line:

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

Remember that to invoke any option, just type the ***capital*** letter of your choice. (e.g.: type: **E**<ENTER> for "zoom extents". The default is "Realtime" which you invoke by pressing <ENTER>. One by one, here are the options available to you. These icons are available on the View ribbon under the Navigate Tool Panel

COMMAND OPTION	ICON	DESCRIPTION

Zoom Extents		This option will display all the graphics that are contained in the drawing (referred to as the *drawing extents*) with the largest image possible.
Zoom Window		This option (also a 'hidden' default) prompts the user to pick two corners of a box on the existing view in order to enlarge that area to fill the display.
Zoom Previous		This option restores the displayed view prior to the current one. For the purpose of this option, up to 10 views are saved so that the last ten views can be recalled. This option includes every time you use the scroll bar, which is one reason to **avoid the scroll bars** for panning a lot in your drawing.
Zoom Realtime		Zoom Realtime provides interactive zooming capability. Pressing <ENTER> (after entering zoom) on the command line automatically places you in Realtime mode. Hold the left mouse button down at the midpoint of the drawing and move the cursor vertically to the top (positive direction) of the window to zoom in up to 100% (2x magnification). Hold the left mouse button down at the midpoint of the drawing and move the cursor vertically to the bottom (negative direction) of the window to zoom out to100% (.5x magnification). *You cannot zoom out beyond the extents of the current view.* When you release the pick button, zooming stops. You can release the pick button, move the cursor to another location in the drawing, and then press the pick button again and continue zooming from that location. To exit Realtime Zoom mode, press <ENTER> or (ESC).
Zoom All		This option causes AutoCAD to display the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).
Zoom Dynamic		This is a very useful ZOOM option once it is understood. It permits very quick movement around the drawing. Once selected, this option redraws the graphics area of the screen and displays two rectangles. The larger box shows the extents of the current drawing. The smaller box shows the current view with an "X" in the middle. This moves with the mouse. This view box should be positioned so that its lower left corner is at the lower left corner of the view required. By pressing the left button on the mouse, the "X" is replaced by an "> " pointing to the right side of the view box. This allows you to change the magnification. As the mouse is moved, the view box shrinks and expands so that the size of the required view can be set. The left mouse button toggles between PAN

		"X" and ZOOM "> " mode so that fine adjustments can be achieved. When the view required has been selected, press <ENTER> or right click to cause AutoCAD to display it.
Zoom Scale		This is a 'hidden' default option. You do not have to type "S" to choose this option. It simply requires the entry of a number that represents a magnification factor. Note that the factor is applied to the entire drawing (as defined by the drawing's limits). Numbers less than 1 will reduce the displayed size of the drawing, while numbers greater than 1 will enlarge it. If "X" is inserted after the number (e.g.. 0.8x) then the factor is applied *to the current view*. If "XP" is inserted after the scale factor, then the view is scaled relative to paper space. This is useful for zooming a view within a paper space viewport to a specific scale, for example, "1/48XP" will produce a view of model space at a scale of ¼" = 1' relative to paper space.
ZoomCenter		This option requires two things: a point that is to be the center of the new display and a value to be its new height*in drawing units*. The existing height is the default for the new height to allow for panning across the drawing. If the new height value is followed by "X" (e.g.. 2x), then it is taken as a magnification factor relative to the current height. If followed by "XP", then it is taken as a scale factor relative to paper space and can be used for scaling the contents of paper space viewports.
Aerial View command: DSVIEWER	None	Aerial View is a zooming tool that displays a view of the drawing in a separate window so that you can quickly move to that area. If you keep the Aerial View window open as you work, you can zoom and pan without choosing a menu option or entering a command. You can change the view by creating a new view box in the Aerial View window. To zoom in to the drawing, make the view box smaller by left clicking a rectangle. To zoom out of the drawing, make the view box larger. As you zoom in or out of the drawing, a real-time view of the current zoom location is displayed in the graphics area. The screenshot shows how the view box looks. Right click in the box and you can move the box to where you want to zoom to.
Zoom Object		This option asks you to select an object or objects, then press <ENTER> and the screen will zoom to those objects only. This is great for when you want to work on object.

Zoom In		Clicking this icon will zoom in to the drawing by about 50%. This option is only available as an icon and cannot be invoked by the command line.
Zoom Out		Similar to 'Zoom In' - this icon will zoom out of your drawing and allow you to see about 50% more of your drawing space.
Mouse Scroll	No Icon	If you have a scrolling wheel on your mouse, you can use it to zoom in and out of your drawing. Scroll towards you to zoom out and away from you to zoom in. You have the option to change the amount of zoom per wheel click with the Zoom factor system variable. Keep in mind that you will zoom in and out using your mouse location as a 'center point'.
PAN		Panning allows you to quickly move around the drawing area at the same magnification you currently have set. Type in PAN (or P) <ENTER> and a hand will appear on the screen. Left click and hold to move around your drawing.

As you can see there are quite a few options. To begin with I would strongly recommend getting really good at these 3:

Use the **Zoom > Extents** whenever you want to see all objects.
Use the **Zoom > Window** option to 'close-in' on one area.
Use the **Zoom > Previous** option to return to where you were.

I generally use them in conjunction with each other. I'll do a zoom extents to see what state the drawing is at, then perform a Zoom Window to get to the area I need to work in, then do a Zoom Extents when I am done in that area. In between, I may need to use a combination of Zoom Window and Zoom Previous.

Additionally, using your mouse wheel to zoom can be very fast for moving in and out of an area - practice this technique as well.

The zoom command can also be invoked *transparently*. This means that you can start it up in the middle of a command. For example, if you are in the trim command and want to see a bit more of your drawing, just type **'Z** (note the apostrophe) at the command line and you can then zoom using any of the available options. Press <ENTER> to get back to your command.

Also, **right clicking** while in the zoom command gives you options. Try this and see which choices are available with this.

View the video for this lesson.

Exercise: Open one of your previous drawings, or one of the samples that came with your installation of AutoCAD and practice these techniques. You need to be good at this.

Review of Level 1: Everything in this Level will be used in your day to day CAD work. These are the basics that you will draw upon as you advance your skills. You have learned the how the coordinate system works and how to enter points so that AutoCAD knows where you want your objects to be. You were shown the most common drawing and modifying commands. You learned techniques for snapping to objects, other ways to input information and how to move around a drawing. For a new user, I cannot stress enough how mastering this level will make you a good drafter. It all comes down to accurate and fast input.

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Take the quiz for this lesson | Take the quiz for this Level

Topics covered in this Lesson:

Drawing Techniques

Putting your skills to use.

So far in this level you have been learning the basic 2D AutoCAD commands. What you learned in this level will be a very large part of what you use in your daily drafting. This tutorial isn't going to teach commands, but will instead show a common technique that is used a lot in 'Mechanical" drafting. It will also ask you to think about what you are drawing, and how it needs to be represented.

Mechanical drafting is a field within the drafting world. In simple terms, it is used to describe the methods for drafting and designing machines, assemblies and in a nut shell, the 'parts' that used in everything from a fork to a Formula 1 race car. It doesn't include anything that involves buildings and structures (Architectural & Structural drafting) or landscapes and roads (Civil drafting and engineering).

Now just because you are dreaming of becoming an Architect, it doesn't mean that you should skip this tutorial. In fact, you will likely need to use these concepts or read diagrams related to this subject. So read on!

What is Orthographic Projection?

If you look at the image below, you will see a drawing for a part. It shows the object with a top view, a front view and a side view. You'll also see an Isometric view that is

sometimes used to give a more visual look. This tutorial won't cover Isometric drafting as it is shown in Tutorial 3-2. Save your drawings that you do in this exercise for more practice in that lesson.

The reason that this method is used is that you can take a designed part, draw it, dimension it and then give all the needed information to the manufacturer.

In some cases only 2 views are needed, but for anything more than a simple part, 3 or more views are needed. Very complex parts will need 6 or more.

There are 2 methods of deciding what views are used and where they are placed in the drawing. I'll borrow some info from Wikipedia to show this:

First-angle projection (European Standards)

In **first-angle projection**, the object is conceptually located in quadrant **I**, i.e. it **floats above and bef** the viewing planes, the planes are **opaque**, and each view is **pushed** through the object onto the pla furthest from it. (Mnemonic: an "actor on a stage".) Extending to the 6-sided box, each view of the obj is projected in the direction (sense) of sight of the object, onto the (opaque) interior walls of the box; t is, each view of the object is drawn on the opposite side of the box. A two-dimensional representation the object is then created by "unfolding" the box, to view all of the **interior** walls. This produces two pla and four elevations. A simpler way to visualize this is to place the object on top of an upside-down bo Sliding the object down the right edge of the bowl reveals the right side view.

Image of object in box, with views of object projected in the direction of sight onto walls using first-angle projection.

Similar image showing the box unfolding from around the object.

Image showing orthographic views located relative to each other in accordance with first-angle projection.

Third-angle projection (USA Standards)

In **third-angle projection**, the object is conceptually located in quadrant III, i.e. it **lurks below and beh** the viewing planes, the planes are **transparent**, and each view is **pulled** onto the plane closest to (Mnemonic: a "shark in a tank", esp. that is sunken into the floor.) Using the 6-sided viewing box, ea view of the object is projected opposite to the direction (sense) of sight, onto the (transparent) exte walls of the box; that is, each view of the object is drawn on the same side of the box. The box is th unfolded to view all of its **exterior** walls. A simpler way to visualize this is to place the object in bottom of a bowl. Sliding the object up the right edge of the bowl reveals the right side view.

Here is the construction of third angle projections of the same object as above. Note that the individ views are the same, just arranged differently.

Image of object in box, with views of object projected in the direction of sight onto walls using first-angle projection.	Similar image showing the box unfolding from around the object.
Image showing orthographic views located relative to each other in accordance with third-angle projection.	

Ok - that was some fun theory - thank for reading it. One last bit of info before we get back to CAD stuff. Since there are 2 kinds of standards, how do you know which is which when you have a drawing in front of you? There is a standard symbol that is used in the title block to indicate which method was used.

This symbol shows a simple cone and displays the projection. Think about which symbol represents which method, then move your mouse over the images to see if you were correct.

Drawing 3 View Orthographic Projection

For these exercises, we'll start by looking at an Isometric drawing of an object and then draw the Front, Side and Top views using the dimensions we're given. In the workplace, you might find that you are given a part to measure and then draw, or you might be designing the part yourself.

Here's the part that we'll draw in this tutorial:

This is a very simple example to get you used to the concepts. You'll have more practice exercises at the bottom.

Ok, the first question that you'll ask yourself, is "Where do I start?". I recommend that you start where you have the most information. This will sometimes be the front or the top - it depends upon each drawing. In this case, I will start with the front and draw it.

You don't need to worry about dimensioning it at this point - wait until you have all of your views drawn. Ok, this should have been easy enough, so now you can start drawing the top view.

To draw the top, you need to 'project' the lines up. Draw lines up from the main points in your front view. Make sure you have your Osnaps on (include "Quadrant").

Now that you have the vertical lines, draw the horizontal lines. Make sure you leave enough room to draw the width (via OFFSET) and space between the views.

Now you almost have 2 views drawn. Trim the lines so that you are left with just the lines you need.

Stop and check to make sure that you didn't forget any lines. It's very easy to miss some.

Now it's time to jump ahead a little and take a side trip. Read Tutorials 4-3 to learn about Linetypes, because you will need them here. After reading the tutorial, return to your drawing and load the **Hidden** and **Center**linetypes. These are needed to add more information in your drawing.

If all went well, you should be able to load the linetypes and scale them (LTS with a value of 10 or 12) to fit with your drawing.

What you see above is the completed Top and Front views. Do you understand why there are "Hidden" lines? They are there to indicate that the hole (circle) is drilled right through the block. Where would the lines be in the hole was only drilled half way through? The center lines are used to show the the hole and the arc have the same center point. These are both common and standard CAD methods and you need to understand them.

One more view to draw. This will be the side/right view. Can you picture it yet?

To get started on the side view, you have to establish where it will be placed in the drawing. In this example and using the 3rd Angle projection, it will be shown to the right of the front view. For exact placement, you need to draw more projection lines.

What you see in the image above is that I established the top right corner of my front view by projecting 2 lines. Then I drew a 45 degree line up from the corner.

Now I can start projecting lines from my top view to create the side view. The line that is indicated by the 'Project Down' leader will be the left side of my Right Side View. I would project another from the other side of the top view and that would establish the width of the right side. Also note that by using this technique the top and side views are the same distance from the front view.

From there I just need to project to the right from my front view.

Almost there. Now you just need to trim up some lines and change the linetype for the hidden lines. Final goal is to draw this:

Once you have all of your lines, your center lines and your hidden lines, you are ready to dimension and add any notes that are needed.

Think about what commands you used in this tutorial. You used LINE, CIRCLE, OFFSET& TRIM. You also use Layers and then learned about Linetypes. What this shows is that you don't need to use a lot of commands, but it's your knowledge of how those commands works that makes your reputation as a CAD user.

Just one more thing I should mention. This method of drafting can be used in other ways as well.

To the right is a simple piece of duct work drawn in AutoCAD. It's a straight piece that has been cut at an angle so that it can be connected to another piece to form a bend.

Drawing in 3D isn't very hard when you know how. But if I sent this to the manufacturer, he wouldn't really know where to start. So I would have to send him a drawing of the tube rolled out.

This would give him a template to cut the shape and manufacture the part.

Think about what this shape would look like if it was flat. How would you draw it? How would get the curve correct? Do you think that this sounds like a job for Orthographic Projection?

The part is drawn in Magenta, and the projection lines are in Blue. What I needed to do was use the DIVIDE command to divide both the circle and the line that represents the circumference. Then it was a matter of projecting down and to the right from the circle, and up from the circumference line. This gave me a grid to use for the SPLINE to create the curve. The Auxiliary view was drawn with a few projection lines and an ELLIPSE.

Try this if you like and follow the command line for the commands you don't know. This might be tough, but take your time.

All the Best

AutoCAD 2019