### Topics covered in this Lesson:

<u>The X-Y Coordinate System | Angular Measurement | Entering Points in</u> <u>AutoCAD</u> <u>AutoCAD Basics | Terminology | The AutoCAD Screen</u>

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Welcome to the world of CAD - In this first tutorial you will be learning the very basics of AutoCAD. This 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 2016 and will work with older versions. By the end of this level you will have the skills to develop basic 2D drawings and print them out to scale.

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 (and not very exciting) but will cover important topics. Learn it, live it. Stick to it! You'll thank me later.

#### 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).



#### 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 **<u>relative</u>** 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. myCADsite.com

#### To review: ABSOLUTE POINTS are exact points on the drawing space. RELATIVE POINTS are relative to a POINT on the drawing space.

It's a simple system, but mastering it is the key to working with AutoCAD and is explained in more detail further below and in the next lesson. 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 faster they will learn.

#### Angular Measurement

AutoCAD measures angles in a particular way. 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 in the next lesson will show you these methods. 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 keyboard 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.)

### Even More AutoCAD Basics...

#### Subjects covered in this section: <u>The AutoCAD screen</u> - <u>Workspaces</u> - <u>Starting Commands</u> - <u>Terminology</u>

### 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 tools.

**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 icons 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 you 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. <u>Watch this line while learning.</u>

**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, you can 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 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 to AutoCAD 2016 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.

Line		
Line Create	es straight line segments	
🛃 LINE		
Press F1 for more help		

Another way is to Right-Click on the drawing space and choose "Recent Input" from the menu. This will give a list of the most recent commands that you have used.



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

#### VIDEO: Starting Commands in AutoCAD

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 (or press **CTRL+O** [number Zero]). To go back to the standard display, click again on the same icon.

	Clean Screen
& Annotation 🔻 🗂	1/2 🐯 🔽 🗖
_	9:25 PM

#### 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. Dimensions that are associated with specific points
Associated Dimensioning	
Backup file	extension
<u>Block</u>	A pre-drawn image you can insert in your drawing to save time and make your file size smaller. A display setting that gives you maximum drawing
Clean Screen	space.
Crosshairs	This is your cursor when it is in the drawing space. Your cursor will change depending on what
Cursor	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, etc are stored in this
Database	manner. AutoCAD uses a large number of dialog boxes to get information from you. You must know how to
Dialog box	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.
	The outer boundaries of the objects you have
Extents	drawn.

<b>.</b>	This is pattern of dots displayed on the screen to guide you. It can be toggled on and off by pressing
Grid	the F7 key. Small 'handles' on objects that allow for quick
<u>Grips</u>	editing.
	All objects are drawn on a layer. You can group objects (such as electrical) on a single layer and
Layer	organize your drawing. A space used for plotting your drawings (formerly
Layout Tabs <b>?</b>	called Paper Space).
	A setting to impose an 'artificial' boundary on your
	drawing that sets the area of the grid, and when
Limits (Grid)	turned on, limits you to drawing in the grid area. All objects are drawn with a particular linetype.
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
	Any item that is in the AutoCAD database. Also
Object	known as an entity.
<u>Origin</u>	The (0,0) point of your current coordinate system.
	This is a drawing mode that allows you to draw only
Ortho mode	perpendicular lines. It is toggled on and off by
	pressing the F8 key. A standard drawing method that shows 2 or more
Orthographic Projection	views of the same part.
<u>ennegrapme rejection</u>	This is a method of 'snapping' to certain, precise
<u> Osnap - Object Snap</u>	points on an object.
	To move around the drawing by dragging the
<u>Pan</u>	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.
	Also known as print. To make a hard copy of your
Plot	drawing.
	A way of inputting points based on distance and
Polar coordinates	angle.
_	Any specific characteristic of an object such as
Property	layer, scale, linetype, start point, etc.
	The Ribbon runs across the top of the drawing space and contains panels - each panel has a
	group of associated tools. Switch to different panels
Ribbon	by clicking on the tabs at the top of the ribbon.
Relative coordinates	A way of inputting points based on a starting point.
	A drawing that represents a cross section of a part
Section View	or assembly.
Selection set	The current group of objects selected for modifying.
	This is a drawing mode that allows you to snap
Snon	your cursor to precise points laid out in a grid pattern. Toggle with the F9 key.
Snap	

Styles	Formatting that defines the look of text, dimensions, etc.
Units <b>?</b>	The basic drawing unit set for your 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) result 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.
<u>World Coordinate System</u> (WCS)	This is the common X-Y coordinate system that is the default. If it is modified, it becomes a User Coordinate System (UCS)
<u>Zoom</u>	To view either a smaller section of your drawing (zoom in) or a larger section (zoom out)

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### Take the Lesson 1-1 Quiz

From which direction does AutoCAD start measuring angles 9 o'clock

6 o'clock

3 o'clock



#### For Further Thought:

As you start to work through these lessons, think about how you might use AutoCAD to produce drawings. Do you have a goal already? Are you in a specific field? Remember that AutoCAD is just a tool like a pencil. As you learn you might think a pencil would be easier, but AutoCAD is used for many good reasons. In later tutorials you'll find out how to edit a drawing rather than start over like you would if you were hand drafting.

This was a long lesson that might have confused some people, but make sure you understand the concepts before moving ahead. Just like building a house, your foundation in learning is very important.

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#### Topics covered in this Lesson:

#### Line | Circle | Erase | Print | Undo

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 **very important** 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 <u>Lesson 1-1</u>. 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	lcon	Ribbon Location	Result
LINE	Line / L	Line	<u>H</u> ome > <u>LI</u> ne	Draw a straight line segment from one point to the next
CIRCLE	Circle / C	Circle	<u>H</u> ome> <u>C</u> ircle > Center, <u>R</u> adius	Draws a circle based on a center point and radius.
ERASE	Erase / E	1	<u>H</u> ome > <u>E</u> rase	Erases an object.



#### **Assignment #1 - Drawing lines to exact points**

For your first assignment, you will duplicate the drawing called Assign #1. Don't worry - I'll walk you through it step by step.

Click <u>HERE</u> to see the finished drawing in .GIF format. Click <u>HERE</u> to download the AutoCAD file.

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 <u>Lesson-</u><u>1-1</u>. 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 NEW Icon at the top left of your AutoCAD window.



There are actually 3 ways to start a new drawing. The fastest is to press **CTRL+N**, but you can use the icon shown in the image above or type in the command **QNEW**. Any way you do this, you will see a dialog box open that asks you to select a template drawing to use (as shown below):

A Select template					X
Look <u>i</u> n:	🚺 Template 🗸 🗸	(= 🖳	(3)	🖳 <u>V</u> iews	▼ Tools ▼
	Name	Date	Preview		
<u>~</u>	PTWTemplates	3/29			
Autodesk 360	퉬 SheetSets	3/29			
e a	acad -Named Plot Styles.dwt	1/19			
	acad -ivamed Plot Styles3D.dwt	1/15			
History	acad.dwt	1/19			
B	acad3D.dwi	1/11			
	acadISO -Named Plot Styles.dwt	1/19			
Documents	acadISO -Named Plot Styles3D.dwt	1/19			
	त्व्या acadiso.dwt त्व्या acadiso3D.dwt	1/19			
	Tutorial-iArch.dwt	1/19			
Favorites	Tutorial-iMfg.dwt	1/19			
	Tutorial-mArch.dwt	1/19			
2 million 1	Tutorial-mMfg.dwt	1/19			
Desktop					
8					
шQ	< III	Þ			
FTP				_	
	File name: acad.dwt				Open -
	File name: acad.dwt			1	
Buzzsaw	Files of type: Drawing Template (*.dwt)			•	Cancel
	\				

If AutoCAD doesn't locate your templates automatically, they can be hard to find. Have a look in your:

C:\Users\[PROFILE]\AppData\Local\Autodesk\AutoCAD 2016 -

**English\R20.1\enu\Template** folder. You will find it there. Of course, you can also copy your templates to a folder that is easier to locate.

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). Also if you find that the grid distracts you, you can press **F7** to toggle it on or off.



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 active. (Dynamic Input affects relative coordinates in some versions of AutoCAD.) Also, leaving Dynamic Input turned off keeps the screen clearer and easier for you to concentrate on what you are drawing.



Your status bar buttons (bottom of the screen) should like the image above with the Dynamic Input button grey.

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 <u>absolute co-ordinates</u>. 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, **a** 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 <u>relative co-ordinates</u>. 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°, then another at 135° 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).

The last thing to draw is the arc (curved line) that you see in the bottom left square. Start the **ARC** command and select the Center option by pressing the "C" key.

Now you can enter the center point which will be 1, 2. Then you'll be prompted for the start point, enter 3, 2 and finally the end point 1, 4.

```
Command: A <enter> ARC
Specify start point of arc or [Center]: C
Specify center point of arc: 1,2
Specify start point of arc: 3,2
```

```
Specify end point of arc or [Angle/chord Length]: 1,4
```

There are many different ways to draw an arc. Try another method in the other square. Also, you'll learn later that you can draw a circle and then trim it to leave an arc. To finish the drawing, try putting a 10"x7" border around the page starting at 0, 0 using the any of the methods shown above (relative, absolute or polar).

When you have done the assignment, the next step is to print (or plot) it out. To do this, bring up the plot dialog box using any method explained above (**PLOT** <enter>). Set it up to print as shown below. Follow these steps for a successful plot (see diagram below):

Select your printer - laser or inkjet will work fine.

Select the paper size - "Letter" (8-1/2" x 11") is needed in this case.

For the "Plot Area", select "Extents" - that will plot everything you drew.

Select the checkbox to "Center the Plot" on your sheet of paper (looks better).

If "Fit to Paper" is selected by default, uncheck it and select a scale of 1 inch to 1 paper unit (1:1). This will make your printout 'life-sized'.

Now Preview your drawing. I strongly recommend that you **preview EVERY drawing** you will ever draw in AutoCAD - a lot can go wrong, so you don't want to waste paper (especially when you're using expensive 3'x4' sheets!). If your preview looks good, Cancel out of the preview by pressing **ESC**.

If you're sure that everything's ok (this is where good habits begin), press the OK button.

A Plot - Model			×	
Page setup		Plot style table	e (pen assignments)	
Name: <pre></pre>	<none>      Add</none>			
Printer/plrtter		Shaded viewpo	ort options	
Name: HP LaserJet 2200 Series PCL 5	▼ P <u>r</u> operties	Sha <u>d</u> e plot	As displayed 👻	
Plotter: HP LaserJet 2200 Series PCL 5 - Window:	s System Driver K 8,5"	<u>Q</u> uality	Normal	
Where: Art's Desk		DPI	300	
Description:	1.0" -	Plot options		
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Paper size	Number of copies	Plot object	t lineweights	
		Plot transparency		
Letter		📝 Plot with p	plot styl <u>e</u> s	
Plot area	Plot scale	✓ Plot paper	rspace last	
What to plot:	Fit to paper	Hide paperspace objects		
Extents		📃 Plot stamp	p on	
	<u>S</u> cale: <u>1:1</u> ▼	📃 Sa <u>v</u> e char	nges to layout	
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<u>Y</u> : 0.468516 inch	Scale lineweights	Plot upsid	e_down	
Preview	Apply to Layout OK	Cancel	I <u>H</u> elp	

<u>Note:</u> You may have to change the paper size in your printer (Use the Windows printer settings to do this.) You may also have to change the rotation or origin of the plot. Check the **Landscape** radio button in the Drawing Orientation section.

If everything worked out, you should be able to measure your drawing and have it exactly the way you drew it (a couple of 2" squares, an angled 1" square and a 1-1/2" circle).

**Save** your drawing as you would any other Windows file. **CTRL+S** will bring up the Save or Save as dialog box. If you want to check your input, click <u>HERE</u>to see the commands needed to complete this assignment.

#### Video: Assignment 1

This assignment has given you a basic introduction to drawing using a variety of methods. Once you have completed the assignment, practice these methods, as you will be using them often.

Your boxes should match the drawing example. They will be the same size and on the same place in the drawing. If you have a ruler, you will see that your lines are life-size (1:1). From these basic concepts, you can now draw lines anywhere if you know where they should go. The next lessons will add to these commands. As you get used to the AutoCAD interface, you will find it easier and easier to learn the new commands. But for now, mastering the co-ordinate systems, inputting commands and reading the command line should be your priority.

#### Extra Practice #1

Extra Practice: Copy this drawing using the same methods as above. - Extra 001.gif

#### Extra Practice #2

**Extra Practice:** Copy this drawing using the same methods as above (all measurements needed are in the drawing) - <u>Extra 002.gif</u>.

myCADsite recommends these exercise E-books for more practice:



#### Take the Lesson 1-2 Quiz

Which one of these will NOT start the Print command

PLOT

PRINT

ALT+P



### For Further Thought:

You have just learned some powerful commands. Look around you now and see that almost everything can be drawn with lines and arcs and represented in a 2D drawing.

You have learned out to print out a simple drawing and this is an important way to communicate ideas to other people. This is the process of designing > drawing > communication. You might not realize it, but you are well on your way to mastering the process of AutoCAD.

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# **CONTINUE TO THE NEXT LESSON: 1-3**



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#### Topics covered in this Lesson:

#### Rectangle | 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 often in 2D AutoCAD work.

Keystroke	lcon	Location	Result
RECTANGLE /		<u>H</u> ome > Draw > Rectan <u>g</u> le	Draws a rectangle after you enter one corner and ther

REC			
TRIM / TR	-/	Home > <u>M</u> odify > <u>T</u> rim	Trims objects to a selected cutting edge.
EXTEND / EX	/	Home > <u>M</u> odify > Exten <u>d</u>	Extends objects to a selected boundary edge.
OFFSET / O	<u></u>	Home > <u>M</u> odify > Off <u>s</u> et	Offsets an object (parallel) by a set distance
OSNAP / OS / F3	CLICK	<u>T</u> ools > Object S <u>n</u> ap 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.

Duplicate the drawing called <u>Assign 2</u>. Click <u>HERE</u> to download the DWG file.

Once again, do not worry about title blocks, text or dimensions, draw only the geometry (in green).

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

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 <u>absolute coordinates</u>. 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 <u>relative coordinates</u>. This is much faster and also makes the square <u>one object</u> and not 4 separate lines.

**ERASE** the rectangle. You will see that all of it is gone with **<u>one pick</u>**. 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 at **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 <u>top half</u> 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.

: 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 I an. 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 I

### **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.

Drafting Settings						
Snap and Grid Polar Tracki	ng Object Snap	3D Object Snap	Dynamic Input Quic			
☑ Object Snap <u>O</u> n (F3)		V Object Snap T	Frac <u>k</u> ing On (F11)			
Object Snap modes						
🔲 📝 <u>E</u> ndpoint	5	Insertion	Select All			
△ Midpoint	上	Perpendicular	Clear All			
○ ☑ Center	ਰ	Tangent				
🛛 🗌 No <u>d</u> e	X	Nearest				
♦ Quadrant	$\boxtimes$	Apparent inter	section			
X 🔲 Intersection	11	Parallel				
Extension						
To track from an Osnap point, pause over the point while in a command. A tracking vector appears when you move the cursor. To stop tracking, pause over the point again.						
Options OK Cancel <u>H</u> elp						

SETTING	ICON	SETTING
Endpoint	53	Insertion Point

Midpoint	<b>-</b>	Perpendicular
Center	Ó	Tangent
Node	<mark>%</mark>	Nearest
Quadrant	×	Apparent Intersection
Intersection	//	Parallel
Extension	<u>M2P</u>	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

**M2P** - This isn't technically an 'Object Snap' as you are not snapping to specific point on an object, but it allows you to select 2 points and it will calculate the midpoint between those 2 points. This is a very handy option to have.

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**Note:** Beside each checkbox is a symbol. That symbol will show up on the screen when you have found a valid snap point. (An endpoint will show a small square). If you select the "Options" button, you can change the aperture size and the color of the Osnaps. Depending upon the background you are drawing on, this may be needed. Osnaps are explained in more depth in tutorial **4-2**, but this will be enough for you now.

Check off the boxes as shown in the dialog box above (Object Snaps On, Endpoint, Midpoint, Center) and press **OK**.

Begin the **LINE** command. Move your cursor around the screen and you'll see that as you get close to an object, it will 'snap' to one of the points that you had checked off in the dialog box. Place your cursor on the circle (not the middle of the circle) until you see a small purple circle appear at its center. Left-click to make this the start point of the line. Move the cursor towards the middle of the vertical line until you see a small triangle appear. *(Remember this is the symbol for 'midpoint')*. When you see it left-click to accept this as your endpoint. Press **<ENTER>** to end the line command.

Save your drawing.

Print your drawing with the same settings as in <u>Assignment #1</u>. Don't forget to use the preview.

#### Video: Assignment 2

**TIP:** Before you select the Osnap you want, you can press the TAB key on your keyboard to cycle through the available Osnaps in the area of your cursor.

**CAUTION:** Although it may seem tempting to turn 'all' the Osnaps on when drawing, you can have too much of a good thing. For example, in shorter lines, Midpoint, Nearest and Perpendicular could all be very close to each other, and you could select the wrong point.

When you have finished the assignment, continue practicing with these commands until they are mastered. These are common commands that you will use in everyday drafting.

Extra Practice: Copy this drawing, using lines, offset, osnaps - click to view extra\_003.gif

#### Video: Exercise 3

Extra Practice: Copy this drawing, it could a little to figure out, but still uses the commands you have learned so far. - <u>click to view extra\_004.gif</u>

#### Video: Exercise 4

**Extra Practice:** Have a look at <u>this drawing</u> of a simple reflected ceiling plan (RCF) and see how the offset, trim and extend commands might be used. If you know how to change units to Architectural, try drawing it. (<u>Here's how to change units</u>)

Video: Bonus 1

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Take the Lesson 1-3 Quiz



#### For Further Thought:

Now you have learned how to modify something that you have already drawn. This is GREAT! You might remember that I mentioned how CAD was faster than hand-drawing (pencil), so this is the first advantage you will see. You don't have to erase something and start over. These are all tools to make you more productive. It's up to you to decide when to use these tools. As you start drawing more complex designs, you'll have to use these tools to make the quickest and most accurate drawing of that design. That's what AutoCAD is for.

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# **CONTINUE TO THE NEXT LESSON: 1-4**



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#### Topics covered in this Lesson:

Review and practice of accurate input

Now it's time to see if you were **paying attention in the** <u>first lesson</u>. In this lesson you will complete a drawing that will test your knowledge of drawing accurately using different types of precision input. Clients expect 100% accurate drawings from **YOU**. It is up to the **YOU** to ensure that **YOU** have the ability to draw quickly and accurately. There is **no room for error** when drafting, as you need to read accurately, transfer the numbers to AutoCAD accurately and draw accurately. It is the job of this exercise to help you learn to draw accurately - speed will come only through practice.

You can download the DWG file HERE and view the image HERE

Print the file out and try to reproduce it. Start at the bottom left corner (1,1). Use the drawing tools you have learned in the previous lessons. You will have to use a combination of **absolute**, **relative**, and **polar** coordinate input.

Open AutoCAD and set up the drawing as you have for the previous assignments using a template.

Complete the assignment by drawing it accurately using the dimensions provided.

If you make an error along the way, remember that you can use your **ENDPOINT** Osnap to begin where you left off.

Save your drawing as Assignment\_3.dwg.

Print your drawing using the **guidelines** given in the previous assignments.

The following is a list of all the commands required to complete Assignment 3. If your results are different than the sample drawing file, check your input against these. Press the F2 key to see your input. (Press F2 to close the text screen).

NOTE: If you are using AutoCAD R14 (or earlier), your command line may look different, but the input will still be the same.

Remember to press **<ENTER>** after entering each co-ordinate point. A video that shows these steps can be played below.

Command: L LINE From point: 1,1 Specify next point or [Undo]: 4.1,1 Specify next point or [Undo]: 4.5,2.2 Specify next point or [Close/Undo]: @2.3,0 Specify next point or [Close/Undo]: @1.4<30 Specify next point or [Close/Undo]: @0,2.1 Specify next point or [Close/Undo]: @1.6<120 Specify next point or [Close/Undo]: @-1.3,0 Specify next point or [Close/Undo]: @0,-1.4 Specify next point or [Close/Undo]: @2.2<135 Specify next point or [Close/Undo]: @-1,0 Specify next point or [Close/Undo]: @1.5<225 Specify next point or [Close/Undo]: @-1,0 Specify next point or [Close/Undo]: @2.3<-60 Specify next point or [Close/Undo]: 1,1

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As mentioned earlier, AutoCAD is accurate. It is up to the user to ensure that they give the proper input to make sure that the drawing is accurate. It can be extremely costly to make mistakes with what seem to be 'minor' errors in accuracy.

By this point in the tutorials, you should able to do the above drawing without referencing other materials. If this isn't the case, you should review the previous lessons. Here are two more practice exercises to work on your input and drawing skills.

Extra Practice: Copy this drawing - extra 005.gif

Video: Exercise 5

Video: Exercise 6

🕒 Share / Save 🖪 🈏 🖻 ...

Take the Lesson 1-4 Quiz
#### If you draw a line at 270 degrees it will point \_\_\_\_\_

Up Left Down

### For Further Thought:

After showing you new things for the first three lessons, why am I already making you review those concepts?

One simple reason: because they are important! I want everyone to make sure they get solid foundation. I'm not there to force you to practice, but I can do my best to remind you! Do you think if you drew this shape 5 times in a row you would get faster? Learning AutoCAD is all about repetition and learning a little more each time. If you draw this 5 times in a row, you might discover a shortcut on certain segments - and those shortcuts add up.

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## **CONTINUE TO THE NEXT LESSON: 1-5**



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<u>Pr</u>

### Topics covered in this Lesson:

#### Creating Selection Sets | Changing Your Selection 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 <u>selection set</u> 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 several 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 selection' and the other is a 'window selection' (or 'box'). If you create the selection 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:

**NOTE:** New in AutoCAD 2015: By default, the crossing selection is a 'Lasso' in place of the 'window' that many users are not used to seeing. This can be changed in the Options Dialog under Selection > "Allow Press and Drag for Lasso"

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:** <u>Zoom</u> 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 (the mouse wheel is great for this).

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

### Video: Selecting Objects - Part 1

### Video: Selecting Objects - 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.

A Options	
Current profile:         <         <         Vinnamed Profile>>           Files         Display         Open and Save         Plot and Publish         System         User	Current drawing: cabin_floor_plan2.dwg Preferences Drafting 3D Modeling Selection Profiles Online
Pickbox size	Grip si <u>z</u> e
Selection modes         Image: Shift to add to selection         Image: Description of the selection         Image: D	Grips         Grip Colors         Show grips         Show grips within blocks         Show grip tips         Show dynamic grip menu         Allow Ctrl+cycling behavior         Allow Ctrl+cycling behavior         Show single grip on groups         Show bounding box on groups         100       Object selection limit for display of grips         Preview         Selection preview         When a command is active         When no command is active         Visual Effect Settings         Command preview         Property preview
	OK Cancel Apply Help

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.



### Take the Lesson 1-5 Quiz

When selecting objects, pressing 'L' will:

Select lines only Select the last object you modified Select the last object created



### For Further Thought:

I just made you read a lesson that didn't tell you how to draw or modify a single thing. It is still a very valuable lesson. If you need to modify something, you need to select it first.

What you are doing now is finding tools to fill your "AutoCAD Toolbox". Just like a real toolbox, you need to know when to reach for the best tool. By the end of this lesson, you will have a nice set of tools - but I can't always tell you when to use the best one.

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## **CONTINUE TO THE NEXT LESSON: 1-6**



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### 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	lcon	Location	Result
Move	Move / M	+□+ ↓	Home > <u>M</u> odify > Mo <u>v</u> e	Moves an object or objects
Сору	Сору / СР	6	Home > <u>M</u> odify > Cop <u>y</u>	Copies object(s) once or multiple times
Stretch	Stretch /	-	Home > <u>M</u> odify	Stretches an object after you

	S	> Stretc <u>h</u>	have selected a portion of it
Mirror	Mirror / MI	Home > <u>M</u> odify > M <u>i</u> rror	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.

Click <u>HERE</u> to see the file in GIF format. Click <u>HERE</u> to download the DWG file.

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  $10^{\circ}$  wide by  $7^{\circ}$  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 right 1 unit and up 1 unit 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 <u>relative</u> <u>coordinates.</u> 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 **<u>copy</u>** 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.)

Command: M MOVE Select objects: <Select the Rectangle> 1 found Select objects: <Press ENTER to finish selecting objects> Specify base point or [Displacement] <Displacement>: <Pick on corner> Specify second point or <use first point as displacement>: @1,1 <ENTER>

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.

```
Command: C COPY
Select objects: <Select the Rectangle> 1 found
Select objects: <Press ENTER to finish selecting objects>
Current settings: Copy mode = Multiple
Specify base point or [Displacement/mOde]
<Displacement>: <Pick on corner>
Specify displacement <-25'-10 5/16", 11'-11 3/16", 0'-
0">: @3,0 <ENTER>
```

**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 unit taller. To modify this, you'll use the **<u>stretch</u>** 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 <u>1-5</u> 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.



Command: S STRETCH Select objects to stretch by crossing-window or crossingpolygon... Select objects: Specify opposite corner: <Select top of Rectangle with a crossing window> 1 found Select objects: <ENTER> Specify base point or [Displacement] <Displacement>: <Pick on corner> Specify second point or <use first point as displacement>: @0,1 <ENTER>

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.



Command: MI MIRROR Select objects: <SELECT THE 3 LINES> Specify opposite corner: 3 found Select objects: <ENTER> Specify first point of mirror line: <SELECT 8,5> Specify second point of mirror line: <SELECT 8,2> Erase source objects? [Yes/No] <N>: <ENTER>

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.

Save the drawing files as Assignment-4.dwg

### Video: Assignment 4

**Extra Practice:** Copy this drawing using the commands you have learned in lessons up to this point - <u>extra\_007.gif</u>

Video: Exercise 7

**Extra Practice:** This exercise uses an option of the **CIRCLE** command called "Tangent Tangent Radius" or TTR. To use it, pick the two tangents of the circle first, then enter the radius. The rest of it uses previous commands. <u>extra\_008.gif</u> This one is tricky, so try it a few times before checking the video.

### Video: Exercise 8



### Take the Lesson 1-6 Quiz

What is the best way of drawing a rectangle?

Using the Line command

Using the Polyline command

Using the Rectangle command



### For Further Thought:

Are you starting to see a pattern? In your daily drafting work, you will draw > modify > draw > modify... repeat.

For many new CAD users, they try to draw > erase > redraw... and you need to learn when modifying is faster and easier than redrawing a shape. Sometimes it isn't, and learning the differences is what makes some people better CAD users than others. While you are learning is the best time to stop and think about these decisions.

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# RETURN TO THE TOP OF THE LESSON | RETURN TO THE TABLE OF CONTENTS

## **CONTINUE TO THE NEXT LESSON: 1-7**



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### Topics covered in this Lesson:

Rotate, Fillet, Chamfer, Array

Now it's time to learn a few more commands. Like all of the commands learned 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	lcon	Menu	Result
Rotate	Rotate / RO	Ò	Home > <u>M</u> odify > <u>R</u> otate	Rotates objects to a certain angle
Fillet	Fillet / F		Home > <u>M</u> odify > <u>F</u> illet	Creates a round corner between two lines

Chamfer	Chamfer / CHA		Home > <u>M</u> odify > <u>C</u> hamfer	Creates an angled corner between two lines
Array	Array / AR		Home > <u>M</u> odify > <u>A</u> rray	Creates a repeating pattern of the selected objects
				Creates an array using
ArrayClassic	ArrayClassic	none	none	a dialog box for input

Once again you will recreate a drawing. This one is called Assignment #5. Click <u>here</u> to see the GIF format file. Click <u>here</u> 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 2 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 <u>AutoCAD measures</u> <u>angles</u>. 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 you 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>

Enter .375 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>:<ENTER>

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.

### VIDEO: RECTANGULAR ARRAY

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

Start the **ARRAY** command - in newer versions you might see a command line prompt:

Command: AR ARRAY

Select objects: <SELECT RECTANGLE> 1 found
Select objects: <ENTER>
Enter array type [Rectangular/PAth/POlar] <Polar>: R
Type = Rectangular Associative = Yes

Select grip to edit array or [ASsociative/Base point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>: CO

Ambiguous response, please clarify... COUnt or COLumns? COL

Enter the number of columns or [Expression] <4>: 3

Specify the distance between columns or [Total/Expression] <2">>: .375 <ENTER>

Select grip to edit array or [ASsociative/Base
point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>: R

Enter the number of rows or [Expression] <3>: 2

Specify the distance between rows or [Total/Expression]
<2">>: .375 <ENTER>

Specify the incrementing elevation between rows or [Expresson]
<0">>: <ENTER>

Select grip to edit array or [ASsociative/Base
point/COUnt/Spacing/COLumns/Rows/Levels/eXit]<eXit>:<ENTER>

As you can see, it's a lot to enter, but there is another version of this command that you can use. This is the older version of Array now called ARRAYCLASSIC. It will bring up a dialog box that allows you to enter this information and then preview the results. Some people find this easier, especially those of us that have used AutoCAD for a while.

**BREAK TIME :** There is one significant difference between the ARRAY command the ARRAYCLASSIC commands.

The ARRAY command will create an "array object" - this is one object that you can then select with one



Erase the array the you just made and redraw the square. Start the ARRAYCLASSIC command and you'll see this dialog box:



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.

Choose the radio button for "**Rectangular Array**". This will array the object in a row/column arrangement.

Next **select the object** you want to array, by picking on the button in the top right corner. (Press enter when done)

Enter the number of rows (going across the page) and column (running up and down the page).

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. Enter the **number of columns**.

Enter the **Column offset** 

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.

The next time of array used is the **polar array**. Check out the video if you like before proceeding or just follow along with the tutorial.

### VIDEO: POLAR ARRAY

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 **ARRAYCLASSIC** command. When asked to select objects, pick the line you just drew.

🛦 Array	×
○ <u>R</u> ectangular Array	<u>S</u> elect objects
Center point:       X:       7.5000       Y:       5.5000         Method and values	1 objects selected
For angle to fill, a positive value specifies counterclockwise rotation. A negative value specifies clockwise rotation.	OK Cancel
Rotate items as copied     More      More      More     More	Pre <u>v</u> iew <

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.)

If you use the text based **ARRAY** command, you can still enter in this information as shown below:

Command: AR ARRAY

Select objects: <SELECT THE LINE> 1 found

Select objects: Enter array type [Rectangular/PAth/POlar]
<Rectangular>: PO <ENTER>

Type = Polar Associative = Yes

Specify center point of array or [Base point/Axis of rotation]: Select grip to edit array or [ASsociative/Base point/Items/Angle between/Fill angle/ROWs/Levels/ROTate items/eXit]<eXit>: I <ENTER>

Enter number of items in array or [Expression] <6>: 6

Select grip to edit array or [ASsociative/Base
point/Items/Angle between/Fill angle/ROWs/Levels/ROTate
items/eXit]<eXit>: F

Specify the angle to fill (+=ccw, -=cw) or [EXpression]
<360>: <ENTER>

Select grip to edit array or [ASsociative/Base
point/Items/Angle between/Fill angle/ROWs/Levels/ROTate
items/eXit]<eXit>: <ENTER>

**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.

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### Video: Assignment 5

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, especially with these commonly used commands.

Extra Practice: Copy this drawing - extra\_009.gif

### Video: Exercise 9

Extra Practice: Copy this drawing - extra\_010.gif

Video: Exercise 10

**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 - <u>extra\_012.gif</u>

### Video: Exercise 12



### Take the Lesson 1-7 Quiz

Objects are rotated around the \_\_\_\_\_

Bottom right of the object

Centre of the object

**Base point** 



### For Further Thought:

This lesson added some great new tools in your toolbox. These are some powerful commands, so make sure you learn them well.

At this point it's good to remind people that we are just scratching the surface of what AutoCAD can do for you. I am trying to show the most commonly used commands first so that you can start practicing them as you work through the rest of the lessons. As you learn new commands, don't push the old ones aside. I wouldn't be showing them to you if I didn't know you would be using them.

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## **CONTINUE TO THE NEXT LESSON: 1-8**



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### Topics covered in this Lesson:

Layers, Text, Dimensioning, Scale

Now that you've learned a lot of the drawing and modifying commands, it's time to go to the next level. This lesson will introduce you to text, dimensioning and the concept of layers. Without dimensions you can not communicate what size your objects are. Is it a real house or a doll house? Text is used to convey information that is needed other than size - materials, manufacturing process, part numbers, etc.

What you will do is take one of your previous assignments, save it to a new name and then add text and dimensions to it. These are the commands you'll be learning.

Command	Keystroke	lcon	Location	Result		
Layer	Layer / LA		Home > <u>L</u> ayers	Starts the Layer and Linetype property dialog box		
Text	Text	A	Home > Annotate > Single Line Te <u>x</u> t	Creates a single line of text		
Dimension	Dim	Many	Home > Annotate > <u>D</u> imension > ( <i>pick</i> <i>one</i> )	Dimensions previously drawn objects		
Scale	Scale / SC		Home > <u>M</u> odify > Sca <u>l</u> e	Proportionately resizes (or scales) objects		

Begin by opening up <u>Assignment #5</u> from your CAD folder. You have a special CAD folder, right?

Using the SAVE AS 2 option, immediately save it under the name Assign6.dwg

Click <u>HERE</u> for the GIF version of the file. Click <u>HERE</u> for the DWG version.

### Layers

The first thing you want to do is create three **layers**. Layers are used to organize drawings. Imagine a large project for a high-rise tower. The designers would create layers for the electrical, plumbing, landscape and more. It is necessary to control the drawing and turn some layers off and view only the ones you want. This is one reason why layers are needed. When you go on to use AutoCAD professionally, every drawing you deal with will have layers (possibly dozens or hundreds).

You will be creating a layer for the dimensions, one for the objects (lines that were drawn) and another for the text. Start the **LAYER** command (**LA <ENTER>**). This will bring up the Layer Properties Manager Palette (shown below). We'll cover some basic things that are used in this lesson.

Cu		layer: OBJE	\$ <b>5</b>	E A	//	Make	Current	Sea	rch for lay	∝ •	× × ×
»	S	Name	0	Fre	L	Color	Linetype	Lineweight	Trans	Plot St	
	_	0	•	×	<b>f</b>	white	Continuous	Default	0	Color_7	
	-	BORDER	1	×.	<b>.</b>	white	Continuous	Default	0	Color_7	~
	-	Defpoints	1	×	•	white	Continuous	Default	0	Color_7	GER
	_	DIMS	<b>?</b>	×	•	red	Continuous	Default	0	Color_1	NA
	$\checkmark$	OBJECT	9	Ó	Ē	📃 green	Continuous	Default	0		MM
	-	TEXT	•	*	Ð	yellow	Continuous	Default	0	Color_2	OPERTIES

Looking at the Dialog Box, you will see a lot of information.

From left to right, this is what the columns are for (not everything is used in this level).

COLUMN	DESCRIPTION
Status	This column has a green check to indicate the current layer (all new objects will be drawn on that layer). Double click here to make the layer current.
Name	Give the layer a good, descriptive name (but not too long)
On	Turns the visibility of that layer on (visible) or off (invisible)
Freeze	Similar to 'On', but can be used in Viewports (later tutorial)
Lock	Retain visibility, but disable modifying objects on the locked layer.
Color	Used to select the color for objects drawn on each layer
Linetype	Used to select the linetype for objects drawn on each layer
Lineweight	Used to select the lineweight for objects drawn on each layer
Plot Style	Plot styles are used to define how objects are plotted - this can be different from the properties described above.
Plot	Set the layer to either plot (on) or not (off)
New VP Freeze	When on, it will freeze the selected layer in new viewports when they (viewports) are created
Description	Use this to give a description to your layer - very important when working with other users.

Now you have the dialog box on your screen, click the **'New'** button <sup>2</sup>. This will create a new layer and give you an opportunity to name it. Name this layer **TEXT**. Click on the small box in the color column and select **yellow** for the color. Create another layer and call this one **DIMS** (for dimensions), and make its color **red**. Finally, create a third layer and call it **OBJECT** and make its color **green**. You have just created three new layers in your drawing. Highlight the TEXT layer and then press the 'Make Current' button. This makes the TEXT layer current and anything you draw will be placed on that layer. Your Dialog box should look like the one above. If it does, press **OK** to close the dialog box.

Now that you have your layers, select all your objects and then look for the layer tool panel. There is a droplist there with all of your layer names. With your objects selected, change the layer to OBJECT. If all went well, your lines and circles will now be green and on the object layer.





### ADDING TEXT

A <u>later tutorial</u> will discuss text in more detail, but for now see what is involved in adding text. Make **TEXT** is your current drawing layer, so what you will be doing is

creating some text now. Type in **TEXT** on the command line. Look at the command line.

#### Command: TEXT Justify/Style/<Start point>: 0,0 <ENTER>

The first thing AutoCAD wants is a starting point for the text. Type: 0, 0 to place it at the bottom left corner of your drawing. Notice that there are other options you could make, but we'll ignore them for now.

#### Height <0.2000>: <ENTER>

Next, AutoCAD wants to know how tall you want your letters to be. For this assignment, you want them to be 1/4" tall. Type in .25 at the Height prompt.

#### Rotation angle <0>: <ENTER>

One more thing before you start typing in text, tell AutoCAD if you want to have your text rotated. Not this time, so press **<ENTER>** to accept the default of 0 degrees.

#### Text: YOUR NAME - ASSIGNMENT #6

Finally you can type in what you want to put on the drawing. At the prompt, type your NAME and ASSIGNMENT #6 then press <ENTER>.

You will see that AutoCAD has placed your name in the bottom left corner at 1/4" high and in yellow on the 'Text' layer. You don't really want your name jammed into the corner like that so move it up and over 1/8" (*Remember your* **MOVE** *command and relative points* : @.125,.125)

You can learn more about text by jumping ahead to Lesson 2-6

### Dimensioning in AutoCAD

Now it's time to start dimensioning your drawing. Bring up the Layer dialog box and make **DIM** your current layer.

This is one time when I recommend to use the icons. Dimensioning is a LOT easier this way. If you don't see your dimensioning toolbar on the screen, right click on one

of your toolbar icons and check the Dimension checkbox. You'll see the toolbar appear. Close the Toolbars dialog box.

There are several different types of dimensions, here are the ones you'll be using in this lesson (Each of the dimensioning icons gives you a quick clue as to which type of dimension it will create.)



Radius

Diameter

Ordinate

Jogged

Here are the ones you'll be using in this lesson:

COLUMN	DESCRIPTION
LINEAR	dimensions are used for dimensioning either horizontal or vertical distances.
ALIGNED	dimensions will measure the actual length of an angled line.
RADIUS	dimensions will give you the radius of either arcs or circles.
DIAMETER	dimensions are used on circles.
ANGULAR	dimensions will measure the angle between two lines that you pick.

# **BASELINE** dimensions are a special type that will automatically stack dimensions along one plane as you pick points. This is only seen on the Annotate Ribbon with 'Continue'.

Once you have your objects drawn, you need to go to the menu at the top of the screen and click on Annotate. This will show the Dimension tool panel as well as the TEXT panel and others.



The type of dimension you pick will depend entirely upon what information you want to convey to the person reading the drawing. Look at the sample drawing and pick out the different types of dimensions and where they are used.

We'll start with the most common (and easiest): **Linear Dimensions**. You'll use this type to add dimensions to the rectangle in the bottom left of your drawing. Pick on the icon and then look at the command line.

```
Command: _dim
Select objects or specify first extension line origin or
[Angular/Baseline/Continue/Ordinate/aliGn/Distribute/Layer/Und
o]: <SELECT TOP LEFT CORNER>
```

It asks for either the origin of the first dimension line (a starting point) or you can press <ENTER> to select a particular line. Turn your **OSNAPS** on to **endpoint**only. Pick the top left corner of the rectangle. AutoCAD then asks for the second extension line origin. You'll then see the dimension appear and next Pick the top right corner. AutoCAD asks for the *Dimension line location*. Pick somewhere just above the line where you think it fits well.

AutoCAD then shows you the length that it found to be the length of the line you just dimensioned. The following lines show what you should have seen on your command line:

Specify first extension line origin or [Angular/Baseline/Continue/Ordinate/aliGn/Distribute/Layer/Und o]: <SELECT TOP RIGHT CORNER> Specify second extension line origin or [Undo]: Specify dimension line location or second line for angle [Mtext/Text/text aNgle/Undo]: <PLACE THE DIMENSION> Select objects or specify first extension line origin or [Angular/Baseline/Continue/Ordinate/aliGn/Distribute/Layer/Und o]: <ENTER>

Now dimension the line on the left side using the same method.

Now you want to dimension the radius of the fillet on the object above the first box. Pick the icon for **radius dimension**. AutoCAD asks you to select an arc or circle. Pick the arc on the top left corner. Again AutoCAD then wants you to pick the dimension line location. Pick anywhere outside the arc where the dimension fits well.

Looking at the top left object on the sample sheet, notice that there are a few ways to dimension the corners. Start on the bottom left. Pick the **Linear Dimension**icon. Instead of picking the two endpoints, press **<ENTER>** and AutoCAD will ask you to select the object you want to dimension. Pick on the angled line in the bottom left corner. As you move your cursor around, you'll see that you could place it either to the left or below the line. Choose one or the other and place your dimension. Repeat these steps to add the other dimension.

At the top left, you want to add an **Aligned dimension**. Pick the icon for this. As with the linear dimensions, you can either pick the endpoints or press **<ENTER**>to choose an object. When you're asked to place the dimension, you'll see that you can only

move parallel to the line that you picked. Place the dimension line somewhere outside the object.

Now you're going to dimension the angle on the bottom right corner of the top box. Pick the **Angular dimension** icon. Pick the bottom line and then the angled line in the bottom right corner. Place the dimension so that it looks like the one in the sample.

Now its time to dimension the circle. Pick the icon for the **Diameter dimension**. AutoCAD then wants you to select the arc or circle. Pick anywhere on the circle. Then you have to place the dimension line somewhere. Pick a good spot for it.

Dimension the 1" line in the circle as shown using a linear dimension.

Dimension the top left box of the array as shown using Linear dimensions.

Dimension the space (0.75) between the two rows by picking the endpoints of the lines.

Dimension the space (0.75) between the columns as shown.

Now for the tricky part. You want to add baseline dimensions to the column distance dimension you just created. Pick the icon for the baseline dimensions. Look at the command line:

You may be asked to select the base dimension. If so, pick on the 0.75 dimension for the column. Then you're asked to **Specify a second extension line origin**. Pick where **P1** shows you on the sample drawing. AutoCAD then will get you to keep repeating this process until you press **<ENTER>** to end the command. Pick near **P2** to continue, then press **<ENTER>**. This is what your command line should have looked like:

```
Command: _dimbaseline
Select base dimension:
Specify a second extension line origin or
(Undo/<Select>): <PICK P1>
Dimension text = 1.50
Specify a second extension line origin or
(Undo/<Select>): <PICK P2>
Dimension text = 2.00
Specify a second extension line origin or
(Undo/<Select>): <ENTER>
```

Your drawing should now look similar to the sample drawing. Check to see if you missed any dimensions.

Now you have dimensioned the drawing, save it and print it.

Next you want to copy everything over 12" to the right. Start the **COPY** command, and when asked to select objects, type **ALL <ENTER>**. Then copy it all 12" to the right.

At this point, I want you to learn the drawing techniques. If you want to (or need to) learn more about dimensioning, jump ahead to these tutorials: <u>4-11(dimension styles)</u> and <u>4-11a</u> (types of dimensions)

What you want to do next is **scale** the copy of everything to double its existing size. Start the **scale** command. When asked to select objects, draw a window around the **new** set of objects and press **<ENTER>**. When asked for a base point, pick the bottom left corner of the border. Enter **2** for the scale factor. Your command line should look like this:

```
Command: SC <ENTER>
SCALE
Select objects: Other corner: 38 found
Select objects: <ENTER>
Base point: <Pick the Bottom Left Corner>
<Scale factor>/Reference: 2 <ENTER>
```

Perform a Zoom Extents (**z <ENTER> E <ENTER>**) at the command line. Now that everything is twice as large, Zoom in on your dimensions and you'll see that they have automatically adjusted to the new sizes! This is called *associative dimensioning*.

There are many different parameters you can change to get just the right look for your dimensions and text (use the DDIM command). This is will not be covered in this lesson, but is explained in <u>Lesson 4-11</u>.

For practice, you can try dimensioning some of your other assignments.

Even if your drawing is 100% accurate, it is still only as good as the dimensions. Some common mistakes when dimensioning are:

- 1. Not using your Osnaps. If you don't get the exact endpoint of a line, then your dimension is wrong.
- 2. Snapping to the wrong point. In detailed drawings, this can be easy to do if you aren't paying attention.
- 3. Incomplete dimensions. If you miss one or two dimensions, you can delay the project while these numbers are located, new drawings made, delivered, etc..
- 4. Confusing dimensions. Make sure that your dimensions don't overlap, or aren't too close, or otherwise unclear.
- 5. If you are drawing other objects and your dimensions are not the right size, **check out this info** about the **DIMSCALE** setting.

### Video: Assignment 6

For dimensioning practice, draw and dimension the extra lessons from the previous lessons. Also put objects and dimensions on separate layers.

Extra_001.gif	<u>Extra</u>	002.gif	<u>Extra</u>	003.gif	<u>Extra</u>	004.gif	<u>Extra</u>	005.gif
Extra_006.gif	<u>Extra</u>	<u>007.gif</u>	<u>Extra</u>	<u>008.gif</u>	<u>Extra</u>	<u>009.gif</u>	<u>Extra</u>	<u>010.gif</u>
Extra_011.gif	<u>Extra</u>	012.gif	<u>Extra</u>	013.gif	<u>Extra</u>	014.gif	<u>Extra</u>	<u>015.gif</u>

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### Take the Lesson 1-8 Quiz

You should use Osnaps when you dimension

Only on large drawings

Always

Never



### For Further Thought:

While a lot of time with AutoCAD is spent drawing and modifying objects, the goal is to present a drawing or set of drawings that communicate to other people what needs to built.

If your drawing is awesome, but your dimensions don't make sense or are not clear, you're not helping anyone. If you have a complex drawing on one layer, anyone who works on it will not enjoy modifying it. Layers take time to set up, but save a lot of time down the road.

Some times you need to add text and notes to make sure that the person reading the AutoCAD drawing knows exactly what you need. You need to make sure this text is clear, easy to read and precise. You need to re-read the text, spell check it and then make sure it's in the correct spot.

AutoCAD is more than drawing, remember - it is communication.

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## **CONTINUE TO THE NEXT LESSON: 1-9**



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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, but you still have to be careful.

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, circle radii or other objects. If Ortho is not active your lines will not be at right angles and 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** (or **DS**) and you get the Drafting Settings dialog box, go to the "Polar Tracking" tab.

A Drafting Settings	×
Snap and Grid Polar Tracking Object Snap	3D Object Snap Dynamic Input Quic
Polar Tracking On (F10)           Polar Angle Settings          Increment angle:         45                Polar Angle Settings          Increment angle:                45                Polar Angle Settings          Increment angle:                45                90                 135                 225                 315	Object Snap Tracking Settings Track orthogonally only Track using all polar angle settings Polar Angle measurement Absolute Relative to last segment
Options	OK Cancel <u>H</u> elp

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 <u>Lesson 1-2</u> and draw it again 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!

#### Video: DDE and Polar Tracking

#### **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 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.

### Video: Object Tracking

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### **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 line 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 1.85 units (which I manually entered). Once I enter the distance, I can press the Tab key to switch over to the Degrees and enter a value there (140). I could also have typed @1.85<140 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.

#### Video: Dynamic Input

**Extra Practice:** Try drawing the **extra exercises** again using these methods. Find out which one works best for you, and when to use the different techniques in various situations. See if you like using Dynamic Input, or if you prefer to have it off. If you can draw all these exercises quickly (about 1-2 minutes) you have learned the lessons and should move on to the next lesson.

## Function Keys in AutoCAD

In this lesson you were shown how some of the Function Keys (F Keys) work in AutoCAD. These are great shortcuts to know and remember. I find that I can press and F key faster than I can find an icon. And for something like turning Osnaps on and off - that's a real timesaver.

Here's a table that shows the different Function Keys in AutoCAD (some of them you haven't seen yet).

F KEY	FUNCTION
F1	Brings up the AutoCAD Help Topics
F2	Toggles the Text Screen(Command Line History)
F3	Toggles Object Snap
F4	Toggle 3D Osnap
F5	Toggles Isometric Drawing Mode
F6	Toggles Dynamic UCS
F7	Toggles Screen Grid
F8	Toggles Ortho Mode
F9	Toggles Snap Mode (snap to screen points)
F10	Toggles Polar Mode
F11	Toggles Object Snap Tracking Mode
F12	Toggles Dynamic Input Mode



#### Take the Lesson 1-9 Quiz

DDE can be used in which direction?

45 degree increments

90 degree increments

All



### For Further Thought:

Here we are at the ninth lesson and I finally showed you the easy way to do something. Did I wait this long because I don't like you? Nope - I make sure my students learn the procedures behind the shortcuts first. That way I know that you'll understand why the shortcut can be used and what to look for when it goes wrong.

Shortcuts are great - I use them all the time in any program that has them. Using shortcuts in AutoCAD will make your a faster drafter - but used incorrectly, they may cause you to be less accurate.

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## CONTINUE TO THE NEXT LESSON: 1-10



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<u>Priv</u>

#### 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

Command	Keystroke	lcon	Location	Result
Properties	PROPERTIES / PR / CTRL+1	No Icon	Home > <u>Pr</u> operties	Displays the properties of the object in the Properties Palette
Match Properties	MATCHPROP / MA PAINTER	Match Properties	Home > Clipboard > <u>M</u> atch Properties	Copies the properties from one object to another

There isn't an icon for Properties, but if you look at the ribbon, you'll see that there are a number of tools that you can select quickly to see things like layer, linetype, lineweight, etc.

. 📃 🔪	ByLayer	•	
Match	Match	ByLayer	•
Properties		ByLayer	•
		Properties 👻	ч

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

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 **right-clicking** on it and choosing Properties from the context menu. Once you do this, you should see the properties palette appear showing everything that makes that line what it is.

CLOSE PALETTE	х н	Line	• 📽 ÷ 🐇	
AUTO-HIDE	×	General	-	
(COLLAPSE)		Color	ByLayer	
		Layer	WALL	
PALETTE PROPERTIES		Linetype	ByLayer	
		Linetype scale	1.0000	
		Plot style	ByColor	
		Lineweight	———— ByLayer	
		Transparency	ByLayer	
		Hyperlink		
		Thickness	0.0000	
		3D Visualization	-	
		Material	ByLayer	
		Geometry	-	
		Start X	-320.5915	
		Start Y	-3.6202	
		Start Z	0.0000	
		End X	-310.3533	
		End Y	6.6181	
		End Z	0.0000	
		Delta X	10.2383	
		Delta Y	10.2383	
		Delta Z	0.0000	
		Length	14.4791	
		Angle	45	
	PROPERTIES			
	PER			
	RO			
DOCK PALETTE	Ē			

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 (2010 and newer). 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 (color, layer, linetype, etc).

Below that is the **'Geometry'** tab and these fields will be different for different types of objects. 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 on the right of the text box.

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Have a 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.

**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. 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.

×	All (13)			-	đ	- <del> </del> -	-4
×	General						-
PROPERT	Color		ByL	.ayer			
BE	Layer		*VARIE	S*			
PRO	Linety	be		— I	ByLa	yer	
₽	Linety	oe scale	1.0000	)			

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).

 Circle			- CUI	X ≓
Color	ByLayer			
Layer	WALL			
Linetype		ByLayer		
Center X	-320.7083			
Center Y	3.5039			
Radius	5.9922			
Diameter	11.9844			
Circumference	37.6501			
Area	112.8033			

## **TEXT OBJECTS**

Now 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 <u>Lesson 1-8.</u> View the properties like you did in the previous examples. Note again, the various fields that in this case are unique to Text.

	Text		X ≓
	Layer	WALL	
	Contents	PROPERTIES	
PROP	Style	mcs	
	Annotative	No	
_	Justify	Left	
	Height	1.1060	
	Rotation	0	

Try "Double-Clicking" on the text.

Now you can change the text and then click away from the text to submit the changes. 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 (shown above), but this method is much quicker.

## Changing an Object's Layer

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 <u>Lesson 1-8</u>) 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 **MATCHPROP** 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.

To try this out, draw 2 lines and make sure that they are on two different layers. Start the **MATCHPROP** command and select the source object. Notice how your cursor changes? Now select your destination object(s).



### 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.

Video: Object Properties

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#### Take the Lesson 1-10 Quiz

How can you change an object's properties?

Right Click and choose Quick Properties

Right click and choose Properties



### For Further Thought:

You may be noticing that I'm showing less drawing tools at this point. Do you know why?

What have learned so far in the previous lessons will account for the bulk of your drawing and modifying commands. You can draw a floor plan or machine part with what you already know. But if I ended the lessons here, I wouldn't be helping you much, would I? I want my students to learn about more than just drawing lines and circles. I want you to learn what is needed to make a professional AutoCAD drawing. I want you to know the best way to create an AutoCAD drawing. This is why myCADsite has been popular for so long. You can watch a Youtube video and learn how to draw a line - or you can follow these lessons and learn how to create a drawing.

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All

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#### CONTINUE TO THE NEXT LESSON: 1-11



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### 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 <u>Level 1</u>, 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 (you 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>:

Below are some of the command line options explained in more detail.

COMMAND OPTION	DESCRIPTION
Zoom <u>Ex</u> tents	This option will display all the graphics that are contained in the drawing (referred to as the <i>drawing extents</i> ) with the largest image possible.
Zoom <u>W</u> indow	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 <u>P</u> revious	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 <b>avoid the scroll bars</b> for panning a lot in your drawing.
Zoom <u>R</u> ealtime	Zoom Realtime provides interactive zooming capability. Pressing <b><enter></enter></b> (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).</enter>
Zoom <u>A</u> ll	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 <u>D</u> ynamic	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.</enter>
Zoom <u>S</u> cale	This is another default option. You do <u>not</u> 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 <i>to the current view</i> . 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.
Zoom <u>C</u> enter	This option requires two things: a point that is to be the <u>center</u> of the new display and a value to be its <u>new height</u> <i>in drawing units</i> . 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.
Zoom <u>O</u> bject	This option asks you to select an object or objects, then press <b><enter></enter></b> and the screen will zoom to those objects only. This is great for when you want to

	work on (or find) the last object you created.
Zoom In	Clicking this menu item 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 menu item will zoom out of your drawing and allow you to see about 50% more of your drawing space.
Mouse Scroll Wheel	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 zoomfactor <u>system</u> <u>variable</u> . Keep in mind that you will zoom in and out using your mouse location as a 'center point'.
Aerial View command: . <b>DSVIEWER</b>	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.
PAN	Panning allows you to quickly move around the drawing area at the same magnification you currently have set. Type in <b>PAN</b> (or <b>P</b> ) <enter> and a hand will appear on the screen. Left click and hold to move around your drawing.</enter>

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 <b>Zoom</b>	>	<b>Extents</b> whenever you want to see all objects.
Use the <b>Zoom</b>	>	Window option to 'close-in' on one area.
Use the <b>Zoom</b>	>	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 in, 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 <u>transparently</u>. 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.



pressing **<ENTER>**.

If you click on the down arrow of the icon show in the image to the left, you'll see that you can find the options for zooming that I described earlier. This is another, slower way to get to them.



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



Video: Zoom and Pan



Take the Lesson 1-11 Quiz

#### To invoke the Zoom command transparently, type \_\_\_\_\_



### For Further Thought:

I hope you followed this lesson well. I can't stress enough how important it is to move around your drawing in an efficient manner. With these simple drawings you're using in the current exercises not much can go wrong. When you are working on large drawings like site plans or a complex assembly drawing, you need to know where are you are all times. Think about having a desk full of papers and shuffling them around - then trying to find the one word in that stack. That is what moving around a large complex drawing will feel like if you're not organized and using your tools correctly.

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## <u>CONTINUE TO THE NEXT LESSON: 1-</u> <u>12</u>



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<u>Priva</u>

#### Topics covered in this Lesson:

#### Orthographic Drawing Techniques in AutoCAD

#### 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 (such as with <u>Elevation Drawings</u>). 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. These three views are 'projected' and aligned with each other. As you've seen with your Orthographic Mode setting (F8), lines are either horizontal or vertical - and in Orthographic Projection, the views are mostly horizontal or vertical in relation to each other. It's this alignment (projection) that makes the drawing easy to read.

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 <u>Wikipedia</u> to explain this:

#### First-angle projection (European Standards)

In **first-angle projection**, the object is conceptually located in quadrant **I**, i.e. it **floats above and before** the viewing planes, the planes are **opaque**, and each view is **pushed** through the object onto the plane furthest from it. (Mnemonic: an "actor on a stage".) Extending to the 6-sided box, each view of the object is projected in the direction (sense) of sight of the object, onto the (opaque) interior walls of the box; that is, each view of the object is drawn on the opposite side of the box. A two-dimensional representation of the object is then created by "unfolding" the box, to view all of the **interior** walls. This produces two **plans** and four **elevations**. A simpler way to visualize this is to place the object on top of an upside-down bowl. 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 behind** the viewing planes, the planes are **transparent**, and each view is **pulled** onto the plane closest to it. (Mnemonic: a "shark in a tank", esp. that is sunken into the floor.) Using the 6sided viewing box, each view of the object is projected opposite to the direction (sense) of sight, onto the (transparent) exterior walls of the box; that is, each view of the object is drawn on the same side of the box. The box is then unfolded to view all of its **exterior** walls. A simpler way to visualize this is to place the object in the 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 individual 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 - thanks 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.

Because I am in North America and was trained to use the "Third Angle Projection" method, that is what I will show in this tutorial. The techniques for drawing are the same, it's just a matter of which direction you 'project' or draw the lines.

Here's a short video for you that explains these concept visually.

#### Video: Orthographic Projection - Part 1

#### 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 <u>**Tutorial 4-3**</u> 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 you draw the lines if 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.

**Extra Practice:** Draw the missing Right Side View and the other views (except Isometric) for <u>this</u> <u>drawing</u>.

**Extra Practice:** <u>Here is a scan</u> from an old (1919) drafting book that I found on Google Books. This image has 4 separate exercise to keep you busy. Draw what you are given, and then draw the missing view.

#### Extra Practice: Draw the 3 views needed for each of these blocks. Extra 1 - Extra 2

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? Yes it does.



The drawing below shows how this could be drawn to show all aspects of this part. I have the diameter of the pipe and the length to the top and the length to the bottom of the cut. Since I have the diameter, I also have the circumference. The circumference tells me the length of material that will be needed to make the piece.

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.



Video: Orthographic Projection - Part 2



### Take the Lesson 1-12 Quiz

Is it important to have exactly vertical or horizontal lines Orthographic projection?

Yes

No

Only in 2D drafting



## For Further Thought:

So far I have only taught you how to use AutoCAD. This is the first lesson that showed you some basic concepts in drafting. For some of you this might be old news. For many of you this lesson opened the door to what might be expected of you when you start your professional career. Not every field will use orthographic projection, but most will. You will need to learn how to draw with these methods and also understand how to read an AutoCAD file that is laid out this way. It is very common and it will cross your path at some time.

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## CONTINUE TO THE NEXT LESSON: 1-13



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<u>P</u>

### Topics covered in this Lesson:

**Section View Drawing Techniques** 

### Putting your skills to use with Section Views.

In drafting there are a times where you need to give a lot of information, but not all of the views are helping to get your message across. Sometimes you just need to slice the object(s) down the middle and show what the guts look like. Using this method, you get a cross section of the part, or a "**Section View**".



Take a look at the drawing above. You'll see a complex assembly drawing and two different section views. On the far left, you see the line that indicates where **Section B-B** is. On the right side, you see the section view. Does it make sense to you? What you are seeing is the cross section view of the metal beam part of the of the bracket. It's shown so that you can see what size the area is without having to look at the entire side view of the assembly. Section views are used for clarity.

Look for the indicators for **Section A-A**. Then look at the top right view. You'll see that the beam part is shown as well as the hole that is drilled through it.

Section views are used in Mechanical and Architectural drafting. Anytime you need to show some detail that isn't readily seen from an exterior view, "cut it up" and draw it.

### Required knowledge before beginning

If you haven't already read through <u>**Tutorial 1-12**</u> (Orthographic Projection), you should to get familiar with some of the concepts for this tutorial.

You should also jump ahead and read <u>**Tutorial 2-5**</u> as it deals with Hatching, which is used in Section views.

## Looking at Section Views

If you understand Orthographic Projection and Hatching (you read the other two tutorials, right?) then you should be able to handle section views easily. Have a look at the image below and see if it all makes sense.



What you see is a very simple block with a hole drilled. This shows that the section view and the front view are almost the same. In fact, I just copied the Front view down to begin the section view. I changed some linetypes and then hatched it.

If you have more than one part in your view, you will need to make sure that it stands out. Here I have the same part as above, but with another piece placed inside it.



Here's an example of how a section view can be used in Architectural drafting. By using standard hatch patterns, someone else can look at the drawing and see that this shows a concrete foundation with earth on one side.



I think by now you're getting the idea of how section views and hatching help to tell the story of what you are drawing.

## Drawing Section views in AutoCAD

For this exercise, we'll start with a simple drawing. Begin by drawing the part below, and as you're drawing, think about what the section view will look like. You do not need to draw the Isometric view, it is just for reference.



To draw the **"Cutting Plane"** that marks the section, draw a line and then use the leader command to make the arrows. Add text for the "A"s. Ok - what is your section view going to look like?

The Cutting Plane points up from the front view, It doesn't mean that it's pointing at the top view, but in this case it works. Also note the the Cutting Plane is in the middle. So if you chose the Top View, you're correct.

Copy the Top View. Straight up will work fine. You can keep the dimensions. It's a good idea to make sure that it is directly above the other view in case you have to stretch part of it. Then you can do both views at once.

What lines need to be changed? Do you still need hidden lines?

This is what your Section view should look like before hatching:



Is it still making sense? You needed to trim the line where the shaft on the right joins the part, because in the section view, this is solid. You needed to change the lines for the two holes from hidden to solid because they are not hidden any more. You should now be ready to add a Hatch pattern to the view.

The Hatch command might be a little different depending upon the version that you are using. This example shows AutoCAD 2016. Hatch is used to show which areas have been 'cut away' in the section view (and where there would be solid material).

Start the **Hatch** command and pick in the spots shown below. You want all three areas picked at the same time so that the hatch creates one object instead of three. This is needed so that you can edit the hatch and all the areas will look the same.



The areas picked will fill up with the last used hatch pattern and settings that were used. That's fine, we'll edit them now.



This is a big toolbar / ribbon for this command. There are really only three items you need to be aware of to make it work:

**Pattern:** The pattern should indicate what material is used in the area that is hatched.

**Angle:** Most of the time this will be 0 degrees, but sometimes you need to change it. Most standard hatch patterns are expected to be at certain angles.

**Scale:** This is the one that gets changed a lot. If your hatch pattern 'looks solid' - then you usually have to increase the scale because the lines are too close together. If the hatch pattern 'looks empty' - then you might have to scale it down because the lines are too far apart. The scale will vary depending upon what size your drawing is.

If everything went well, the Section view you have should look like this:



It's up to you if you want to put your hatch on a separate layer. It can help a lot as you get into more complex drawings. As you learn about printing through Layout

Tabs, think about how having your Hatch can be useful on it's own layer - and maybe more than one layer.

### Other uses for Section Views

Here is a section view that is used to show a break in a solid bar and a pipe. This is a common way of showing long pipes that don't fit in your drawing, just add a break, but dimension the full length.



Here's an exercise for you: reproduce the image of a foundation below:



More exercises from <u>Textbook of Machine Drawing</u>. Draw <u>these parts</u> out in Orthographic View and then draw the Section view.

### Conclusion

Section views are not usually very difficult to draw, because you should have a lot of the information first. You will draw your sections last and try to base them off of existing drawings.

If you have the opportunity, look at drawing sets and see where they are used. If you are drawing something try out a section view, it's good practice for Hatching and also for developing your "CAD" eye.

Here's a video that shows helps explain the Section view drawing technique.

### Video: Section Views

**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.

To 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.



### Take the Lesson 1-12 Quiz

Why is the Hatch command used in section views?

Hatch is not used

To show the correct color

To show what area has been 'cut away'



### For Further Thought:

Another drafting lesson. Why am I showing you how to draft? I have given you the tools in the earlier lessons, now I'm going to show you how to use them. I don't want to send you out in the world thinking you know how to draft when you only know how to draw lines. I want to provide you with a solid background in using AutoCAD and creating useful drawings. The next level will show you some drafting concepts as well as some very powerful AutoCAD tools. Hang on! It's going to be fun!

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# TAKE THE LEVEL 1 QUIZ

# **CONTINUE TO THE NEXT LEVEL**



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# 2.1: USING THE TOOLS FROM LEVEL 1 IN A REAL PROJECT

LESSON 2-2	INTRODUCTION TO BLOCKS - CREATING AND INSERTING
LESSON 2-3	ALIGN AND ROTATE
	ATTRIBUTES - NON-GRAPHICAL INFORMATION
LESSON 2-5	HATCHING - FILLING AREAS
LESSON 2-6	WORKING WITH TEXT
LESSON 2-7	POLYLINES
LESSON 2-8	LAYOUT TABS / PAPER SPACE
LESSON 2-9	PARAMETRIC CONSTRAINTS
<u>LESSON 2-</u> 10	ELEVATION DRAWINGS

Topics covered in this Lesson:

Starting and Organizing a Project

This lesson will have you using the tools you learned in the <u>first level</u>. Believe it or not, if you understand the commands taught in the first level, you understand almost all of the common commands used in AutoCAD 2D Drafting - for any version.

Now that you know how the tools are used, you need to know when to use the best tool for the job. In this tutorial you will draw a simple floor plan. You can consider this your first project.

I'm not referring to being a Project Manager in this tutorial, but organizing a project in AutoCAD takes a standard approach. Consider drawing a room a 'project', or a floor. On your first day of the job given a project and asked to draw it. What you draw will depend upon the 'Scope of Work'

This example will assume that you've been put in a small room and told to draw it. The first thing you want to do is set up your drawing. There are a few basic steps to approaching **EVERY** drawing you do in CAD.

**Make sure you have the ability to draw it.** See if there is anything in the drawing or area that you would not be able to reproduce. You should also see if you have all the information you need to complete the drawing, or at least get a majority of it done. For example, if there are angled walls, can you find that angle? Curved Walls?

**Are you using a template** to start the drawing with? Your Project Manager should provide you with either the standard company (or a client's) template.

**Once you have this basic information, you can begin.** As you can see, there is a bit you have to do before drawing your first line. Get into the good habit of beginning your drawings properly and **never** with the attitude that you can always "Fix it later!"

Below is the project that you will be drawing.

Once your drawing is set up, think about how you will actually draw it. You should start with the most basic components first. Consider how you would build this structure. Start with the foundation and add more detail as you go. Look at the outer walls and start there. Then add more details such as the door openings, doors, windows, etc. Finish up with the details that are on the scope of work - fire extinguishers, outlets, etc. Be careful with your measurements, because if you make a mistake at the start, it will cause BIG problems later on as you continue through the drawing.

A general rule I use is to **draw like I would build it**. This basic approach will at least give you a starting point for any project in any discipline.

Just like in Previous Lessons, start AutoCAD and a new drawing by using the menu option **File > New**. You will see a dialog box open that asks you to select a template drawing to use. In these examples you will use AutoCAD's default template. This will give you the chance to practice creating and using layers. Have a look at the types of objects in the drawing example and create layers for each.

The first two projects are designed using imperial, architectural units (i.e.: 3'-6") as opposed to the decimal measurements you used in the previous level. If your template is not set to Architectural units, you can do this by using the DDUNITS command and set "Length > Type" to Architectural.

#### Notes on entering Architectural Units:

Entering Architectural units requires a different way of inputting numbers that regular decimal units. Here are some ways of entering them:

11'10-1/2" 11'10-1/2 3' (or) 36" (or) 36

You don't ever need to use the inch symbol. Inches can be entered instead of feet (i.e. 48=4').

### **Computer Room Project**

Start a new drawing and create the <u>layers</u> that you need. You should have 4 layers. (Walls, Furniture, Doors, Dims).

#### Where to start:

Here is the room you will be drawing. Imagine that you just walked in the door at the top right, have a look around...

You'll see two more doors, and some parts of the wall sticking out - where the columns are. Imagine that you have to measure it, and draw it. You would start with the largest measurements first for the walls, then the columns and then draw the 23 desks last.

You're ready to start drawing now. As mentioned above, start with the outside and work your way in. If you are drawing the first project, you can easily draw the walls (using **DDE**), then offset them by the thickness of the wall. From there, it's just a matter of inserting the door openings. I usually do that by offsetting lines, then trimming away what I don't need. For the doors themselves, just draw a rectangle and rotate it (later on you will learn about blocks for this). So with just a few commands, you can draw this room - all the commands you learned in Level 1. Of course, there are many ways to draw the same project.



Click here to see **<u>one way to draw doors</u>** if you're not sure what to do for them.

Once you have drawn the basic plan, dimension it (refer to <u>Lesson 1-8</u>). From the "Computer Room" plan you will be adding computers and other accessories to the desks later in the course, so save the file when you are done.



If you want to dimension this drawing, you might find that the default size is too small. The scale / size of your dimensions is controlled by the **DIMSCALE**system variable. Try a value of about 12 for this drawing and try 100 for the metric drawing.

If you want to modify your dimensions further, please jump ahead to **this tutorial**.

### Video: Computer Room Project - Part 1

### Video: Computer Room Project - Part 2

Below are 3 projects to work on. Draw them and think about how you might approach them.

PROJECT	GIF IMAGE	AUTOCAD DWG FILE
Project # 1 - Computer Room	GIF	20
Project #1a - Computer Room (Metric)	GIF	28
Project # 2 - Office Space	GIF	20
Project #3 - Cabin	GIF	Not Available
Project #4 - Cabin 2	GIF	28

### Video: Office Space Project

**Extra Practice:** Here is a PDF of an <u>older scan of a floor plan</u>. One job you might have as an entry level CAD user is to translate older blueprint drawings into CAD

drawings. Note the use of Door and Window schedules. These are a key to sizes used in the plan. See if you can draw the elevations (front and rear views) using the floor plan you have drawn.

🔁 Share / Save 🖪 😏 🖻 ...

### Take the Lesson 2-1 Quiz

Which one is NOT a vaild type of unit?

Architectural			
Decimal			
Open			



### For Further Thought:

This lesson isn't really about drawing floor plans. If you are a mechanical or civil drafter you may never draw a floor plan in your life. This lesson is about how to look at a larger project and develop a process to complete it.

In the first level of tutorials, I showed you the tools and procedures to make "lines and arcs". This level is about becoming productive and figuring out when to use which tool. This is what will make you a better drafter.

I hope that you'll reflect back to this tutorial when you have to tackle a new large project.

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## **CONTINUE TO THE NEXT LESSON: 2-2**



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### Topics covered in this Lesson:

#### Blocks in AutoCAD | <u>Dynamic Blocks</u> | <u>Write Block</u> | <u>Redefining</u> <u>Blocks</u> | <u>Exploding Blocks</u>

In this lesson you will be introduced to blocks. By definition, a block is a collection of objects (lines, arcs, circles, text, etc.) that form a more complex entity that normally represents an object in the real world, e.g. a door, a chair, a window, a computer.

#### There are many advantages to using blocks, here the major ones:

**Blocks are a single entity.** This means that you can modify (move, copy, rotate) a block by selecting only one object in it.

You can build up a library of blocks consisting of the parts that you require many times in your workday. These blocks can be stored in a separate folder and even on a network so that all drafters have access to them. Think of plumbing parts, valves, elbows, etc as well as electrical symbols or furniture.

**Using blocks can help keep your file size down.** AutoCAD stores block definitions in its database. When you insert a block, AutoCAD only stores the name of the block, its location (insertion point), scale and rotation. This can be very noticeable in large drawing.

**If you need to change something, you can redefine a block.** For example, you draw a chair and turn it into a block. Later, you're told that the size of the chair has changed. Since you used a block you can redefine the block and all of your chairs are updated automatically. If you had drawn (or copied) 100 chairs in your drawing, you would have to manually change each one.

**Blocks can also contain non-graphical information.** These are text objects called <u>attributes</u>. For example, you have made blocks of different chairs. You can add information to the block such manufacturer, cost, weight, etc. This information stays with the block, but can also be extracted to a database or spreadsheet. This would be useful for things such as a bill of materials. Attributes can also be visible or invisible in your drawing. Another good use of attributes could be a title block.

You can even easily add internet hyperlinks to blocks so you can connect a block to a page on a supplier's online catalog.

There are two types of blocks you can create: blocks that are **internal** to your current drawing, and those that are **external**, or saved as a separate file. To create the different types, different commands are used. Many companies use a template that will include a number of blocks for use in the project.

Here are the commands that you will need for using blocks in this lesson:

Command	Keystroke I	con	Location	Result
BLOCK	Bmake / B 🗔	Create	Home > Block > Create	Creates a block from separate entities (internal to current drawing)
WRITE BLOCK	Wblock / W	lone	None	Creates a block and writes it to a file (external)
INSERT	Insert / [ I ]	insert	Home > Block > Insert	Inserts a block (internal or external)
EXPLODE	Explode / X	4	Home > Modify > Explode	Explodes a block or other compound object into its component parts

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For this assignment, you will be using any one of the floor plans you drew earlier in **Lesson 2-1**.

Open the drawing.

**Zoom** in to one section of the room close to a desk (Draw a rectangle to represent a desk if you don't have any in your drawing.).

Create a new layer called COMPUTERS and make the color #73 (remember LA invokes the Layer Properties Manager).

Make the Zero Layer current.

#### ZERO LAYER :

Zero Layer has special properties. When creating blocks, if the objects in the block are drawn on Zero layer, they will assume the properties of the current layer when they are inserted. Example: If you draw the computer below on Zero layer, and insert it on the 'COMPUTERS' layer, it will assume the color, linetype and lineweight of the Computers layer. If you drew and created it on the 'DOOR' layer, and inserted it in the Computers layer, it would retain the properties of the Door layer. For this reason, blocks are drawn on the Zero layer - you need them to assume the layer's properties, whether it is in your template, or a client's.

Draw the following objects to create what you need for the computer block (top view of keyboard and monitor) You do not need to dimension the computer. **Click here to view the metric version.** 



Start the **BLOCK** command by either typing **B** or using the pull down menu or the icon. You will see a dialog box that looks like the one shown. Enter information the

same way though. Remember to approach all new dialog boxes from the top and work your way down.

A Block Definition				
Name: COMPUTER				
Base point	Objects	Behavior		
Specify On-screen	Specify On-screen	Annotative		
Pick point 2 3	-↔ Select objects	Match block orientation to layout		
<u>X</u> : 12'	© <u>R</u> etain	Scale uniformly		
<u>Y</u> : 16'-6"	Onvert to block	Allow exploding		
<u>Z</u> : 0"	© <u>D</u> elete			
	9 objects selected			
Settings	Description			
Block <u>u</u> nit:	Standard Block to use for Computers in school drawings.			
Inches 🗸 🗸	6			
Hyperlink		-		
Open in block editor     Unchecked     OK     Cancel     Help				

**1**: The first thing that you want to do is give your block a name.

Type **COMPUTER** in the edit box beside Name. Some names may need to be more descriptive, such as part number, or size.

2: Now you need to select an insertion/base point. Pick the **Pick Point** button and then pick the midpoint of the bottom line of the keyboard. Make sure that the retain button is selected (this will keep your objects on the screen as individual objects. (You will see in a moment that selecting the Pick Point with blocks is very important when you later insert them into the drawing - always pick a point that will allow you to place the block easily.) If you don't select a base point, your block will default to 0,0,0 and you will insert all your blocks at the same location - the origin. (This is where many students find their blocks when they tell me "It didn't work!")

**3**: Next you want to select the objects for your block. Pick the **Select Objects** button and then select all the parts of your computer and press **<ENTER>**. Be careful not to select any other objects, or you'll just get to do it over again.

4: Select the drawing units you used to create the original objects in.

**5**: This is optional, but you can add a description here. This is good if you are creating specific parts, like maybe a motor and want to add some quick specifications. It's also great if co-workers know what the block is used for (more information = better).

**6:** It's usually a good idea to give a block a short description and it's a great idea if you think other people will be using this block. Remember - more information is always better. Your job as a CAD drafter is to convey information to other people.

7: Uncheck the "Open in Block Editor" checkbox if it is currently checked.

**8:** Pick the OK Button and the dialog box closes. It will look like nothing happened, but the drawing file now has a "Block Definition" for a Computer in it. Congratulations, you have created your first block.

Now select any object in the new block and you will see that all of the objects are selected and the basepoint you picked is highlighted as it's (Insertion Point) Osnap.



Now that you have created a block, it's time to add another one to your drawing by **inserting** it.

Change to the Computer Layer. Start the Insert command by typing I **<ENTER>**. You will see this dialog box on the screen:

🔺 Insert		<b>X</b>
Name: COMPUTER	▼ <u>B</u> rowse	
Path:		
Locate using <u>G</u> eographic Da	ata	
Insertion point Specify On-screen	Scale Sp <u>e</u> cify On-screen	Rotation Spe <u>c</u> ify On-screen
<u>×:</u> 0"	<u>X</u> : 1.0000	<u>A</u> ngle: 0
<u>⊻</u> : <b>О</b> "	<u> </u> ビー 1.0000	Block Unit
<u>∠</u> : <b>0</b> "	⊑: 1.0000	Unit: Inches
	<u>√</u> <u>U</u> niform Scale	Factor: 1.0000
Explo <u>d</u> e	ОК	Cancel <u>H</u> elp

By default, all the options you need are pre-selected. Since you only have one block in your drawing, its name is displayed (in some drawings, you might have a long list of block names to choose from).

Make sure that the **Insertion Point - Specify On-screen** box **is** checked, and the **Explode** button is **not** checked. The Scale **- Specify On-screen** should not be checked. Then press the **OK** button.

You now have the block attached to your cursor and it is ready to be placed in the drawing. Click on one of the tables in your drawing. Notice how the block that you drew on the white Zero Layer is now Red and on the Computer layer.

Now insert a computer on every desk in your drawing. You can also copy the block instead of re-inserting each time, but make sure you know how to insert.

**Extra Practice:** Find something in the room where you are and measure it (approximate is fine) and make a block out it.

:Take a moment to think about what you just did. You drew a bunch of lines and turned them into a digital comp but the power of blocks is immense. In any line of CAD work, you'll need to be fluent in blocks.

### Video: Creating Blocks

### DYNAMIC BLOCKS

In AutoCAD 2006, Dynamic Blocks were introduced. These are parametric blocks which can be easily modified by the user. In this exercise, you will create and insert a simple Dynamic Block. The goal is to create a Chair block that can be easily rotated to orientate it to a desk.

Draw the Chair below on the Zero layer.


Start the same **Block** command you used last time, only this time make sure that the "**Open in Block Editor**" checkbox is checked. This time when you select OK, you will be magically taken to the Block Editor (see below). If you didn't select the box, you can just select the block, then right-click and choose "Block Editor".



Now that you in the block editor, you can edit the objects in your block, or in this exercise add dynamic interaction to your block. As mentioned earlier, our goal is to create a block that we can easily rotate. First you will need to add a parameter to the block, followed by an action.

Click on the "Rotation Parameter" icon in the palette. Then check the command line for prompts (as usual). Use the entries below:

```
Command: _BParameter Rotation
Specify base point or
[Name/Label/Chain/Description/Palette/Value set]:<select
midpoint of the seat>
```

#### Specify radius of parameter: 9 Specify default rotation angle or [Base angle] <0>:<ENTER>

What you have done is selected the location of the parameter, then set the radius of the parameter (in this case, where you will have a 'grip' to rotate it, and finally the default setting for the parameter - 0° means the block will look like you drew it when inserted.



After defining the **parameter**, you then need to apply an **action** to the parameter. Select the tab on the palette that says "Actions". You will need to select the "Rotate Action" (makes sense). You will be asked to select the parameter that you want to apply the action to (select the parameter you just drew). Finally, select the location of the Action )- once again select the center of the seat, and then the objects that you want the action to act on (in this case, all of them). Your block should now look like this:



Now that the Block is complete, you can select the "Close Block Editor" at the top of the drawing screen. You will be returned to the regular drawing screen, and your block will be created. Click on it, and you should see all the objects highlight, and the grip for the block's Pick Point. You will also see a grip for the block's dynamic rotation parameter.



Now that your Dynamic Block is in your drawing, pick on the rotation parameter, and move it around. You'll see that you can rotate the chair without using the rotate command. Features like this can be a real time saver. Think of other blocks that could benefit from parametric abilities : windows, doors, ceiling lights - the list is endless. Also, when you work on a new drawing from someone else, be aware the Dynamic Blocks could be present.

#### Video: Dynamic Blocks (Part 1)

#### Video: Dynamic Blocks (Part 2)

This exercise has shown the basic steps that are used in creating a Dynamic Block. There are many other parameters and uses, and they can make your CAD life much easier. Check out the next video to see some more options in how these are created.

#### WRITING BLOCKS

This time you will be creating an **external block** using the **wblock** command. This difference here is that the block will become a separate, external drawing file for use in other CAD drawings.

In the dialog box below, you will see that you have almost all the same options. Instead of giving the block a name like you did before, you give it a filename in a specific folder.

🛦 Write Block	×				
Source         CHAIR7           ○ Entire drawing         ○           ○ Objects         Base point           ○ Pick point         ↓:           ✓:         765.0432094561274           ✓:         1065.621578030983           ∠:         0.0000	<ul> <li>Objects</li> <li>→ Select objects</li> <li>→ Select objects</li> <li>→ Select objects</li> <li>→ Delete from drawing</li> <li>1 object selected</li> </ul>				
Destination <u>File name and path:</u> C:\CAD-Blocks\ROTATING-CHAIR.dwg Insert <u>u</u> nits: Unitless					
OK Cancel <u>H</u> elp					

After creating the other blocks above, you should be able to work your way through this dialog box. Make sure you put the block in a logical path and give it a good, descriptive name.

When you insert an external block, use the same **INSERT** command that you did above and use the Browse button to navigate to the folder where you stored your block. Insert it like you did before. Put some chairs in front of the desks in your drawings, and rotate them if needed.

**NOTE:** For some reason, Autodesk won't allow us to use the write block command to create Dynamic Blocks. Perhaps this will be added in a later version. You can always create a drawing with just the block you want separate and then save the file.

Now you have created three blocks. The process is the same for any other block that you need to create from drawing objects.

If you want more practice, draw more objects and create blocks from their geometry.

### Video: Writing Blocks (WBLOCK)

#### **EXPLODING BLOCKS**

Now that you know how useful block are, you should know that there are times when you need to explode a block. The **EXPLODE** command works on blocks, rectangles and other objects that are not the basic lines, arcs, circles, etc. If you have a block on your screen, type **X <ENTER>** to start the explode command. Select the block that you want to explode and press **<ENTER>**. Now you have all of the components that made up the block as individual objects.

#### **REDEFINING BLOCKS**

If you want to see how to update a block, explode and modify the computer block by stretching the monitor out by one inch on each side. Create a new block using the same COMPUTER name. After re-defining the block, you will see this box pop up to warn that you are about to update the block definition:



Select Yes, and all your 'computer' blocks in the drawing will update to the new definition.

This is a good example of how blocks save you time. In a typical work situation, the original specifications for the computer could have changed from large CRT monitors to flat screens and you would need to change them all. Using blocks is much easier than changing each individual object one at a time.

**Conclusion:** Remember that blocks are powerful tools for the <u>reasons</u> listed above. In any discipline of CAD drafting, you will use them a lot. Usually, you will insert the block first and then copy then to other locations. They are powerful, yet easy to work with. Remember, Create the Block, Insert it - it's that simple (In most cases, you will use the blocks in a template to save time). When creating a block, your choice is if you want an external, internal or dynamic block. That choice depends entirely on how that block will be used.

#### **ONE LAST TIP**

There are times when you might name a block and decide later that you want to change it. Maybe you designed a bracket and now that the design is finalized, it is given a part number (or maybe you noticed a spelling mistake). Now you want to change the name of the bracket from "BRACKET-01" to "BRACKET-BR734". The easiest way to do this is to use the **RENAME** command. This will open a dialog box that allows you rename a number of AutoCAD objects.

👗 Rename	
Named Objects	<u>I</u> tems
Blocks Detail view styles Dimension styles Layers Linetypes Materials Multileader styles Section view styles Table styles Text styles UCSs Viewports Views	CW3030 CW3036 DESK2 DESK3 DESK4 DIGITIZE DISHWASH DOOR DOOR2 DOOR2 DOOR3 FC15X27A FC42X18D FLATFILE FNCASE3 FNCASE4
<u>O</u> ld Name:	DESK2
Rename To:	4-FOOT-DESK
	OK Cancel <u>H</u> elp

This command is simple to use. Select the type of object first in the left column, and it will show you all the different items in the right column. Make sure that the 'Old Name' field is filled, and then type the new name in the box below it. Check your edits and press OK.

This command can be used for changing layer names, linetypes and most other AutoCAD objects.



Take the Lesson 2-2 Quiz

What is an advantage to using blocks in your drawing?

They are easier to modify

All

It keeps file size down



## For Further Thought:

Here you have learned about one of AutoCAD's most powerful and commonly used tools. Do you realize how much time and effort you can save by using blocks? I've given a few examples in the tutorial - but try to think of more.

If you are not fully aware and comfortable with how blocks work after going through this tutorial, I ask you to work through it again. Working with blocks should be as easy as drawing a line.

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# **CONTINUE TO THE NEXT LESSON: 2-3**



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#### Topics covered in this Lesson:

#### Working with real world examples

This tutorial will give you some drawing practice using real world examples. You will be asked to draw the sample drawing using some techniques you haven't been shown yet. This lesson requires you to have Adobe's <u>Acrobat reader</u> for PDF files. Here I'm using an example of someone who has to measure and draw a room.

#### Example 1 - Loading Dock

This is a real example from when I worked preparing 'as-built' CAD drawings. These drawings are used by designers who need to know the existing dimensions of the building that will be remodeled. When you are doing a survey and measuring rooms, it goes quickly as long as you have nice rectangle walls which the contractors built perfectly square, and you don't have to deal with angles or other problems. This example will deal with drawing a room that has angles. Below is the room you will be drawing, notice that some of the walls are at an angle, and you don't know those angles.

Here is the **<u>drawing</u>** with all the dimensions on it (print it out if you can). Imagine that you have just sketched out all the measurements and you need to draw it now. How

do you start? The best way to start is to draw what you can - in this case, you can draw the horizontal and vertical lines easily. Next draw the lines for the angles lines, but draw them horizontal and vertical as well (as shown below):

Now you should have all the lines drawn, and the door blocks inserted or drawn in. Make sure that everything is on the correct layer.

Finally you will adjust the 'soon to be angled lines' using the Align command.

Make sure your OSnaps are turned on or your drawing will become a mess rather quickly.

Command	Keystroke	lcon	Location	Result
ALIGN	ALIGN		Home > Modify > Align	Aligns (and scales) selected objects to specific points.

This command requires that you follow the command line prompts carefully. Start the command and use the input shown below. You are selecting source points (where it starts) and destination points (where it will end up). Choose the endpoints nearest the text.

Select objects: <SELECT BOTTOM AND RIGHT SIDE LINES> Specify opposite corner: 28 found Select objects: <ENTER> Specify first source point: <1S> Specify first destination point: <1D> Specify second source point: <2S> Specify second destination point: <2D> Specify third source point or <continue>: <ENTER>

Command: ALIGN

#### Scale objects based on alignment points? [Yes/No] <N>: N

Just so you know, the 3rd source point is used in 3D drawing.

If everything went correctly, your lines should be very close, but not touching. Zoom in if you can't see this. In the real world, your lines won't usually line up 'exactly'. When that happens, you'll need to re-check your measurements, and correct the lines (but in some cases, close will work - it all depends upon your tolerances for the project). Also, if you have drawn your lines to specific lengths, you don't want to scale them.



But what if you have an older version of AutoCAD LT. Prior to AutoCAD LT 2015, it doesn't have the align command. You're screwed, right? Not really - you can use the **ROTATE** command. This is the same rotate command you used in Level 1 and it works in all versions of AutoCAD.

Undo the align command so that your lines are back to horizontal and vertical.

Here's how you can use the rotate command to to rotate objects at an angle you don't know. Once again, follow the command line prompts carefully. What you need to do is tell AutoCAD what the existing angle is (Reference angle) and what the new angle is.

Command: ROTATE Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0.0000

Select objects: <SELECT OBJECTS> Specify opposite corner: 28 found

Select objects: <ENTER>
Specify base point: <1R>
Specify rotation angle or [Copy/Reference] <329.8507>: R
Specify the reference angle <0.0000>: <1R> Specify second
point: <2R>
Specify the new angle or [Points] <0.0000>: <N>

Both commands work well and quickly. Which one you use depend on the situation. If your lines are connected like they were in this example, the Rotate command works fine. If they aren't then the align command works easier. Personally, I use the Align command each time for consistency.

To clean up the walls, use the **FILLET** command with a Zero radius.

#### Video: Align and Rotate Reference Commands

#### **Extra Practice:**

For this exercise, you will draw a room and place it into another drawing. Then you align the room to the existing walls. This is a common of collaborating on a drawing. A project manager might have the complete floor plan and delegate smaller areas to be drawn. Once the smaller area is complete, it needs to be inserted into the master drawing.

This can be done a few ways. You can copy and past the objects into the master drawing (good for small drawings). You can WBLOCK out the drawing objects and then insert them as a block. Finally, you use the smaller drawing as an External Reference (XREF) - this is covered in a later tutorial. For now, just use the copy and past method to complete this exercise.

Complete this floor plan drawing: <u>ALIGN EXERCISE</u> (PDF file)

When you are done, download and open the MASTER FILE (DWG)

With both drawings open, use the Windows Copy (CTRL+C) to copy the objects you drew to the clipboard. Then switch to the Master Drawing and paste them into the drawing (CTRL+V). Place them somewhere near the walls in the Master Drawing.

Finally use the ALIGN command to line the room your drew to the walls in the Master Drawing and you are done. Check out the video below if you need help with this.

### Video: Align Exercise

**Review:** The real world isn't all straight lines and right angles. Knowing how to draw lines at strange and weird angles can be a challenge. Accuracy is always the priority, but quickness is a close second. Remembering all the commands you have learned (even if you don't use them all the time) is essential to being a good CAD drafter.

🕒 Share / Save 🖪 🈏 🖻 ...

#### Take the Lesson 2-3 Quiz

Does the Align command work in AutoCAD LT?

Only in later versions

Yes

No



For Further Thought:

There's always room in your toolbox for new tools, right? The align command might not be a tool that you use often, but you'll be very glad it's there when you need it. You learned two similar ways to do the same thing in this lesson, so it's up to you to decide when you want to use one instead of the other. Learning these differences is what will make you faster and more comfortable when you draft. Luckily, there is never a single 'correct' way to use AutoCAD - but you will develop ways that work best for you and the type of drawings you create.

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# **CONTINUE TO THE NEXT LESSON: 2-4**



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## Topics covered in this Lesson:

#### Adding Attributes | Editing Block Attributes | Extracting Attributes

So far in using AutoCAD you have created geometry: lines, circles, etc. You have also added things such as text and dimensions. All of these things could also be done by hand, so what else does CAD have to offer? As noted at the beginning of Level 1, AutoCAD is also a database of information. Most of that database contains the information for reproducing what you have drawn, but you can also add information that is *non-graphical*. One of the easiest ways of adding non-graphical information is to use attributes. An attribute is text that can be attached to a block that conveys more information than just the geometry on its own could convey.

Look at the two drawings below for an example; the first example shows some lines and arcs that could be anything.



The second example shows the same geometry with the attributes visible so that you see what the shapes represent.

ACME FURN. CO.	
ACME FURN. CO. ABC-123 GREEN \$2397.99	

The pictures above show a couch. The attributes describe what the model number, color and cost are as well as the manufacturer. From this simple example, you can see that AutoCAD has a useful tool for showing more than just geometry.

This example shows information about furnishings in an office. Once you have drawn the floor plan, you could insert blocks of furniture that have information about the manufacturer, price, weight, and any other information you may need. This information can then be extracted out of AutoCAD and then used in a spreadsheet or other program which could generate a Bill of Materials (BOM) or a Request for Quote (RFQ). Although you might not normally use the more advanced features of Attributes in your daily work, it's good to know how they work.

In this lesson you will be creating attributes and attaching them to the computer block that you made previously. Like everything you do in AutoCAD, there are particular steps involved when you work with attributes.

First you have to define (or create) the attribute and draw the geometry.

Next you will create the block with attributes.

Finally, when you insert the block you will give it the specific information

Many of the commands that you will be using are tough to remember at first. You may find it easier to use icons (but then the icons look very similar). Pull down menus are another option, so you can see exactly what the command is that you are starting.

Command	Keystroke	lcon	Location	Result
Define an attribute	ATTDEF / ATT	Ø	Home > Block > Define Attribute	Creates an attribute definition
Edit attributes	DDATTE / ATE	10	Home > Block > Edit Attributes	Edits the contents of an existing attribute
Display Attributes	ATTDISP	\$	Home > Block > Retain Display	Hides or shows attributes
Extract Attributes	EATTEXT		Insert > Linking & Extraction > Extract Data	Extracts attributes using the wizard

Open one of the floor plans you drew earlier with the Computer blocks in <u>Lesson 2-</u> <u>2</u>.

Insert a computer block and then explode it (type **X** <**ENTER**> and then select the block - press <**ENTER**>.)

Begin the Attribute definition command: ATT(or) ATTDEF (or) DDATTDEF

(AutoCAD has kept the commands from previous versions active in the new versions.)

Look at the dialog box below. Fill in the edit boxes just like the example. Make sure to adjust the text height if necessary.

Attribute Definition		×
Mode Invisible Constant Verify Preset	Attribute <u>T</u> ag: Pro <u>m</u> pt: Defau <u>l</u> t:	CPU What is the CPU Speed? X.XX GHz
Lock position   Multiple lines     Insertion Point   Specify on-screen     X:	Text Settings Justification: Text style:	Left   Standard
Y:     0.0000       Z:     0.0000	Text h <u>eig</u> ht: <u>R</u> otation: Boundary <u>w</u> idt	1 ↔ 0 ↔
Align below previous attribute	e definition	Cancel <u>H</u> elp

What you did is give the attribute its definition: a **name** (tag), a **prompt** to help the user while they're inserting it, and a (default) **value** for the prompt. Once everything is entered, select the OK button.

When you are back on the drawing screen, pick a point to the right of the keyboard on the computer block. When you are done, the dialog box comes back so press **OK**.

Add the following attributes the same way, except instead of picking a point, check off the **Align below previous attribute** checkbox.

TAG	PROMPT	VALUE
MONITOR	What is the MONITOR SIZE?	XX"
HDD	What capacity is the Hard Drive?	X.XX GB
RAM	How much RAM is installed?	XX GB

STATION What station is this XXX	STATION Wh	at station is this	XXX
----------------------------------	------------	--------------------	-----

Once all the attributes are created you should have something like this:



Next you will create a block that includes your four attributes.

Start up the **BLOCK** command. Create it as shown in a **previous lesson (2-2)**. When you select the Attributes, select them individually from top to bottom. When you are asked for the block name, give it the name **COMPUTER-WITH-ATTRIBUTES**.

Insert the block and you will get a dialog box where you can enter the values for the tags. You will be prompted to provide answers to the prompts that you defined in the attributes.

🛦 Edit Attributes	×
Block name: Computer with Attributes	
What is the CPU Speed	3.4 GHz
What is the monitor size	24 "
What capacity is the Hard Drive	750 Gb
How much RAM is installed	8 Gb
What station is this	108
OK Cancel Pre	evious <u>N</u> ext <u>H</u> elp

After inserting the block and answering all the prompts, your block should look like this:



Now by looking at the drawing, you can see exactly what type of computer it is and what station (location) it is. This is just one application, but you can see how it can be used in any discipline of drafting. You can either insert more blocks, or copy the one that is in the drawing.

## **Editing Attributes**

Of course, in any project, the data is subject to change. In this example, the company may decide that it cannot afford 24" monitors. You can easily change the value of attributes within a single block insertion by double-clicking on the block. That will bring up this dialog box:

		Enhanc	ed Attri	bute Editor		×
A DESCRIPTION OF THE OWNER OWNER OF THE OWNER OWNER OF THE OWNER OWNE		Block Tag	Select block			
	At	ttribute	Text Opt	ions Properties		
		Tag		Prompt	Value	*
	MONITOR		TOR	What is the monitor s	24 "	
		HDD		What capacity is the	750 Gb	E
		RAM		How much RAM is in	8 Gb	
		STAT	ON	What station is this	108	~
	L	1	Value: 2	1"		
		<u>A</u> pply	,	ОК	Cancel	<u>H</u> elp

Remember that this will **only edit the one block** that you double-clicked on. If you want to change more than one block at a time to SAME values, you can select the blocks, then select your blocks, right-click and choose Properties. Look for the Tag value you want to change and remember you are changing all values to the same one you input now.

To clean up the walls, use the **FILLET** command with a Zero radius.

## Video: Creating Attributes in AutoCAD

## DISPLAYING ATTRIBUTES

Sometimes you don't want to see the attribute values displayed (say for plotting). You can turn them off. This can be done by typing in the command **ATTDISP** and then **OFF**. To turn them back on again, type **ATTDISP** and **ON**. It can't get much easier. You can also navigate the ribbon to **Home > Block > Attribute Displa**y and choose from these buttons.



# EXTRACTING ATTRIBUTE DATA IN AUTOCAD

So now you're wondering what can you do with all of this information. A CAD drawing can be used to track inventory and export that data to another file (such as Microsoft Excel - a spreadsheet application) for others to use. Maybe the purchasing department wants to know what to order, based on the design you are working on.

Exporting the data became MUCH easier in recent versions of AutoCAD.

Insert 6-12 of the blocks of the computer you made with Attributes and fill out the values.

Start up the **EATTEXT** command and you will see the Data Extraction wizard dialog box appear. Accept the default setting of "Create a new data extraction" and press Next. You should be prompted to save your "Data extraction files as...". Select a folder and name the file "attributes\_001". This will take you to step 2 of 8. This is where you tell AutoCAD what drawings you want to extract data from.

A Data Extraction - Define Data Source (Page 2 of 8)	
Data source ◎ Drawings/Sheet set ✓ Include current drawing ● Select objects in the current drawing	
Drawing files and folders:  Drawings E:\we-R-here-FULL-BACKUP\cad-16\tutorials\evel_2\inks\comproom.dwg (Current drav	Add Folder Add Drawings Remove
<pre></pre>	Settings Next > Cancel

You will have the option here to add more drawings (not in this example), but make sure that "Select objects in current drawing" is selected, and then use the icon to select the blocks that you want to include in this extraction. When you are done, press Next.

Now you need to tell AutoCAD what objects in the drawing you want to include in the extraction.

A Data Extraction - Select Objects (P	age 3 of 8)		
Select the objects to extract data from: Objects			Preview
Object 🔺 Display Nam	e	Туре	
Computer with Att Computer wit	h Attributes	Block	
Display options			
Display all object types	Display blocks with <u>a</u> ttributes only		
Oisplay blocks only	Display objects currently in-use only		
Display non-blocks only			
		< <u>B</u> ack <u>N</u> ext >	> <u>C</u> ancel

Check the box for "Display blocks with attributes only" and you should see that the only item in the list is your Computer-with-Atributes block. Press Next.

This takes you to the next stage where you can narrow down the extraction data even further. Check only the box for "Attribute" in the Category Filter. Press Next.

The fo	ta Extraction - Sele following properties v of the properties you ore the right-click m	vere found based o	n the objects you selected.		
	Property A CPU HDD MONITOR RAM STATION	Display Name CPU HDD MONITOR RAM STATION	Category Attribute Attribute Attribute Attribute Attribute		Category filter          3D Visualization         Attribute         Drawing         General         Geometry         Misc
			<	Back	Next > Cancel

So far in each step you have narrowed the criteria for extraction from which drawings, to which blocks, and now you can select which attributes you want to extract. Press Next.

This step allows you to further filter and organize the data you want. You should have 2 extra columns than what is shown above - right click on the extra column and select "Hide Column". Next, select the column header for "Station" and drag it over to the left (this is to define the order of display which you'll see soon). Finally sort the Station column by clicking on the column header. Your dialog box should look like the one above now. Press Next.

Data Extraction - Refine Data (Page 5 of 8)								
links.								
STATION 🔺	CPU	HDD	MONITOR	RAM				
108	3.4 GHz	500 Gb	24"	8 Gb				
109	3.4 GHz	750 Gb	24''	8 Gb				
110	3.4 GHz	750 Gb	21"	8 Gb				
111	2.4 GHz	750 Gb	24"	8 Gb				
112	3.4 GHz	750 Gb	24"	2 Gb				
113	3.4 GHz	500 Gb	24"	4 Gb				
Combine identical I	rows			🐻 Link E	xternal Data			
Show count column	1			ALS	alaana Oofiaaa			
Show name column	,			Z‡ Son C	olumns Options			
E Show hame could in the could in the could be a could be could be could be a could be a								
< Back Next > Ca								

On the next step, select both boxes. This will allow you to save the data to an external file and also place the data in your current drawing. Use the box shown below to select a location for the external file. Press Next.

A Data Extraction - Choose Output (Page 6 of 8)	
Output options Select the output types for this extraction: Insert data extraction table into drawing Output data to external file (.xls .csv .mdb .txt) C:\Users\Me\Desktop\comproom xls	
	< Back Next > Cancel

Step 7 allows you to format the table that will be created in the drawing. Table are covered in **another tutorial**. Add a title for your drawing called "COMPUTERS". Select the box for "Use property names as additional column headers". Press Next.

A Data Extraction - Table Style (Page 7 of 8)			
Table style Select the table style to use for the inserted table:	Title		
Standard 🔻	Header	Header	
Formatting and structure     O Use table in table style for label rows	Data	Data	
<ul> <li>Manually setup table</li> <li>Enter a title for your table:</li> </ul>	Data	Data	
COMPUTERS	Data	Data	
Title cell style: Title   Header cell style: Header	Data	Data	
Data cell style: Data 💌	Data	Data	
✓ Use property names as additional column headers	Data	Data	
	L	·]	
	< Back	Next > Cancel	

Read the instructions on the final step. In this example you chose to include a table so you will be prompted to select an insertion point after you click Finish. Press Finish, and your insertion point and you should have a table that looks like the one below. Note that the table is formatted like you wanted it with the Stations on the left and sorted numerically.

COMPUTERS								
STATION	CPU	HDD	MONITOR	RAM				
108	3.4 GHz	500 Gb	24"	8 Gb				
109	3.4 GHz	750 Gb	24"	8 Gb				
110	3.4 GHz	750 Gb	21"	8 Gb				
111	2.4 GHz	750 Gb	24"	8 Gb				
112	3.4 GHz	750 Gb	24"	2 Gb				
113	3.4 GHz	500 Gb	24"	4 Gb				

You also have a table that is still linked to the blocks in your drawing. Edit the attributes in a few of your blocks. Then right-click select a area in the table (not text) and select "**Data Extraction > Update Data Extraction**". Deselect your table and you will see that it is now updated with the most recent block data. Powerful stuff!

For more about tables in AutoCAD, please refer to **<u>Tutorial 4-9</u>**.

	Cut	-		C	D		E		
	Copy Paste			OMPUTERS					
	Paste Recent Input	•		HDD	MONITOR	R	АМ		
	Cell Style	+	Ιz	500 Gb	24"	8	Gb		
	Background Fill Alignment	•	Ιz	750 Gb	24"	4	Gb		
Ξ.	Borders	,	Ιz	750 Gb	21"	8	Gb		
	Locking	•	ĪZ	750 Gb	24"	8	Gb		
	Data Format Match Cell		Ιz	750 Gb	24"	2	Gb		
	Remove All Property Overrides		z	500 Gb	24"	4	Gb		
	Data Extraction	×	l	Jpdate Data Extraction	1 I	1			
	Insert	+	E	dit Data Extraction Se	ttings				
	Edit Text		[	Detach Data Link					
	Manage Content	L							
	Delete Content	+							
	Delete All Contents								

You also have an Excel file on your computer showing the data from the drawing. Open it if you have Excel or another spreadsheet program and you should see something like this:

	М	icrosoft Exce	el - comproc	om.xls			x
	)	<u>F</u> ile <u>E</u> dit	<u>V</u> iew <u>I</u> ns	sert F <u>o</u> rma	at <u>T</u> ools	<u>D</u> ata <u>W</u> in	ndow
li F	<u>l</u> e	lp Ado <u>b</u> e	PDF				- 8 ×
1	3	10	-   B   ≣	≣ ₽=	🔛 🗕 🌺 🗸		
		A1	<b>▼</b> ;	🛿 'STATIO	N		
		A	В	С	D	E	F
1		STATION	CPU	HDD	MONITOP	RAM	Â
2		108	3.4 GHz	500 Gb	24"	8 Gb	=
3		109	3.4 GHz	750 Gb	24"	8 Gb	=
4		110	3.4 GHz	750 Gb	21"	8 Gb	
5		111	2.4 GHz	750 Gb	24"	8 Gb	
6		112	3.4 GHz	750 Gb	24"	2 Gb	
7		113	3.4 GHz	500 Gb	24"	4 Gb	
8							
9			_				Ψ.
•	•	► • •  \ Sur	nmary /		_ <b>(</b>	11	the 📕 📲
Rea	ady	/			N	UM	H.

#### Video: Extracting Attribute Data

This is just a simple walkthrough of some of the power of AutoCAD. With more complex drawings and attributes, you can work wonders. Think about how easy it would be to build a table of all windows in a house plan if the windows were created as blocks with attributes. Some assembly drawings consist of nothing but standard blocks, this lesson shows how much this simple process of creating attributes would help the workflow. Attributes are powerful - remember to use them.

In everyday work, your exposure to attributes may be minimal. Still you might find yourself inserting blocks and entering some data, and now thanks to this exercise, you know the power of attributes. Get to know how this works and you can really impress your boss!

**Extra Practice:** Here is an exercise for creating a title block with attributes. This is a handy item to have. It allows you to quickly enter the data for the title block using attributes, rather than create text every time. <u>Link to the exercise</u>. You could then

use the information in the Title Blocks to create a file showing information about all the drawings on your computer by extracting the data.



## Take the Lesson 2-4 Quiz

Which item isn't part of an attribute?

Prompt Scale Tag

## For Further Thought:

When you drew your first line in AutoCAD did you think that you would soon be learning about data extraction? To me, this is the true power of AutoCAD. If you think of the tools in your toolbox - consider Attributes a 'power tool'. If you can use these in your daily drafting, you'll be valuable to your company. Depending on where you work, you might not use Attributes often, but never forget how they work and how powerful they can be. Any time you need to track inventory - you can use attributes. Any time you need to draw a date table, you can use attributes.

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# **CONTINUE TO THE NEXT LESSON: 2-5**



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Last Update: 02/25/2019 18:16:25

#### Topics covered in this Lesson:

Hatch, Hatch Edit, Gradient, Boundary, Solid

Hatching in AutoCAD is a way of filling in areas of your drawing with a pattern that represents certain materials. It is usually used in <u>sectional views</u>. Starting with AutoCAD release 14, you can use a solid fill to completely fill in areas such as walls in a floor plan.

Command	Keystroke	lcon	Location	Result
Boundary Hatch	Bhatch / H	Hatch	Home > Draw > Hatch	Covers an area with a predefined pattern
Hatch Edit	HatchEdit / HE	<b>1</b>	Home > Modify > Edit Hatch	Edits an existing Hatch
Gradient	Gradient / GRA	Gradient	Home > Draw > Gradient	Creates a gradient hatch pattern
Boundary	Boundary / BO	Boundary	Home > Draw > Boundary	Creates a polyline or region from enclosed area.
Solid	Solid / SO	None	None	Creates a solid filled shape by picking 3 or 4 points

Draw a rectangle 10 by 10 and put a circle with 2 radius in the middle of it (remember the M2P OSnap?).

Start the Hatch command by typing **H <ENTER>**. When you start the command, you will see a prompt on the command line (shown below). Press **T <ENTER>** to select the **settings** option to get the dialog box for the hatch and gradient settings.

```
Command: H <ENTER> HATCH
Pick internal point or [Select objects/Undo/seTtings]: T
<ENTER>
```

A Hatch and Gradient	X
Hatch Gradient	Boundaries
Type and pattern	Add: Pick points
Type: Predefined ▼	Add: Select objects
<u>P</u> attem: STEEL ▼	
Color: ByLayer	Remove boun <u>d</u> aries
Swatch:	<u>R</u> ecreate boundary
Custo <u>m</u> pattern:	View Selections
Angle and scale	Options
Angle: <u>S</u> cale:	Annotative
0 • 6 •	✓ Associative
Double Relative to paper space	Create separate <u>h</u> atches
Spaging: 1.0000	Dra <u>w</u> order:
IS <u>0</u> pen width:	Send Behind Boundary
Hatch origin	Layer: Use Current
Use current origin	<u>Transparency:</u>
Specified origin	ByLayer
Click to set new origin	0
Default to boundary extents	
Bottom left	Inherit Properties
Store as de <u>f</u> ault origin	
Preview	Cancel Help 🕥

As usual, start at the top of the dialog box and work your way down. We're going to say that this is a cross section of piece of steel with a hole drilled in it, so choose the predefined Hatch pattern called **STEEL**.

Now you want to pick the area to be hatched. **Pick** somewhere inside the rectangle, but outside of the circle and press **<ENTER>**.

Set the scale of the hatch to **6**. This is a number that works for this sized object. A larger number will make the spacing between the hatch lines bigger (maybe so big
you won't see it) and a smaller number can make the hatch so dense that it looks solid (try different numbers later to see if I'm wrong).

Make this "**Associative**" - this means that if you adjust the rectangle or circle, the hatch will automatically correct itself to the new boundary.

Finally, hit the **Preview** button to see if this is what you are after, it should match the image below.



Note that AutoCAD recognized the circle in the middle and didn't hatch over it. If the lines appear 'jagged', don't worry - it is a video display issue - the prints will come out clean and straight.

**SPECIAL NOTE:** In this exercise, you used a rectangle and a circle - both these objects are 'closed'. If you draw four lines to make a rectangle and leave a gap at one of the corners (no matter how small) the Hatch command will NOT work. All boundaries detected by the Hatch command must be closed with no gaps at all. If this happens you will see this dialog box pop up:

Hatch - Boundary Definition Error	×
A closed boundary cannot be determined.	
There might be gaps between the boundary objects, or the boundary object be outside of the display area.	ts might
Red circles are temporarily displayed to identify possible gaps in the bound These circles can be removed with the REDRAW or REGEN commands.	ary.
<ul> <li>Show details</li> <li>Do not show this message again</li> </ul>	Close

If you want to edit the hatch, the easiest way is to just double-click on it. This will bring up the Properties pallet. Try different settings in the Hatch Edit dialog box and preview the results. If you use the HatchEdit Command (HE), you will see the HatchEdit dialog box that looks like the one used in the Hatch command.

To see what "Hatch Association" is, pick on the rectangle (be careful not to select the hatch) and you'll see 4 squares appear on the corners. Click and drag one of the corners and release it. If your hatch is associative, it will update to the new shape. If you have ever done hand drafting, this feature will make you smile.

**HATCH TIPS:** For monochrome printing: If you are using a solid hatch, make the hatch WHITE so that it will print out as a solid black, any other color could appear gray (with lines) when printed.

If you have one hatch area drawn and want it to match the hatch of another, use the "Inherit Properties" button on the Hatch Edit dialog box. Try this by creating 2 different hatch styles in different objects.

#### Gradients

In recent versions of AutoCAD, you can add a 2 color gradient to an object. Draw another rectangle with a circle inside it. Type in **GRA <ENTER>** to start the Gradient command. This command works just like the Hatch command, but with different

looking results. You can select the "**T**" option for settings or use the Ribbon to edit the gradient you just created. The HatchEdit (**HE**) command also works on Gradients.

Below is a sample of a basic 2D 'rendering' using this method. See if you can recreate it using the Gradient command.



#### More about Hatching:

You can create custom hatch patterns, or find custom ones available on the internet. Other websites offer some <u>free ones</u> as well as extensive <u>collections for sale</u>.

Hatches can be exploded (x < ENTER>)- but be VERY careful if you do this. In fact, don't explode hatch patterns unless you *really* need to as you will get 100's of lines instead of 1 hatch.

There is a setting in the Options dialog box under the drafting tab that allows you to turn on or off the ability to snap to hatch objects. Off is the default.

## Boundary

Although the Boundary command does not create a fill of any kind, it works in the same manner as hatching so I am including it in this tutorial.

Start by drawing four random lines in the pattern that is shown below. The length of the lines doesn't matter (for once).



The goal of this exercise is to create a shape that matches the inside section of the four lines. This is a very simple example, but this tool can be very helpful when you need to create a complex shape quickly.

Start the Boundary command (BO <ENTER>) and you will see this dialog box:

A Boundary Creation
Pick Points
✓ Island detection
Boundary retention
✓ Retain boundaries
Object type: Polyline
Boundary set Current viewport
OK Cancel Help

This is quite straightforward, see the "Pick Points" button at the top, select it and then pick the area in the middle of your four lines and press enter. Depending upon your settings, it may look like nothing has happened. To see what you created, start the **Move** command and use the "L" option to select the last option your created and you'll see a new object be highlighted. You've just created a polyline - and these are covered in tutorial 2-7.



#### Solid

Here is another command that can sometimes be quicker than a hatch. Solid allows you draw an solid filled object using 3 or 4 points. For this example, use the same four lines that you drew for the Boundary exercise..

Start the **SOLID** command. Pick 4 corners of the rectangle and press Enter. You should end up with something like this:



Is that what you were expecting? I don't think so. This command works great when you only use 3 points. To fill this shape completely, you would need to draw two Solids using 3 points each, or just use the Hatch command.

Draw a Rectangle of any size and then start the **SOLID** command. Select 3 corners and press enter. You should see something that resembles the image below:



**Extra Practice:** Complete <u>this drawing</u> and copy the gradient hatches shown in <u>this image</u>.

**Extra Practice:** Draw <u>this shape</u> and hatch each possible section. It could look something like <u>this</u> when it is done (check that you hatched all areas and the scale is correct.

**Bonus Practice**: Here's a real world example to practice many of the skill that you have learned so far (including hatching). Duplicate this <u>drawing</u> from the National Forest Service.

## Video: Hatch and Related Commands



Take the Lesson 2-5 Quiz

Once a Hatch is drawn, can you edit it?

Yes, using the Hatchedit command

No

Only if you don't close the drawing



#### For Further Thought:

After showing you the power of blocks and attributes, I thought it would be good to slow down and show you a simpler tool. If you've already forgotten how blocks and attributes work, please go back and review those lessons.

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# **CONTINUE TO THE NEXT LESSON: 2-6**



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Last Update: 02/25/2019 18:16:42

#### Topics covered in this Lesson:

#### Formatting Text Styles | Creating Text | Editing Text

Creating text in AutoCAD is easy. You may have already done this in the first level of tutorials, but this lesson will go into more detail and explore more options. Like most things in AutoCAD, there is more than one way to do things and creating text is no exception. Below are some common commands for working with text.

Command	Keystroke	lcon	Location	Result
Text Style	STYLE / ST	A	Home > Annotation > Text Style	Opens the Text Style Dialog
Single Line Text	DTEXT / DT / TEXT	A Single Line	Home > Annotation > Single Line	Creates a single line of text
Multiline Text	MTEXT / T / MT	$A_{^{\text{Multiline Text}}}$	Home > Annotation > Multiline Text	Creates formatable Multiline, paragraph text
Edit Text	DDEDIT / DDE (or double- click)	No Icon	None	Edits and formats text - edits attributes
Spell Check	SPELL	ABC	Annotate > Text > Check Spelling	Checks for spelling errors in Text, Attributes and Xrefs
Text along an arc	ARCTEXT	A B C Arc Aligned	Express > Text > Arc- Aligned Text	Aligns test along a selected arc

#### Formatting Text Styles

When you start a new drawing and use the default acad.dwt template, you will find that there is only one text style defined. In older versions of AutoCAD, the default font was an ugly, simple "TXT.SHX" font, but newer versions will use Arial as the default font. This is a clean, easy to read font and works for most situations.

Most company template drawings will have your text styles defined. You might find text styles for regular text, title blocks, notes, dimensions and others. The number of styles will depend upon the complexity of the drawing. Generally you won't change these.

Occasionally you may need to create a new text style (a definition of the way the text will look). For example, the font that is used is defined by the Text Style. To format text properly in AutoCAD, you have to manually create a new text style for each different font and style of text. It is possible to edit and format text 'on the fly' but this is considered bad practice and leads to drawings that can look confusing, unprofessional and hard to edit. Keep your text styles organized.

Before starting, we'll just add a line of text in a new drawing that is created from the acad.dwt template file. Start the **MTEXT** command and follow along on the command line:

Command: MT <ENTER> MTEXT Current text style: "Standard" Text height: 0.2000 Annotative: No Specify first corner: <PICK ANYWHERE IN THE SCREEN> Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width/Columns]: <PICK OTHER CORNER>

This will give you a text window, so you can now type something on the screen like "DEFAULT TEXT" and then click anywhere outside of the text window to end the command.

It is easy to load in a new style for all of your text. Here's how to do it. Type **sT**, and this dialog box appears.

📥 Text Style		∑∑
Current text style: Standard <u>S</u> tyles: Annotative Standard	Font <u>Font Name:</u> <u>Fr</u> Arial <u>Use Big Font</u> Size	Font Style: Regular ▼ Set <u>C</u> urrent <u>N</u> ew <u>D</u> elete
	Annotative	Height 0.0000
All styles	Effects Upside dowr Style Name:	Custom OK
AaBb123	Backwards Vertical	Cancel
		Apply Cancel Help

Select the **New...** Button and type in a name for your new text style and press OK.

With your new style selected, click on the **Font Name** drop list and select **Times New Roman** as your new font. For Font Style, select **Bold**. This style will be noticeably different from the default Arial font. Optionally you can set the width factor to a .8 which allows you to fit text in narrower spaces at an 80% width.

🔺 Text Style			×
Current text style: Custom <u>S</u> tyles:			
Annotativo	Font <u>F</u> ont Name:	Font Style:	Set <u>C</u> urrent
Custom Standard	Times New Roman	Bold	<u>N</u> ew
	Size		Delete
	Match text orientation	Height_ 0.0000	
	to layout	0.0000	
All styles 🔹	Effects		
	Upsid <u>e</u> down	Width Factor:	
AaBb123	Backwards	1.0000 Oblique Angle:	
Aadu123	Vertical		
		<u>Apply</u>	Help

**IMPORTANT:** Do **NOT** change the Height in this dialog box unless you really know why you are doing it. If you set a height here, AutoCAD will not be able to scale if for uses such as dimensioning, in other words, it is fixed at the height you set. Annotative text is a new feature in AutoCAD 2008. This will be covered in a later tutorial.

Use the **MTEXT** command to create some text with your new style. Make sure that you pressed the "Set Current" button when you created it.

# DEFAULT TEXT CUSTOM TEXT

You should now see two different styles of text in your drawing - the original 'Standard' text style (Arial) and your new 'Custom' style (Times New Roman - Bold).

#### Adding Text to a drawing

Of course, there a few ways of entering text in AutoCAD. For simple one line text, use the **TEXT** command. There's a few parameters needed to add text, but if you follow the command line, it's quite easy.

Command: TEXT

Current text style: "Annotative-Custom" Text height: 0.2000 Annotative: Yes Justify: Left Specify start point of text or [Justify/Style]: <PICK> Specify paper height <0.2000>: <ENTER>

Specify rotation angle of text <0>: <ENTER> <TYPE YOUER TEXT>

You should now have a line of text in your drawing. Single line text created by the **TEXT** command is a different object that the Multiline (**MTEXT**) text that you created earlier. It is harder to format and you can not add a second line to it. In most cases these days **MTEXT** is the preferred way to add text.

Start the **MTEXT** command and pick the two corners of the text window. Stop and take a look the ribbon and see what options you have (these are not available when you create single line text).



You'll notice two distinct areas shown here. On the left you have button that displays each of the text styles that are defined in your drawing. In the other area outlined in red are some settings you can choose that will override the settings you created in your text styles. As a rule, I recommend that you do not get into the habit of overriding your styles. It is MUCH easier to modify a style than find and change all of the unique overrides that you have made in a drawing.

You will also see that there are section for justification. Try some of these settings on your text to see how they work. Middle left for example will justify the lines of text to the middle of a paragraph on the left side. Another very handy section is the 'Symbols' drop list that lets you easily add symbols for 'Center Line', 'Angle' and other drafting terms.

There is an easy way to change existing text to a different style. First, select the text, then go up to the Text Style Menu (Home > Annotation > Text Style) and select the style you want from the droplist (below) and the text will change automatically.



Professional looking and clear text is essential in any CAD drawing. Practice with different styles until you are comfortable with these concepts.

## Multiline Text

Often you will want to enter a block of text, maybe a paragraph explaining a problem. In this case, **MTEXT** is what you need. Mtext allows you to format your text much like a word processor.

Start the Multiline Text command, and pick a spot in the drawing. Drag the cursor over to create a 'rectangle' to represent the area you want your text in, and pick the other corner. As soon as you pick the second point, the full Multiline text editor will appear. Type your text, and press OK when done.

All of a sudden, AutoCAD turns into a word processor.



You now have options for editing your text. You can change the font, size, color - all without having to create a new text style. If you use a True Type font for your text style, you have more options for formatting (just don't get carried away). Imagine a large drawing where everyone decides their own way of showing text.



**NOTE:** AutoCAD uses two types of fonts: true-type and 'SHX' A vector fonts. Truetype fonts are indicated by a small **T** symbol beside the font name. In older versions (Pre-2010) you would use the SHX fonts, as AutoCAD could draw these faster. Newer releases of AutoCAD handle true-type faster than before. Another concern when using True-type fonts is that if you are exchanging drawing files with other people, they may not have the same fonts installed on their system. This could lead to formatting problems when they open your files.

## Editing text

The easiest way to edit the contents of your text (what you wrote) is to double click on it. Depending up your method of entering the text, a different editor will appear (Mtext or single line text).

If you want to change the shape of your Mtext box, just pick on the text, and use grips to drag the frame to the right or to the bottom.



Single line text cannot be changed like this. When deciding when to use single or Multiline text, think about how the text will be used, if it might be edited later, it's generally a safer bet to use Mtext.

## Arc-Aligned Text

Recently AutoCAD added a selection of commands called "Express Tools". These are regular AutoCAD commands, but they aren't fully supported (meaning don't cry to Autodesk if they don't work). They are also sometimes installed separately from the regular AutoCAD installation, so you may not have them on your screen. But

they do offer a nice option to some drawing problems. One of these Express Tools is Arc-Aligned Text. Just like the name suggests, you can place text along an arc.

Draw an arc and start the command and type in **ARCTEXT** to start. As prompted, select the arc, and this dialog box will appear:

	ArcAlignedText Workshop - Create				
	Custom				
	Text: THIS IS TEXT ALONG AN ARC Properties: Text height: 0.425 OK				
/	Text height: 0.425 Width factor: 1.0	Offset from arc: Offset from left:	0.325	Cancel	
	Char spacing: 0.095	Offset from right:	0.0		

As you can see, there are quite a few options. Use the default settings and you should have text that looks something like this:



Note that you cannot edit this text like regular MTEXT. If you double click on it you can edit the size and the 'arc offset' if you need to, but you can not change the style or font.

You can also erase the arc and the text will remain. This isn't a command that you will use a lot, but it is comforting to know that it's there. Check out the other tools in the Express Tools area.

**Review: Text in a drawing should be clear, concise and uncluttered.** Always use **CAPS** unless you're told otherwise. Use different styles for different situations, but don't get carried away. Use Multiline text when creating more than one line of text. Check your spelling. Reread what you just wrote and ensure that it will make sense to someone else reading it that is not familiar with the drawing.

#### Video: Text in AutoCAD

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Take the Lesson 2-6 Quiz

What can be a problem with using True Type fonts?

They are too big

None

Other users may not have the same fonts



For Further Thought:

If you could explain everything about your design with just a drawing, we wouldn't need the Text commands. Instead, you need to realize that any text you add to your drawing should be there to clarify and inform the person reading the drawing. You need to make sure the font is clear, the placement and size are correct and the content is easy to read. Just like any other document, grammar and spelling count! With technical drawings, you need to make sure that if you typed "Aluminum Alloy 6063" you didn't mean "Aluminum Allow 6061". Spell check won't catch that one for you - and a mistake like that could cost your company a lot of money (and you, your job!).

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# **CONTINUE TO THE NEXT LESSON: 2-7**



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Last Update: 02/25/2019 18:17:03

#### Topics covered in this Lesson:

Polylines - creating and editing, **Revision Clouds** 

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.

Polylines have some unique qualities that make them very useful:

They can have width (constant or varying)

They can consist of arcs and lines.

They can be edited

They can be joined together.

They can be exploded into individual segments

Command	Keystroke	lcon	Location	Result
Polyline	Pline / PL	Polyline	Home > Draw > Polyline	Creates a polyline of arcs and/or lines.

Polyline Edit	Pedit / PE	۵	Home > Modify > Polyline Edit	Edits polyline objects
Explode	Explode / Exp	<b>4</b>	Home > Modify > Explode	Separates objects into individual components
Revcloud	Revcloud	Rectangular	Home > Draw > Revcloud	Draws a (rectangular by default) Revcloud

When you start the command, you will notice that there are several options available:

```
Command: PL PLINE
Specify start point:
Current line-width is 0.0000
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]:
```

Remember that any time you want to choose an option, you type in the **C**apitol letter of that option.

One important tool is the Close option. This truly closes a polyline, which is different that just having two ends meeting at the same point - it's like welding the ends together. A closed polyline is required for extruding into 3-D space and other modifications. If you use the c option while drawing to complete your polyline, it will automatically be closed.

The **Spline** option allows you to draw a shape of lines using arcs and/or straight lines and then change them into one flowing curve. (Spline is also a separate command on its own.)

Also by changing the **width** of a polyline, you can get some different looks to your lines. This is one way you can draw an arrow in AutoCAD:

```
Command: PL PLINE
Specify start point: <PICK FIRST POINT>
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: W
Specify starting width <0.0000>: 1
Specify ending width <1.0000>: <ENTER>
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: <PICK</pre>
SECOND POINT>
Specify next point or
[Arc/Close/Halfwidth/Length/Undo/Width]: W
Specify starting width <1.0000>: 4
Specify ending width <4.0000>: 0
Specify next point or
[Arc/Close/Halfwidth/Length/Undo/Width]: <PICK THIRD POINT>
Specify next point or
[Arc/Close/Halfwidth/Length/Undo/Width]: <PICK FOURTH POINT>
```

Depending upon the length of your lines, you may end up with a polyline that looks like this:



Look at the examples below and then practice drawing polylines and try some of the options available.



Below is a sample drawing you can reproduce using a polyline (it could represent a slot to be cut out with a <u>CNC</u> router. First, think about how you would draw it using lines, arcs and circles.



Here's how you draw it using one polyline:

Remember <u>DDE</u>? Start by entering your first point, then (with Ortho or Polar on) move your cursor to the right and type 5 <ENTER> then A <ENTER> for arc and move your cursor up and type 2 <ENTER> then type L <ENTER> (to go back to straight lines) and more your cursor to the left and type 5 <ENTER> then back to arc (A <ENTER>) and then type CL to close the polyline. Wow! That was easy!

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]: L 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

To edit a polyline, use the **PEDIT** command or type **PE** at the command line.

Command: PE PEDIT Select polyline or [Multiple]: <SELECT A POLYLINE> Enter an option [Close/Join/Width/Edit vertex / Fit/ Spline/ Decurve/ Ltype gen/Undo]:

One of the most common editing tools is the **Join** (J) option. Use this to combine two or more polylines and their endpoints.

Use the **LINE** command to draw a line from 0,0 to 5,5. Now use the line command to draw a line from 5,5 to 8,5. This will leave you with 2 line objects that share a common point (5,5). Start the PEDIT command (**PE**) and select one of the lines. AutoCAD will ask you if you want to turn it into one, select **Y**<enter>.

```
Command: PE PEDIT Select polyline or [Multiple]:
Object selected is not a polyline
Do you want to turn it into one? <Y> <ENTER>
Enter an option [Close/Join/Width/Edit vertex/ Fit/Spline/
Decurve/ Ltype gen/Undo]: J
Select objects: <select a line> 1 found
```

Select objects: <select the other line> 1 segments added to polyline

Enter an option [Close/Join/Width/Edit vertex/ Fit/Spline/ Decurve/Ltype gen/Undo]:<Enter>

Now you have one object (a polyline). This is a handy tool to use for editing.

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

Once you have a polyline, it is sometimes useful to break it into the individual segment. For example, you might want to offset segments by varying distances.. To do this, use the **Explode** command. It is used to separate many other AutoCAD objects as well such as blocks.

Polylines are useful - try to use them whenever possible. Sometimes you can draw the outside walls of a building, offset the polyline for the wall thickness, then explode both of them for more versatility.

#### Video: Polylines in AutoCAD

#### Revcloud (Revision Cloud)

Another different version of the Polyline command is **Revcloud** (Revision Cloud) command. This is something you don't want to see on your drawings when the boss reviews your work. A Revcloud is used to highlight problems, revisions or errors in a drawing.



You might also want to create a Revcloud to highlight a question you have with your own work.

To create a Revcloud, start the command. Then set the arc length to about 12 for an architectural drawing, or test out different arc lengths.

Command: REVCLOUD

```
Minimum arc length: 8.0000 Maximum arc length: 8.0000 Style:
Normal Type: Rectangular
Specify first corner point or [Arc
length/Object/Rectangular/Polygonal/Freehand/Style/Modify]
<Object>: A <ENTER>
Specify minimum length of arc <8.0000>: 4 <ENTER>
Specify maximum length of arc <4.0000>: <ENTER>
Specify first corner point or [Arc
length/Object/Rectangular/Polygonal/Freehand/Style/Modify]
<Object>: <PICK FIRST CORNER>
Specify opposite corner: <PICK SECOND CORNER>
```

You might have ended up with something like this:



One thing with revclouds is that it can be hard to define a specific area clearly. Therefore, you have another option for creating accurate, clean revclouds like the one in the first example. Start by drawing a polyline over the area you want to highlight, then start the Revcloud command and choose the Object option. Select the polyline and press enter to end the command. This will work on most AutoCAD objects.

#### Video: Revclouds in AutoCAD

**Extra Practice:** Copy this drawing - <u>extra 2-001.gif</u>. You will need to use the Arc option when drawing the polyline. Try to draw it as one line, but if you have trouble, draw two polylines and join them. After drawing it, use the **S**pline option of **PEDIT** to change the polyline. Once you have the spline, use the **D**ecurve option.

**Extra Practice:** Draw the exercise from <u>Lesson 1-4</u> as a polyline. This is also a good chance to review your input options.

**Extra Practice:** Copy this drawing - <u>extra 2-002.gif</u>. The centre line in the middle indicates that both halves are the same. Create one polyline, then offset it to get the smaller one.

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#### Take the Lesson 2-7 Quiz

Can a polyline be given a variable width?

Only in line segments No Yes

#### For Further Thought:

If the 'line' command is a tool like a hand saw, then the 'polyline' command is like a jig saw. You now have an even more powerful way to draw a line. Why don't we use only polylines and let the line command go away?

Think about the offset command for example - you can offset a single line (which is usually what you want) instead of a complex polyline. Grip editing is much easier on a single line - especially for rotate.

What I'm getting at is that you now have two tools that do 'almost' the same thing, and you get to decide when to select the best usage for each one.

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# **CONTINUE TO THE NEXT LESSON: 2-8**



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#### Topics covered in this Lesson:

Layout Tabs | Printing from a Layout | Rotating your View | Freeze/Thaw Layers

In <u>Level 1</u> you printed out your project using 'model space'. This lesson will show you the preferred way of plotting your drawings.

In AutoCAD there are two different workspaces: model space and Layout / Paper Space. Think of model space where you make your model, or draw. Think of the Layout Tabs as where you layout the prints of your drawing or layout the final drawing complete with dimensions, notes, title block, etc. Look at the images below to see a visual explanation of the concept.

> Note about the terminology in this Lesson: 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, you can re-sequence them and more. I'm going to use the term paper space as I feel it explains the process better. (Plus I'm old and set in my ways!)

#### What is a Layout?

A layout is a page that allows you to set up a plot or printout of your drawing. Below is a sample of what a layout looks like (a really simple one).



The floor plan and dimensions (above) were created in **MODEL SPACE at a 1:1** scale. The walls were drawn to be 30' or more long.

The title block and viewport were created in a **LAYOUT TAB / PAPER SPACE at a 1:1 scale**. The title block is 8-1/2 x 11 inches.

The viewport is a window into "model space". In the image below, the viewport is the heavy blue rectangle and the contents of the viewport are displayed inside it (shown as grey for this example). The contents of the viewport are scaled to either fit the viewport or (preferably) to conform to a specific drafting scale (i.e.: 1"=1')

Below is a visual representation of a layout for a print to be plotted from a Layout tab.



This conceptual image shows the relationship between model space and a Layout.

Think of paper space as being 'above' model space. To see through 'into' model space, you have to create a viewport (Make Viewport **Mv** command). Think of a viewport as a window. As soon as you create the viewport, the window is closed and the extents of your drawing are displayed. You can see through it, but you can not 'touch' anything in model space.

To have access to your objects in model space from paper space, you have to enter the viewport by typing MS <ENTER>. This 'opens the window' for access. Type in PS <ENTER> to 'close the window'.

You may be wondering why you shouldn't just plot from model space. You can, and many people do, but the advantage of plotting from paper space is that you can have many layouts from one drawing. You can even add detail views without having to copy and scale your geometry. This feature was radically changed in AutoCAD 2000 and is much more versatile. You now have multiple tabs to organize your plots (or 'sheets'). As a rule, unless you're working for a behind-the-times company with a confused CAD Manager, use Layouts.

To toggle between the two spaces, you can pick on one of the layout tabs and back to the Model tab. You can also enter the command **TILEMODE** and set it to **O** for paper space and **1** for model space. When you do this, you will notice that the UCS icon in the bottom left corner changed to a triangular icon. This new icon confirms that you are paper space.



When you are in paper space you can draw or insert a title block. In paper space you are still drawing at a 1:1 scale.

Before we go further, there is a setting that I want you to make. This will make your life a lot easier as we work through Layouts and when you are creating your own. For some reason, AutoCAD 'hides' the layout tabs from the screen. We need to turn them on, so this is how you do it. Type in **OP** (for options) and go to the Display tab of the dialog box. Check the box shown below for "Display Layout and Model Tabs". Press OK.

A Options	
Current profile: < <unnamed profile="">&gt;</unnamed>	🍋 Current drawing:
Files Display Open and Save Plot and Publish System User P	references Drafting 3D Mod
Window Elements	Display resolution
Color sche <u>m</u> e: Dark 🔻	500 <u>Arc</u> and c
	8 Segment:
Display <u>s</u> croll bars in drawing window	0.5 Rendered
Use large buttons for Toolbars	🐴 4 Contour li
Resize ribbon icons to standard sizes	
Show <u>T</u> oolTips	Display performance Pan and zoom with
Show shortcut keys in ToolTips Show extended ToolTips	Highlight raster image
2 Number of seconds to delay	Apply solid fill
Show rollover ToolTips	칠 🔲 Show te <u>x</u> t boundary
✓ Display File Tabs	Dra <u>w</u> true silhouette
Colors <u>F</u> onts	Crosshair si <u>z</u> e
Layout elements	5
✓ Display Layout and Model tabs	Fade control
	Xref display
Displays the layout and Model tabs at the bottom of the d area. When this option is cleared, the tabs are replaced by	
buttons on the status bar.	e edit and annotative
Create viewport in <u>n</u> ew layouts	
	OK Cano

After you press OK - you'll see that new tabs have shown up on the bottom of your AutoCAD screen:


Start this exercise by drawing a simple title block in the tab called Layout 1. Make sure you draw it at 1:1 scale. Start with an 8-1/2"x11" rectangle and make sure it overlaps the edge of the 'virtual' sheet you see. Offset it .375 inside and erase the outer rectangle. Make a couple of small rectangles for the text and add some text as shown below. This ensures that all of your title block will be printed as many printers do not print to the edge of the paper.



Once your title block is drawn, you can create a **floating viewport C**. This is a 'window' into your model space. To create a viewport, type in **MV** (for Make Viewport) and then pick two points just as if you are drawing a rectangle. Use as much of your title block as possible.

As soon as your viewport is created, you will see all that you have drawn in model space appear to the extents of the viewport. The next thing you need to do is to scale your viewport for accurate plotting.

To do this, you have to 'enter' your viewport. Do this by either typing **Ms** (for model space) or clicking on the square labeled **PAPER** (it will switch to Model) on the status bar. In newer versions of AutoCAD like 2005 you can also double-click in the viewport. You will notice that the familiar WCS icon appears in the bottom left corner of the viewport. If you have more than one viewport on the screen, left-click in the one that you want to scale or type **CTRL+R** to toggle through them to the one you want. To scale a viewport, you just use the **zoom** command. Type in **z** <ENTER>

At the command prompt, you must tell AutoCAD what scale you want the viewport to be. This coincides with the scale that you will plot at.

IMPERIAL SCALES		METRIC	SCALES
PLOT SCALE	ENTER:	PLOT SCALE	ENTER:
1⁄4" = 1'	1/48XP	1:10	1/10XP
<sup>3</sup> ⁄4" = 1'	3/48XP	1:20	1/20xp
1⁄2" = 1'	1/24XP	1:50	1/50XP

The table above gives you some sample scales. Take ¼" = 1' for example. First remember that 1'=12". If you cross-multiply the 12 by the 4, you get 48. Therefore this scale will be at a 1:48 scale (ratio), or in other terms, your model will be 1/48<sup>th</sup> the size of real life on paper. To get AutoCAD to scale the viewport, you must type in **Z <ENTER>**, **1/48XP <ENTER>**. This means 1/48<sup>th</sup> times (**X**) in relation to paper space (**P**).

Figuring out what scale you require takes some calculation (unless you are told which scale to use). You can see what your extents in model space are and then see what your viewport size is. You can also use trial and error to see what fits in your viewport. This can sometimes be quicker than calculating.

To sum this up, here are the basic steps required for using paper space layouts:

Finish your drawing (with dimensions) in model space.

Change to paper space. **TILEMODE** to **0** or click on the Layout tab

Add a title block.

Create a viewport using the MVIEW (MV) command.

Enter your viewport by typing MS (or double-click in the viewport)

Zoom in using a specific ratio by typing in z <ENTER> \_\_\_/\_\_\_XP <ENTER>

Leave your viewport by typing **PS** (or double-click outside the viewport) and return to paper space.

One thing that is very important to remember is that you should use paper space only for plotting. **Do not modify your model in paper space.** One reason for this is that you could have layers turned off in your viewport that are related to the objects you are modifying in the viewport. Return to true model space for any editing that you need to do. Also, do not draw objects on the Layout screen that are part of the drawing.

To practice this concept, open up one of the drawings you did in Lesson 2-1.

Change to paper space. Draw or insert a title block. If you do not have one, you can find a good 11x17 (B size) title block in your AutoCAD support folder (filename: ANSI\_B.DWG). Maybe you drew one in <u>this tutorial</u>? Insert this and then create a viewport. Enter your viewport and scale as explained above. Return to paper space and print using the settings, EXTENTS / SCALE: 1=1.

Alternately, when you have a viewport selected or active, you will see a list of scale options at the bottom of the screen to define the scale. Select the viewport, and then select the scale from menu:

	4:1 / 400%
	8:1 / 800%
	10:1 / 1000%
	100:1 / 10000%
	1/128" = 1'-0" / 0.07%
	1/64" = 1'-0" / 0.13%
	1/32" = 1'-0" / 0.26%
	1/16" = 1'-0" / 0.52%
	3/32" = 1'-0" / 0.78%
	1/8" = 1'-0" / 1.04%
	✓ 3/16" = 1'-0" / 1.56%
00 PAPER 🗜 🐀 ⊾ 🥝 🕶 📉 🕶 🔼 🕶 🚍 📾 ゛ 🖾 🗶 🗶 🕾	[ 3/16" = 1'-0" / 1.56% 🔻 🗾

# Printing from a Layout Tab

Now that you have a layout of your drawing scaled, you can start to print it. The first thing you need to do is right-click on the layout tab and select "Page Setup Manage" as shown below



This is where you will select your printer and other parameters required for printing from the layout tab. Press the "New" button and give your new Page Setup a name and press OK:

🔺 Page Setup Manager	X
Current layout: Layout1 Page setups	
New Page Setup	
New page setup name: New-Layout Start with: <none> <default device="" output=""> <previous plot=""> *Layout1*</previous></default></none>	Set Current New Modify Import
OK Cancel Help OK Display when creating a new layout Close	Help

In the "Page Setup" dialog box, there are a few settings that you will need to make.

🔺 Page Setup	o - Layout1	A CONTRACTOR OF A CONTRACTOR O
Page setup		
Name:	New-Layout	DWG
Printer/plott	er1	
Na <u>m</u> e:	HP LaserJet 2200 Series PCL 5	▼ P <u>r</u> operties
Plotter:	HP LaserJet 2200 Series PCL 5 - Win	dows System K— 8,5″ →
Where:	Art's Desk	
Description:	:	
Paper si <u>z</u> e	3	
Letter		<b></b>
Plot area		Plot scale
What to plo	t:	Fit to paper 4
Extents	<b>▼</b>	<u>S</u> cale: 1:1 ▼
	origin set to printable area)	1 inches 💌 💻
<u>X</u> : -2.69	98198 inches Center the pla	t 1 u <u>n</u> it
<u>Y</u> : -1.03	31746 inches 5	Scale lineweights
Preview		ОК

- Select your printer from the list
- Select the Plotstyle (colors) I have a B&W laser printer, so I have selected Greyscale.ctb. If you have a color printer, try this or select acad.ctb.
- Select your paper size (usually 'Letter' for most home printers).

- Set your scale to 1:1 remember why we're using layouts
- Center the plot it just looks better.
- Check the radio button for either portrait (vertical) or landscape (horizontal).
- Press OK to close the dialog box and save your settings.

Back in the Page Setup Manager, select the **Set Current** button to apply your new settings to the layout and then press the Close button.

Now you are ready to print your layout. Press CTRL+P or type PLOT to bring up the plot dialog box. You should see all of the settings that you just entered. Click the Preview button to see how your plot will look. If it looks like it should, you can now plot. If you need to make changes, press Escape and make the needed changes.

## Rotating your view in a Layout

There are sometimes when the view that you need in a layout is not orientated the way that you want it. for example, <u>elevation</u> drawings are often drawn in place and projected from the original drawing like this:



This is fine for model space, but it's not how you want to present it in your printouts. Luckily for you, AutoCAD has an easy way of fixing this.

Download the file **Floorplan-with-elevation.dwg** and open it up in AutoCAD. It should look just like the image above.

Go into a Layout tab and create a new viewport or use the one that is there. Doubleclick to enter the viewport and type in **DVIEW**. This will give you the prompt to select the objects. You can select all of the elevation objects, (or just select a few if you have a slower computer.)

Command: DVIEW <ENTER>

Select objects or <use DVIEWBLOCK>: <SELECT THE ELEVATION OBJECTS> Specify opposite corner: 5 found

Select objects or <use DVIEWBLOCK>:<ENTER>

Now that you have your objects selected, you will see quite few options, but we're only interested in one right now. Type **TW** for TWist.

```
Enter option
[CAmera/TArget/Distance/POints/PAn/Zoom/TWist/CLip/Hide/Off/Un
do]: TW
```

Now now enter the 'twist' angle that you need (think of this as the rotation angle).

Specify view twist angle <0.00>: 90 <ENTER>

Enter option
[CAmera/TArget/Distance/POints/PAn/Zoom/TWist/CLip/Hide/Off/Un
do]: <ENTER>
Regenerating model.

Now that you have completed the **DVIEW** command, your elevation should be at the correct viewing angle. If you have other drawings, you will need to enter different angles depending upon their original rotation. Now you can freeze the unneeded layers in the viewport. Also check in model to space to find that you have not changed anything there.

The DVIEW command is also a lot in 3D work to obtain perspective and natural views.

#### **Top 10 Viewport Tips:**

- You can create new layout tabs by right-clicking on an existing tabs and choosing New Layout.
- You can rename a tab by right-clicking and choosing rename.
- You can create viewports of various shapes by creating the object and then using the MV command with the **O**bject option.
- You can also choose to dimension in the Layout. This can be very convenient in AutoCAD 2005. Your dimscale is 1 and all your dimensions are consistent. Try this out on a sample drawing. AutoCAD will scale the dimension to size of the object in the viewport. But you will need to change the DIMLFAC variable first. For example, if your viewport is 3/16 change it (16/3)X12 or 64 if you are using imperial measurements.
- You can resize viewports with regular modifying commands and grips.
- Put viewports on the layer called DEFPOINTS (created by dimensions) and they won't plot or put them on a separate layer with the plot option turned off.

- To quickly find the zoom factor of your viewport, use the **LIST** command on it.
- You can lock and unlock viewports by using the MV > L option.
- You can freeze (turn off) layers in any viewport. (See below)
- You can now have your viewports plot for 3D drawings as wireframe, hidden line and rendered views when you plot - this is great! (See below)



Freeze and Thaw Layers in a Viewport

To turn layers off in a viewport, you need to be on a Layout Tab and in Model Space of a viewport. In the viewport, then move your cursor to the layer droplist and select the layer you want off and hit the "Freeze in Viewport" icon (in red box) for that layer, then click on the drawing area. You can also do this directly from the Layers Dialog. Note that the kitchen fixtures likes the stove are frozen (not visible) in the large viewport, but are thawed (visible) in the smaller viewport that shows the kitchen detail.



To reverse this, use the same process which will "Thaw" the layer in that viewport.

## Video: Layouts / Paper Space in AutoCAD

**Extra Practice:** If you are feeling ambitious, check out <u>this PDF file</u> that shows a complete plan for a shed. See if you can reproduce the plan and details into a set of drawings. Save the drawing when you are done to use in the next level, so that you can make a 3D model of the shed.

**Review:** The ability to use layouts successfully leads to well organized drawings and print sets. From there, you can put together <u>sheet sets</u> for distributing to other people. When it comes to plotting 3D drawings, Layouts are essential to capture the views you need and put them on paper. It may take some time to master the concept and procedures, but the results will be well worth it.

🕒 Share / Save 🖪 😏 🖻 ...

# Take the Lesson 2-8 Quiz

When you are in a Layout tab what do you draw to see your model? A viewport

A Window

A Box



# For Further Thought:

A lot of people might tell you that it's easier to plot from model space. In many cases they might be right. On the other hand, you need to know how to properly plot from a layout tab. It can be a very tough concept to learn and I've seen a lot of students struggle with this one. But you know what? Once you learn it and get comfortable with it, you'll plot all but the very simplest AutoCAD drawings from layout tabs. If you start drawing in 3D, you'll almost always use layout tabs. Make sure you know these concepts before moving on - this is the sort of professional technique an employer will expect you to know.

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# **CONTINUE TO THE NEXT LESSON: 2-9**



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Topics covered in this Lesson:

Parametric Geometric Constraints | Dimensional Constraints

#### This is a new feature in AutoCAD 2010.

Parametric constraints allow you to force an object to behave the way you want it to. If you need a line to remain vertical at all times, you can set a constraint on it to do just that. Need 2 circles to remain the same diameter? How about a circle for a bolt that must stay 1/2" diameter? No problem with constraints. These are just some examples that will make it easier to control your drawing.

Constraints can be divided into two groups and are on the **Parametric Tool Panel**:

COMMANDS	ICONS	RESULTS
Geometric Constraints	L > ○ 0 Auto → ↓ □ =	Constrains a object based on geometric properties : vertical, horizontal, etc
Dimensional Constraints	Linear Aligned	Constrains an object based on a set length or radius.
Display Constraints	i 등 Show/Hide i 등 Show All i 등 Hide All	Use this icons to turn the visibility of the constraints on or off.
Delete Constraints	Delete Constraints	Use this icon or the <b>DELCON</b> / <b>DELC</b> command to remove the constraints from the object.

This tutorial is not going to explain each one, but jut show you how they're used and then you can explore the others and learn how they work. This may not be something that you use in everyday drafting, but it can very useful at times.

## Geometric Constraints

Let's start with an easy one. Draw 2 random lines without having Ortho on.

Click on the Parallel Constraint icon  $\swarrow$  and then select one line and then the other. The second line should now be parallel with the first.

The angle of the first line will be the constraint put upon the second line. Now if you try moving one of the lines around, you'll find that the two will stay parallel. If you highlight one, the other becomes highlighted. You also now have constraint indicators next to the objects.



Most of the geometric Constraints work the same way. Select an object to create the constraint and then select another to match. You can only constrain 2 objects at a time, but you can use multiple constraints to constrain multiple objects. If you copy one of the constrained objects, **the constraint does not copy with it**.

Try out a bunch of the geometric constraints before moving on to dimensional constraints. Notice how the procedure is pretty much the same. See what happens when you add a new constraint to an object that already has a constraint. Some will work, some won't - try to figure out why.

#### **Dimensional Constraints**

Dimensional constraints are different from what you just worked with. Instead of making a line vertical (for example), you can make a line 10 units long and make it stay that way until you change the constraint. You can have more than one dimensional constraint on certain objects.

Draw a random angled line on the screen. Pick on the Aligned constraint



icon. Aligned Pick two points on the line. Notice that even if you have your Osnaps off, you can only pick the endpoints and midpoint on the line. After selecting the 2 points, you can now enter a length that you want the distance between those points to be.



With the constraint still highlighted, enter a number. d3 in this example refers to the 3rd dimensional constraint in the drawing. If you added a constraint from end to middle, add another from end to end (or vice versa). Notice that the constraint will be double the first one. If you change one, the other will change.

#### Mixing it up

You'll notice in the geometric constraints panel, there is one for equal. this can used to make objects the same size. In this example, draw 2 different sized circles. Make

one a dimensional constraint for diameter. now use the Equals constraint = to make the other one the same.

Imagine using this when you are designing a part with holes to be drilled. If the part isn't finalized yet, the hole size might change. Using the Equals constraint would allow to change one and make the others the equal at the same time. Draw four random sized circles and make them constrained to the same **equal** size diameter. Then use grips to change the diameter on one. Add a dimensional constraint to keep them the same distance apart.

If you use a lot of constraints in a drawing, your screen can get cluttered up very fast. Use the display tools to turn their visibility on or off.



This tutorial has shown you the basics of working with parametric constraints. The are simple to use, easy to delete if you don't need them any more, and they can be a great time saver. Practice with them. Go over some previous exercises using constraints.

## Video: Constraints in AutoCAD

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### Take the Lesson 2-9 Quiz

The two types of constraints are...

Geometric and Dimensional

Parallel and Perpendicular

Geometric and Orthographic



For Further Thought:

When I first learned AutoCAD, parametric constraints were not part of the program. I used other programs that had them and saw their value. Honestly I don't use these much in what I draw these days. But for mechanical drafters, this is a concept that you will want to learn. You'll need to recognize when to apply them and when they are not needed.

If you are noticing a pattern in these "For Further Thought" summaries, you should. What I'm trying to let you know is that you have a LOT of options when you are using AutoCAD. Your toolbox is filling up and you need to know when to use those tools to be most effective. I can't guide you on that. I can show you how the tools work, but it is up to you to develop the best practices for you and your drawings.

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# CONTINUE TO THE NEXT LESSON: 2-10



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### Topics covered in this Lesson:

#### **Elevation Drawings and Techniques**

# What is an Elevation Drawing?

By now you should be familiar with floor plans. A floor plan is a view of a building drawn as if you were looking at it from the ceiling. An elevation drawing is the view that you would see in real life as you stood on the floor or ground and looked at the wall. This view can be either an exterior or interior view.

Like any CAD drawing, an elevation view is used to convey information. An exterior elevation might be used to show a home buyer what the front of their new



home will look like, complete with bushes, bricks, and window trim. Another example is an interior elevation drawing to show a contractor where finishes and items are to be used. The first example would be quite detailed, while the second would be more basic (but might contain dimensions). The number of elevation drawings needed will be based on each project's needs. This exercise will be your first chance to start designing. You'll start with a floor plan and draw a front exterior elevation and an interior elevation. The techniques will be the same, but the difference will be in the detail that you provide. The end of this tutorial features a **video** for another look at this concept.

### Getting Started with Elevation Drawings

Before you start the elevation drawing, you will need a floor plan. You will need to establish the overall size of the building and placement of windows and doors and other features.

If you are not familiar with Orthographic Projection, take time to review <u>this tutorial</u>. The concepts are very similar. To create an elevation, you will project down from the floor plan to create a new view. For this exercise we are going to use the Cabin drawing that you might have drawn earlier in this level. If you didn't draw it, you can download it <u>here</u>. Have a look at it and start thinking about what the cabin might look like from the outside.



The left side has a deck, some steps, a door and two windows. You will also have to consider the roof and the fireplace (which will have a chimney. I haven't given any heights to the walls or windows nor the slope of the roof. You will need to figure these out (I'll help!) and you will also have to design the exterior materials. Will your cabin have brick, stone, wood or some other material on the exterior wall?

### First Steps

Now that I have the drawing open, I'm going to turn off any layers that I don't need. Next, I'll make a copy of the floor plan and move it well away from the original. I have lots of room in model space, and don't want anything to overlap. Then I will rotate the drawing so the the side that I'm drawing the elevation of is pointing down.

Now I can start drawing (but of course, I created some new layers) and project some lines down from the key points of the floor plan. Use Osnaps to be accurate. If you followed along, you should see something like this:



So far this is just like Orthographic projection. What's next? Remember how I've said earlier to "draw in the order that you would build it"? In this case, you want to draw a line that represents the top of the foundation that the cabin sits on. This will be the baseline from which the other lines will be offset.

How tall are your walls? Allow from for a small attic or insulation on top. How tall is the door? The top of the windows will usually be at the same height as the door.



Now I've got my windows roughed in, a door and a wall. Good start, but lots more to do.

I'm going to use a pitch of 5/12 meaning that for every 12 units of length, the roof will rise 5 units. Since my cabin is 24 feet wide and the roof is in the middle, this is an easy one to figure out. For 12 feet of length, the roof will rise 5 feet. Therefore my roof will be 5 feet high.



Next I want to add a overhang for my roof. To do this, I could calculate is again, or I could draw it as another projection. I did this and then rotated it so that you could see it clearer. Have a look:



Looks like a 5 inch overhang if I extend the roof out 12 inches past the wall. Now I just need to offset a line 5 inches. The image below shows my rough outline for the elevation. I've added in a hidden line to show where the wall is, but this is only for reference and will be erased.



Next I'm going to draw the deck. If you look at the floor plan, there is a break line that shows the deck is bigger. I am going to make my wrap around to the side and make it larger to accommodate a table, hairs and a bar-b-que. I also need to add the steps and the railing around the deck.

I haven't designed a deck before, so I did what I normally do when I need information: I asked google. I went to deckplans.com and got the specs that I needed. Here's what I downloaded and will use as reference: <u>Link</u>. Deckplans.com will also give me the specifications for the foundation blocks: <u>Link</u>.

Looking at the plans, I see that I need some foundation blocks, 1-1/2" of overhand and 2x6 boards. Since this is for demonstration purposes, I'll keep it somewhat simple.



If you looked at the plans, you would have seen that 4x4 posts are needed to support the deck. I've added in the foundation blocks at 3 feet spacing, and added the deck and the steps. Now I need to add the railing.

I check the code to see that the minimum height for a railing is 36 inches about the deck surface. I'm going to make mine a little taller at 40 inches. I also checked the codes for my baluster spacing. I can have a maximum of a 4 inch opening between balusters. Now that I know that, I just need to know what size my materials will be. I will use a 4x4 post at all of the corners and 2" wide straight baluster. To draw this is just a simple matter of offsetting and then trim the objects 'behind' the deck railing.



Note that I used the 4x4 posts for the handrails by the steps. It's also a lot of lines. I have changed the walls and windows to a Blue layer. Notice how you can still see the walls through the balusters? The window has been trimmed.

With the deck done, I'm going to go back to the main wall. I wanted to get the deck done first, because I knew I would be blocking some of the wall. My usual rule for detailing elevations is to add detail first to the objects 'closest to you'.

Before I add details, though I will make a chimney for the roof. I checked code and found that most chimneys should be 2 feet about the peak of the roof. So I drew a line down from the center of the fireplace (notice the small circle for the flue?) and then offset it to make a 18 inch wide chimney.

Now I can add my hatch patterns to make the walls and chimney brick and add shakes to the roof. After a few hatch commands and some other small details, you might end up with something like this:



Remember that elevations are a visual representation, so you can make them as detailed as needed. You can go a step further and add colors in a program like Photoshop (for an example, click <u>here</u>). Later on if you start drawing 3D, you can create rendered images that look almost photographic, but for 2D work, elevation drawings do a great job.

Learn how elevations work, and be careful about what objects appear in front of others. There are not a lot of commands that are needed when creating elevations, but you need to learn how to use them wisely, and be fully aware of what you are drawing.

**Note:** You can also draw the elevations in place without rotating the original floor plan. In this case you can easily add this to the layout by using the **DVIEW** command. Check out this **tutorial** for details.

**Extra practice:** Copy the floor plan to the left, then rotate it 90 degrees counterclockwise. Draw you projection lines down, then draw lines over from your existing elevation to get the heights. Finish this and the other 2 sides.

**Extra practice:** Draw some interior elevation from this floor plan. Add outlets, lights and other details.

Video: Creating Elevation Drawings in AutoCAD

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### Take the Lesson 2-10 Quiz

What is an elevation drawing?

A 2D drawing of the side of a building

A 3D Rendering

A photographic representation



## For Further Thought:

If you started learning AutoCAD at Lesson 1-1 with myCADsite and have just finished the 2nd level, then you have learned a lot about AutoCAD. You've gone from drawing your first line to having the knowledge to draw a set of house plans in 2D. That's a lot of power and a lot of tools at your disposal.

The next level will show you how to draw in 3D and you'll need a bigger toolbox now. Even if you don't plan to draw in 3D, please work through the first few lessons of Level 3 as I deal with options that most 2D drafters still need to know.

Level 4 is where I fill in the blanks with some random topics. So again, if you don't plan to go through the 3D tutorials, please review the first few in Level 3 and complete Level 4.

Congratulations on getting this far - and thank you for using myCADsite.

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# **CONTINUE TO THE NEXT LEVEL**



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# Topics covered in this Lesson:

An introduction to drafting in 3 Dimensions

# **INTRODUCTION - WHY 3-D?**

You may have already figured out why CAD has many advantages over manual drafting. One big advantage is that once you've drawn something, you shouldn't have to draw it again. If you manually drew a house plan, you would have to draw a front elevation, side elevations, and possibly a perspective view. With one 3-D CAD model, you can generate views from any angle either inside or outside the house and animations. Afterwards, if your client needs something changed, you can then make the changes once and re-render your views. If you're drawing mechanical parts, you can generate virtual prototypes or even create rapid prototypes. In this manner Boeing was able to **design and prototype** the 777 jetliner. This level of engineering would be impossible without CAD.

Your company may not do a lot of 3D work, but it is still a good skill to have and it's also more fun than 2D.

You'll be learning the 3-D concepts in the following order:

Isometric Drawings (not true 3D)

Wire-frame (very basic 3D)

Surfaces / Regions (primitive 3D)

Solid Objects (advanced 3D)



You will have a chance to draw the same object different ways to see the differences between the various methods.

Before entering the exciting world of 3-D, you'll have to learn some more CAD terminology. This level presumes that you have a good understanding of 2D commands.

#### 3-D CAD TERMINOLOGY

2-D	A concept of displaying real-world objects on a flat surface showing only height and width. This system uses only the X and Y axes.
3-D	A way of displaying real-world object in a more natural way by adding depth to the height and width. This system uses the X Y and <b>Z</b> axes.
<u>Boolean</u> operations	Commands that allow you to add, subtract or intersect solid objects in AutoCAD.
Complex surface	Generally a curved surface. Examples: car fender, landscape contour.
Elevation	The difference between an object being at zero on the Z-axis and the height that it is above zero.
Extrude	The extrude command raises the shape of a 2D outline into a 3D solid. For example, a circle would be extruded into a cylinder.
Face	The simplest true 3-D surface.
Facet	A three or four sided polygon that represents a piece (or section) of a 3-D surface.
Hidden line removal	A way of hiding lines that would not be visible if you were viewing the actual object you have drawn in AutoCAD. (Command: HIDE)
<u>Isometric</u> Drawing	A simple way of achieving a '3-D' appearance using 2-D drawing methods.
Plan View	Also known as the top view, a plan view looks directly down the WCS Z-axis to the X-Y axis.
<b>Primitive</b>	A basic solid building block. Examples would be boxes, cones, cylinders.
<b>Region</b>	A 2-D area consisting of lines, arcs, etc.
<u>Rendering</u>	A complex way of adding photo-realistic qualities to a 3- D model you have created.
Shading	A quick way of adding color to a 3-D object you have drawn. (Command: <b>SHADE</b> )
Solid Model	A 3-D model creating using solid 'building blocks'. This is

	the most accurate way of representing real-world objects in CAD.
Surface Model	A 3-D model defined by surfaces. The surface consists of polygons. (See facets.)
<u>Thickness</u>	A property of lines and other objects that gives them a 3-D like appearance.
<u>UCS</u>	The user co-ordinate system. This is defined by the person drawing to have easier access to portions of a 3-D model.
View	A particular view of the object you have created.
<u>Viewport</u>	A window into your drawing showing a particular view. You can have several viewports on your screen. Different from the viewports used in plotting.
<u>Wire-frame</u> <u>Model</u>	A 3-D shape that is defined by lines and curves. A skeletal representation. Hidden line removal is not possible with this model.
<u>Z-Axis</u>	The third axis that defines the depth.

As computers and software get more powerful, working in 3D becomes more popular. You have more power in desktop machine today, than was dreamed of when CAD first appeared. Be prepared that you will likely need to learn 3D at some point in your career. Once you are comfortable working in 3D, you'll find that you will rarely want to draw in 2D again. Autodesk (the makers of AutoCAD) make other products designed specifically for certain disciplines - <u>Revit</u>, <u>AutoCAD</u> <u>Architecture</u>, <u>AutoCAD Civil 3D</u>, <u>3ds Max</u> and more. Depending upon your budget and needs, you may need to look into programs like these. The concepts taught in these lessons will aid you when you work in any 3D program.

**Reference:** Have a look at some <u>AutoCAD renderings</u> before you start in this new world.

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# Take the Lesson 3-1 Quiz

How many views can you generate from a 3-D drawing?

As many as you need Only One

Three



# For Further Thought:

We live in a 3D world, yet for centuries have drawn the world around us in 2D. There is still a place for 2D drafting, but if you are serious about learning AutoCAD, you should know how to draft in 3D. 3D modeling opens up new roads that just aren't possible in 2D drafting. Learning how exploit the power of 3D will give you step up on other drafters who may only know to use 2D.

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# **CONTINUE TO THE NEXT LESSON: 3-2**



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### Topics covered in this Lesson:

#### **Isometric Drafting Techniques**

#### ISOMETRIC DRAWING

Using Isometric commands is one of the simplest ways to give a 3-D representation while using only 2-D commands. This has been the usual way of doing things before CAD allowed true 3-D work to be done. Many times an isometric drawing is used to compliment or give more information to a <u>3 view orthographic drawing</u>. See the sample below.



You can see that it is a very simple drawing. This basic isometric drawing of the object gives a very good idea of what it looks like. If this is all that is needed then isometric works well. Unfortunately, as soon as you change anything, like the block's height, you'll need to redraw all four views.

AutoCAD has a command called **ISOPLANE** which allows you to easily draw at a 30 degree angle as needed for an isometric drawing. You can switch between the three 'isoplanes' (top, right, left) by using this command or by pressing the **F5** key.

```
Command: ISOPLANE <ENTER>
Current isoplane: Right
Enter isometric plane setting [Left/Top/Right] <Left>: T
<ENTER>
Current isoplane: Top
```

Newer versions of AutoCAD have an Isoplane icon on the status bar. You can click this to enter Isometric Drafting Mode as shown below. You can also change your Isoplanes here (but F5 is faster).



By invoking this command, AutoCAD is now set to draw on the top isoplane by default. Your other choices would be Left or Right. Your first exercise will be to draw the object shown above using isometric methods.

#### Exercise 1 - Isometric Drafting

Begin a new drawing using the acad.dwt template

Create a <u>layer</u> called **OBJECT** and give it a green color. Make this your current layer.

Type in **DDRMODES** to bring up the Drawing Aids dialog box. Make your settings the same as what you see below (just turn on Isometric Snap).

A Drafting Settings					
Snap and Grid Polar Tracking Object Snap	3D Object Snap Dynamic Input Quic				
Snap On (F9) Snap spacing Snap X spacing: 0.86602540 Snap Y spacing: 0.5000 Equal X and Y spacing	☑ Grid On (F7)         Grid style         Display dotted grid in:         □ 2D model space         □ Block editor         □ Sheet/layout         Grid spacing				
Polar spacing Polar <u>d</u> istance: 0.0000	Grid X spacing:         0.86602540           Grid Y spacing:         0.5000				
Snap type © Grid snap © R <u>e</u> ctangular snap © Iso <u>m</u> etric snap © P <u>ol</u> arSnap	Major line every:       5         Grid behavior       ✓         ✓       Adaptive grid         ✓       Allow subdivision below grid spacing         ✓       Display grid beyond Limits         ✓       Follow Dynamic UCS				
Options	OK Cancel <u>H</u> elp				



change the radio button from "Isometric snap" to "Rectangular snap" to get back to the regular drawing settings.

Press **OK** and you'll see that the grid is set up for isometric drawing for the left isoplane in 1/2 unit increments. Your crosshairs are now angled to show you which isoplane you are currently on and the Grid is laid out differently from what you may be used to.

Begin by drawing the left side of the box (shown at the top of the lesson) using the line command. Ignore the hole at this point. You will want to use the <u>Direct</u> <u>Distance Entry System</u> for this exercise and make sure that you have Ortho (F8) and Osnaps (F3) turned on.

Switch to your right isoplane (F5) and draw the right side.

Switch to your top isoplane (F5) and draw the top view.

Create the angle lines to add the angled surface.

Switch back to the left isoplane and start the **ELLIPSE** command. At the command prompt, press **I** for isocircle. This will allow you to create an ellipse at the correct angle based on the radius of the circle in the orthographic drawing. Use the Osnap to pick the correct center point.

Save your drawing in your CAD folder. (You have a CAD Folder by now, right?)

To dimension an isometric drawing, you have to do a few things first. Create a text style called Left, and give it a 30° obliquing angle, then create another called right with a -30° obliquing angle. Then create a new dimension style that has the text aligned with the dimension line. After you have placed a dimension, use the **DIMEDIT** command to change the obliquing angle of the dimension + or - 30

degrees. You may also need to use the properties to change the text in the dimension to left or right depending upon the orientation of the dimension.

Keep in mind that this is still only 2D. Remember that in some instances, it may be quicker and easier to use this method rather than the more complex 3-D methods you'll be learning in the following lessons.

Extra Practice: Copy this drawing using isometric drawing methods.

**Extra Practice:** Copy <u>this drawing</u> using isometric drawing methods. If you want to add the hidden lines, jump ahead to <u>Lesson 4-3</u> to learn about linetypes.

Even More Practice: Try to reproduce <u>these drawings</u>. (Also keep this link bookmarked for when are looking for practice in 3D techniques.)

#### Video: Isometric Drafting

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#### Take the Lesson 3-2 Quiz

Is isometric drawing true 3-D drawing?

No

Yes

Only in Newer versions



#### For Further Thought:

Although this is still a 3D representation in 2D, you are already taking steps to 'think' in 3 dimensions. This is important and you will continue through these tutorials learning 3D AutoCAD techniques and performing them on a 2D monitor. Organizing your drawing with layers and 'seeing' and thinking in 3D takes practice, skill and sometimes a lot of patience.

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## **CONTINUE TO THE NEXT LESSON: 3-3**



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### Topics covered in this Lesson:

3D Coordinate system | Rotation in 3D

### THE 3-D CO-ORDINATE SYSTEM

By now you should be very comfortable working your way around the X-Y coordinate system. Anyway, here is a quick review. Looking from the plan view, this is what you see to figure out where is positive X and positive Y.



If you were to look at the same picture, but at a slight angle, you would see the third axis. This new axis is called the Z-axis. Imagine that the positive Z-axis is coming towards you out of the monitor.



The Z-axis has always been there, *lurking in the background*, waiting for you.

When you entered points previously, you would enter them in the format:  $\mathbf{x}$ ,  $\mathbf{x}$ . By doing this, you let AutoCAD know that in these cases, Z was equal to zero. Entering  $\mathbf{4}$ ,  $\mathbf{3}$  would be the same as entering  $\mathbf{4}$ ,  $\mathbf{3}$ ,  $\mathbf{0}$ . Now if you drew a line from the origin ( $\mathbf{0}$ ,  $\mathbf{0}$ ,  $\mathbf{0}$ ) to a point at  $\mathbf{4}$ ,  $\mathbf{3}$ ,  $\mathbf{2}$ , you would get a line that goes 4 inches to the right, 3 inches up and 2 inches towards you. The properties of this line would be this:

×	Lir	ie		42	
×	General				
	3D Visualization +				
	Geometry -				
		Start X	0.0000		
		Start Y	0.0000		
		Start Z	0.0000		
		End X	4.0000		
		End Y	3.0000		
		End Z	2.0000		
		Delta X	4.0000		
IES		Delta Y	3.0000		
PROPERTIES		Delta Z	2.0000		
Q		Length	5.3852		
Ы		Angle	37		

Notice that the line is actually **5.3852**" long. If you were to look at it from the plan view, it would look exactly like a line drawn from **0**, **0** to **4**, **3** Draw a line from 0,0 to 4,3 and then compare the properties.

The diagrams below, show this line from 4 different views to illustrate how things can look different in 3D. Look at each one carefully, and see if it makes sense to you.



This is the usual view you have seen when using AutoCAD in 2D. You are looking straight down the Z axis (positive Z is pointing at your). It looks like any other line you have drawn, going from 0,0 to 4,3 - but there is a difference...



If you were to look at the line from the front, instead of the top (as shown above) you would be able to notice the elevation of 2 units in the Z axis. This is the same line as above, only viewed from a different angle. In this view, you are looking straight down the -Y axis.



**Southeast View** 



Finally, here is the line as viewed in 3D space from the Southeast view. This is where viewing 3D objects on a 2D monitor gets tricky. You need to visualize the Z Axis.

What the above images show you is that you will have to get used to looking at a 3D world on a 2D monitor. In each image, the black line looks flat, but you have to use your reference points to determine where it truly is. If you don't understand this

perfectly right now, don't worry. It's just an exercise to expose you to 3D viewing. As the lessons progress, you will get much more familiar to this.

Why is this important to look at before entering the world of 3-D? If you were to only look at a 3-D model from the plan (top) view, you would not be able to see any difference between the two lines. (Draw them and see for yourself) On a 3-D model, you can easily have many points over top of each other. This would be very difficult to work with. You may think you're snapping to a particular endpoint, but the reality of it could be very different (think of how the top of the wall looks the same as the bottom of the wall if you're looking straight down it).

Fortunately, AutoCAD provides different viewing options for 3-D drafting. This will be discussed in more detail during in a later lesson. For now, one way to change your view is to type in the **VIEW** command. It will bring up this dialog box:

📐 View Manager			
Current View: Current			
Current Model Views	General -		Set <u>C</u> urrent
	Restore Ortho	Yes	
Layout Views	Set Relative to	World	Resta <b>New</b> he selected view
Top Bottom Left			Update <u>L</u> ayers
			Edit <u>B</u> oundaries
Right			Delete
2 Back SW Isometric GE Isometric			
NE Isometric NW Isometric			
	ОК	Cancel	Apply Help

Using the options shown above, you can see what the SW Isometric view looks like by selecting that choice on the left and then pressing the "Set Current" button. This will close the dialog box and change the view.

To get back to the top view, just start the **VIEW** command and choose "Top" from the list on the left and press "Set Current" again.

#### **3D** Rotation

Now for the confusing part. You already know how to rotate 2D objects, but you also have to know how AutoCAD measures angles of rotation in 3-D. There is a somewhat simple rule for this called **"The Right Hand Rule"**. To figure out which is the positive rotation angle, imagine that you are wrapping your right hand around the axis with your thumb pointing towards the positive end. The direction that your fingers are wrapped is the positive direction of rotation. This applies to all three axes.



Direction of positive rotation using the right hand rule

The main point of this lesson is to tell you that objects can trick you in 3D space. Shortcuts don't always work, you have to be careful with Osnaps and **your drawing** can turn into a mess very quickly if you're not paying attention. Trust me, I've seen enough students take the easy route and have to start over. If you want to learn 3D, review each lesson before progressing. Make sure you know the concepts inside and out. This is just an introduction to the concepts, you will learn more in the following lessons. You may still want to refer back to this tutorial, though.

#### Video: Working in 3 Dimensions



#### Take the Lesson 3-3 Quiz

In a new drawing, which direction is the Z axis pointing?

Up to the top of the monitor

Down to the bottom of the screen

Towards you



#### For Further Thought:

Now you have learned how adding an extra axis gives you an extra dimension. If you are already good at 2D drafting, then the X and Y axes are natural to you. Working with the Z axis will soon be comfortable as well, but the problem can being seeing the Z axis on your monitor. You need to learn the tools best visualize and work with your models.

Take you time to learn how to 'see in 3D' and it will make your drafting a lot easier.

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## **CONTINUE TO THE NEXT LESSON: 3-4**



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Last Update: 02/25/2019 18:19:07

Topics covered in this Lesson:

#### Viewing Objects in 3 Dimensions | <u>Tiled Viewports</u> | <u>Perspective Views</u> | <u>3D</u> <u>Orbit</u>

AutoCAD has a few ways for you to view an object other than the plan view that you have used for your 2-D drawings.



When you are in the 3D Basics workspace, your tool palettes on the ribbon will change. This will make it easier to find tools and icons that you'll use more often in 3D.

The first one we'll look at is the Visualize Palette.



I find this is the quickest way to learn the different views, and I'll show another way in a minute, but for now this is the fastest and clearest way to see different views of your AutoCAD objects in 3D.

Open this DWG file and then we'll look at it from a few different angles.

When you first open it, you'll see the drawing from the plan (top) view looking straight down the Z axis towards the origin. You'll see the X axis in its familiar location with positive to the right and the positive Y axis pointing up.

Using the Visualize tools shown above, select the **Bottom** View. You'll now be looking up from the Negative Z axis towards the origin.



Notice how the Positive X axis is now pointing to the left? And it's backwards? You're looking UP now instead of DOWN. It's simple to see in this example, but what if you were looking at a cube? (It would look the same from top or bottom.) Have a look at your UCS Icon - **check its orientation**, because even if you view changes, your UCS / WCS doesn't necessarily change with it. This is REALLY important to pay attention to when you start drafting in 3D.

Try a few other views from this list like the SW Isometric view, then return to the Top view.

### **Tiled Viewports**

There will be times when you may need to see around to the other side of your object. Then you can easily use the other preset views to see your model. You can also have more than one view on your screen at a time.

To achieve this, use the **Viewport Configuration** option of the **Visualize** tool palette and select the **"Four: Left"** option. Any view or viewport configuration can be saved under a unique name. Later you can easily restore this view when needed. Here are the preset viewport configurations that you can use in model space.



Change the view by picking in one of the viewports and selecting a new view in the droplist. See if you can get this same configuration in your drawing.

Here are a few general notes about views and viewports:

You can have viewports in either model space (tiled) or paper space (floating).

They can have different zoom ratios.

There is a difference between views and viewports.

If you go to a lot of work to get a view you need, save it using the **VIEW** command.

When saving a view, give it a descriptive name.

Keep an eye on the UCS icon in the corner of the screen or viewport.

The smaller your monitor is, the harder it is to see tiled viewports.

When you have more than one viewport, click inside the one you want active.

Change back to a single tiled viewport and go to the top view. On the right your screen you should see the "View Cube" This is a quick tool that lets you change your view with a click or two using a visual cube for reference and orientation.



To use the View Cube, you can click on different sections of the cube or on the directional letters. You can also select a corner of the View Cube to change to isometric views. In the example below, the highlighted corner will give you the SW Isometric view.



If this is confusing, that's understandable, but notice how the highlighted corner is between the W (West) and S (South) views? Click around on the cube and see if the views you get are what you are expecting. Once again, it will take a while to be comfortable viewing in a 3D world (on a 2D monitor).

For more practice viewing your 3D drawing, open up up this sample 3D file of a <u>table</u> <u>and chairs</u>. Change to different viewport layouts as shown above. Try some of the different Isometric views. Depending upon the size of your monitor, it may not be practical to use multiple viewports. In the end it is a matter of personal preference that decides what setup you are comfortable drawing in.

When you're finished exploring, change back to 1 viewport and switch to the **SW Isometric** view. Also, if you need to, change to 2D wireframe in your **Visual Styles** tool palette.



### PERSPECTIVE VIEWS

There is another command called **DVIEW** (Dynamic View) that gives you more control over the view of your objects. It will also allow you to see a perspective view of your model. This can be a very confusing method if you don't have a system. What follows is my approach to creating perspective views.

The basis of generating a perspective view is to have a virtual **camera** and **target**. Think of where you would like to 'stand' (the camera) and what direction you want to look at (the target).

In this example, I want a view that gives me slight perspective view of the table and chairs. The first thing I need to do is draw a point for my camera location.

Draw a line from the centre of the bottom leg 6' in the negative X axis. Now draw a line 5'6" straight up (@0, 0, 5'6) and finally draw a line to the middle of the far end of the table. It should look like this:



Now start the **DVIEW** command. You will be asked to select objects. You can select everything, or for the purposes of setting points, I would normally just choose the line as it makes the command faster in large drawings.

Start the command and then select the line. You'll have a bunch of options on your command line now, but you want to select POINTS (PO). This allows you to select a target and a camera, AutoCAD will aim the camera at the target and give you a perspective view. Follow the steps below to try this out.

#### Command: DV DVIEW Select objects or <use DVIEWBLOCK>: <SELECT THE LINE> 1 found

Select objects or <use DVIEWBLOCK>: <ENTER>
Enter option
[CAmera/TArget/Distance/POints/PAn/Zoom/TWist/CLip/Hide/Off/Un
do]: PO <ENTER>
Specify target point <84'-0", -113'-7 9/16", 4'-2
7/16">: <PICK THE ENDPOINT ON THE RIGHT (TARGET)>
Specify camera point <66'-10 11/16", -130'-8 15/16", 21'-3
13/16">: <PICK THE ENDPOINT ON THE RIGHT (CAMERA)>
Enter option
[CAmera/TArget/Distance/POints/PAn/Zoom/TWist/CLip/Hide/Off/Un
do]: <ENTER>
Regenerating model.

Start the **DVIEW** command again and select all the objects. In larger drawings, select the objects that are furthest to the extents of the drawing (this will save time and let you know the limits of your view) and any objects that are critical to your view. What this means in that if you are using the command on a very large drawing, the DVIEW command can slow down your system to a crawl. By selecting only the ones on the outside limits of your drawing, you will still be able to use the command quickly and efficiently.

This time, choose the Distance option by pressing D **<ENTER>**. This option effectively zooms in and out while in a perspective dynamic view. Move the slider at the top until it is just right of the '1x' and then left click with your mouse. Note that slower computers will take longer to regenerate the drawing for you. You can also enter distances manually on the command line - try this out as well. Each 3D model will require different settings to get the view you want.



While still in the Dview command, type in **PA** for the Pan option and move the table and chairs up slightly in the drawing area so that they are centered.

And lastly, type in **H** for the hide option for a more realistic view. and then press **<ENTER>**.



To preserve this view that you worked so hard to achieve, type in  $\mathbf{v}$  (for view) and select the New button and give your view a name and press OK. Now when you need to go back to this view you just type in  $\mathbf{v}$  again, select your named view, press the **Set Current** button and press OK. Your new view is also added to the top of the list in your View Tool Palette.

And View Manager Current View: Current Views			
Current	General 🔺		Set <u>C</u> urrent
	Name	DVIEW EXAMPLE	
Layout Views	Category	<none></none>	<u>N</u> ew
⊕ □ Preset Views	UCS	Unnamed	Update Layers
	Layer snapshot	Yes	
	Annotation sc	1:1	Edit <u>B</u> oundaries
	Visual Style	3D Hidden	Delete
	Background o	<none></none>	
	Live Section	<none></none>	
	Animation		11-+
	View type	Still	南中在
	Transition type	Fade from black i	
	Transition dur	1"	
	ОК	Cancel	<u>A</u> pply <u>H</u> elp

If you are using this command on larger drawings, like a home or building, the same process is used, but you should create the camera location to be further away from the objects you want to view. Practice makes perfect and this command can take some trial and error sometimes to get it just right. In the end, it is a very powerful viewing tool. When practicing, save views so you can go back to them if needed - it will save you a lot of time.

### 3D Orbit

Another way of viewing your objects is to use the **3DORBIT** command. This will give you a free hand to rotate your drawing around in space. Type in **3DO <ENTER>** and you'll see that your cursor changes and you can start rotating your objects around freely by clicking and moving your mouse. You can always reset your view as you learned above.

This is a handy way to quickly view your drawing and see if there is anything that doesn't look 'quite right'. You might find lines that snapped to the wrong OSnap or other errors using this command. When it comes to generating views, I still recommend the DVIEW command so that you have more control over what your view will look like.

#### Video: Using DView to create perspective views

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#### Take the Lesson 3-4 Quiz

Tiled Viewports are used in ...?

In both Model and Paper Space

**Model Space** 

Layouts / Paper Space



#### For Further Thought:

You may wonder why I am showing how to view objects in 3D when you might not know how to draw objects in 3D. You'll soon notice that having the tools to view what you are drawing is very important. There will be times when you get lost in your drawing and may not know which side is 'up' (it happens to the best of us).

Learning how to use the viewing tools allows you to see your drawing from the most useful angles. Viewing and drawing go hand in hand. Learn these tools - they're important.

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## **CONTINUE TO THE NEXT LESSON: 3-5**



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#### Topics covered in this Lesson:

#### **3D Wireframe Techniques**

Wireframe models are the simplest form of true 3D drawings. They can be used for conversion to other programs or other simple uses. Still, this is not a common way of drawing in 3D these days as the models are easily modified or have (digital) mass.

In this lesson, you will be drawing two objects. First, you will draw the shape from the **isometric exercise** using the same **LINE** command that you have always used. Then you will draw a simple chair using the same method. This exercise is good practice to work on the XYZ coordinate system.

Start a new drawing using the acad.dwt template.



Create a layer called **OBJECT** and give it a green color. Make this your current layer.

Using the line command, draw the outline of the front view placing the bottom left corner at 0,0,0 Then draw the circle.

Copy the outline and circle 2" up in the Z-axis. When asked for the base point, select anywhere on the screen, and copy it relative to the base-point: @0, 0, 2(X, Y, Z)

Command: CO COPY Select objects: <SELCT YOUR OBJECTS> 2 found Select objects: <ENTER> Current settings: Copy mode = Multiple Specify base point or [Displacement/mOde] <Displacement>: <PICK> Specify second point or [Array] <use first point as displacement>: @0,0,2 Specify second point or [Array/Exit/Undo] <Exit>: <ENTER>

Switch your <u>view</u> to the SW Isometric view. Notice that you have the front and the back drawn now. Using your endpoint Osnap, connect the corners of the object.

Your finished drawing should look like this:



Pick any line and snap to an endpoint grip. Look at the coordinate display on the left of your status bar and see if it makes sense to you. Try this at other points. What you have just done is created a simple object in 3D drawing space. It has length, depth and height. Get used to looking at it, and review what you did to create it.

Use the **3DORBIT** command to spin you view and how it looks from different angles.

Save your drawing under the name **wireframe1.dwg**.

#### EXERCISE 2

Start a new drawing using the acad.dwt template.

Create 3 layers called **SIDES**, **SEAT**, and **BACK** and give them different colors. Make <u>back</u> your current layer. Multiple layers make your drawing easier to work with - trust me.

Look at the drawing below. This is the basic chair shape that you will be drawing a few different ways in this course. For this lesson, you'll be using rectangles. Switch to your SE Isometric view.



Start with the back. Draw a **RECTANGLE** to define the plan view of the back of the chair. Copy it 34" in the Z-axis and then connect the corners with vertical lines.

Next draw the sides (on the sides layer). Use the same method you used for drawing the back (except copy them up 24")

Once you have drawn one side, you can copy or mirror it to make the other side.

Finish up by drawing the seat. This will take a bit of thought as you have to make it fit between the sides. Start by drawing the rectangle for the seat in the correct location, then move it up 12" so that it is at the correct elevation. After that, copy it up 4" and connect the corners.

Save the file as wirefame2.dwg in your CAD folder.

Try to get the viewport configuration you see below using tiled viewports



You don't need to dimension this drawing, but use the **Distance** command to confirm that the sizes are correct.

When you are sure that your drawing is correct, save it again for later use.

From these simple exercises, you can see that the world of 3D (so far) isn't really different from 2D. You used the same rectangle, line, copy, move commands that you would drawing in a 2D world. The only difference is that you added the Z coordinate to give your point 'height'. Once again, make sure you are familiar with what happened in this lesson before moving on to the next. Redo it if you have to, but get used to the concept of height.

#### Video: Creating a Wireframe Model

Extra Practice: Create a 3D wireframe model of this drawing.

Extra Practice (B): Create a 3D wireframe model of this drawing.



#### Take the Lesson 3-5 Quiz

If you copy an object 2 units up in the Z axis, can you see it from the plan view?

Yes, if you zoom in close enough

No, it is on top of the original

Yes, it will look different than the original object



### For Further Thought:

Congratulations, you have just drawn something in 3D! These were very simple examples to get you started, but you are now on your way to drawing 3D parts and by the end of these tutorials you will be able to draw a 3D house, add materials, add lights, render a realistic model and animate it. But it's best to start small and simple. I've always had fun working 3D. The more I learned about 3D, the less I wanted to draw in 2D. But when I started professional drafting in 1997, there was very little call for 3D. Still, there were times when I needed 3D to calculate angles and connections to use in my 2D drawings.

If virtual reality becomes commonplace in the coming years, 3D skills will become more valuable as you'll be able to apply your knowledge in ways we can't imagine today.

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# **CONTINUE TO THE NEXT LESSON: 3-6**



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Last Update: 02/25/2019 18:19:38

### Topics covered in this Lesson:

Line Thickness - Simulated 3D

In this lesson you'll create the same chair you drew in the wireframe tutorial by drawing lines and then giving them *thickness*. Think of thickness as the height of the line or how tall it is (like a fence in your yard). This is different from the width of the line that you learned in the **polyline** lesson or lineweight (which is a display property). This is a quick, simple way of achieving some degree of 3-D feel. Once again, remember that in some instances, simple concepts may do the job. This is also a good technique for AutoCAD LT users, who don't have the option to draw in true 3-D.

Start a new drawing using the acad.dwt template.

Create 3 layers called **SIDES**, **SEAT**, **BACK** and give them different colors. Make **back** your current layer.

Below are the dimensions for this simple chair (it's the same one you drew in the wireframe tutorial).


Draw rectangles to create the back, sides and seat on their respective layers (as if you were drawing the top view). Do not worry about height, thickness or anything but the basic outline of the top view.

Select the rectangle on the Back layer and right click and select 'Properties".

You'll see the Properties Palette appear and it should look like the one below.

Leave the palette open for the rest of this exercise.



For this lesson, there are two important areas to look at in the properties palette. Make sure that your line is located at an elevation of **Zero**. From there you want to go into the **Thickness** text box and enter **34**. This will give you a polyline that is based where you drew it and will be 34" thick, meaning that it will extend 34" in the positive Z-axis. (Entering a negative number will extend the thickness in the negative Z-axis.)

Change the properties of the sides and give them a thickness of 24".

Give the seat a thickness of 4 inches. The seat should have an elevation of Z=0.0000 as drawn. To move it up to the correct location, enter the move command and give it a displacement of @0, 0, 12.

Look at your drawing in the SW isometric 3-D view and it should look like the one you drew in the wire-frame exercise. There is one major difference between the two models. As stated earlier, wire-frame models do not allow you to hide lines that you might not see in the real world. With your cursor in the SW isometric viewport, enter the command **HIDE**. You should see that your chair now looks like the one below.



You'll see that it is now starting to look like a chair. It's not perfect yet. You can still see the sides through the seat and the tops of the sides and back are hollow. Once again - think of a fence around your yard.

Save the drawing under the name **chair\_line\_thickness.dwg** in your CAD folder (you'll use it in the next lesson).

So far you have learned two ways to construct a "3-D-like" object and still haven't learned any new drawing commands. These methods use 2D commands to achieve 3-D appearances. Another advantage of these methods is that they can be used in AutoCAD LT, which doesn't have any true 3-D capabilities at all.

Extra Practice: Create a line thickness drawing of this drawing.

Extra Practice: Create a line thickness drawing of this drawing.

## Video: Line Thickness in AutoCAD

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## Take the Lesson 3-6 Quiz

Is line thickness a true 3D method of drawing?

No

Yes, but only in AutoCAD LT

It depends upon the layer



## For Further Thought:

Sorry for taking you back to 2D, but I want to give you as many tools as possible for your shiny new 3D toolbox. This isn't something you're going to use a lot, but when you need it, you'll be very thankful you have this knowledge.

Learning how to use the right tool at the right time is sometimes more important than knowing about all of the tools. Learning how to get the job done fast and accurately is important. You employer doesn't care about how you get a job done, just that you completed it before the deadline and that it was done right.

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## **CONTINUE TO THE NEXT LESSON: 3-7**



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## Topics covered in this Lesson:

#### **Regions | 3D Surfaces**

In this lesson, you'll begin using true 3-D commands. First you will create a 2-D region and then extrude it into a 3-D solid. Then you will add 3-D surfaces to the tops of your **previous** drawing (line thickness) to close the back, sides and seat.

Begin a new drawing with the acad.dwt template.

Create 3 layers called **SIDES**, **SEAT**, and **BACK** and give them different colors.

Draw the outlines (using rectangles) of the back, sides and seat as you did in **Lesson 3-6**, but don't add line thickness.



You'll start by creating a region for the back of the chair. Make sure that back is your current layer. (Any region you create will be on the current layer, regardless of which layer the object was on when you selected it.) Start the region command by entering **REGION** (or **REG**) at the command line. Pick the polyline that will become the back of the chair and press enter. Your command line should look like this:

```
Command: REG
REGION
Select objects: <SELECT A RECANGLE> 1 found
Select objects: <ENTER>
1 loop extracted.
1 Region created.
```

If the Region wasn't created, it usually means that you have a gap in the shape. Polylines must be closed to be used for Regions. Repeat this for the other polylines representing the 2 arms and the seat, ensuring that you have the correct layer current. At this point, you have four 2D regions. The next step is to **extrude** them into 3-D solid objects. Make BACK the current layer and start the **EXTRUDE** command. Pick the region on the back layer. Look at the command line. You will be asked a few things:

Command: EXT EXTRUDE Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid Select objects to extrude or [MOde]: <SELECT REGION> 1 found Select objects to extrude or [MOde]: <ENTER> Specify height of extrusion or [Direction/Path/Taper angle/Expression] <4>: 20

The first thing that AutoCAD needs is the height of the extrusion. This is similar to the thickness that you gave the polylines in <u>Lesson 3-6</u>. Remember that if you give a positive number, it will extrude in the positive Z-axis. If you give a negative number, it will extrude the region in the negative Z-axis.

**Note:** In earlier versions you were asked for the 'Extrusion Taper Angle'. In most cases this will be the default of zero. If you're using an older version and see this, press **<ENTER>** to accept the default.

**Note:** You can also extrude directly from rectangles, circles and closed polylines, but I wanted to show you the Region command. If you have 4 lines that make up a shape, you can't extrude the lines, but you can use the Region command to create an extrudable object.

**EXTRUDE** the sides and seat. Remember to move the seat up 12" in the positive Z axis if you haven't already. Check that the model is correct by viewing it in the SW isometric view. Type **HIDE** (**HI**) and you should see a solid chair like the one below.



Now you can see how solids are a good way to go in some cases. They will usually give you the most accurate way of viewing the model. You can also get a lot of information out of a solid. Start the Mass Properties commands (MASSPROP). Click on one of the solid objects and press <ENTER>. You'll see that AutoCAD can give you a lot of information about it. A lot of these are useful in engineering applications, but the volume is something that may be needed in many different instances. As you'll see in the next lessons, you can do a lot with solids.

Save your drawing under the name: chair\_extruded.dwg

## Video: Extruding in AutoCAD

## **Adding Surfaces**

Open your **chair\_line\_thickness.dwg** drawing and immediately save it under a new name: **chair\_3D\_face.dwg** 

What you're going to do is add 3-D faces to the tops of the back, sides and sides. Begin by viewing your model in the SW Isometric view. Type **HIDE** to see where you need to cap off the extruded lines. Type **REGEN** or **RE** to regenerate your screen so you can see all the corners again.

Make sure that **BACK** is the current layer. Type **3DFACE** to begin the command. You will be asked to select some points. Make sure that your endpoint Osnaps are on.

When asked to pick the points, select them in the order as if you were drawing a rectangle. Do not go from one corner to an opposite one. After you have picked the fourth corner, press (enter). Your command line should look like this:

Command: 3DFACE Specify first point or [Invisible]: <1> Specify second point or [Invisible]: <2> Specify third point or [Invisible] <exit>: <3> Specify fourth point or [Invisible] <create three-sided face>: <4> Specify third point or [Invisible] <exit>: <ENTER>



Repeat the command to add 3-D faces to the top of the sides and the seat. 3D faces are great for making odd shapes that include any four lines (or arcs) that are touching).

Enter the **HIDE** commands to check that it was done correctly. If it all worked, you should see an image that looks identical to the solid model. You'll notice that the 3D face covers what would be the opening at the top of the arms and the back. If it was done incorrectly, there may be a line running down from the back corner of each piece. **Save** the file.

3DFace objects can be extruded. You can also create more complex shapes using this command by continuing to select points. Here's another exercise to show how this works. Draw the shape below:



Look at the next image and see where you will be picking points. The goal is to cover this area in (three sided) 3DFaces using the command just once.



Start the 3DFACE command and pick the points on the corners as shown above. After picking the 11th point, press enter to create the last 3DFace and end the command. You can now extrude the faces for more practice.

Now you have created some true 3-D objects. Take some time to think about how much work was involved in each method: Wireframe, line thickness and extruding. Think about instances where one might be better than the other. Review the methods that you have used so far.

### Video: Faces & Extruding Text in AutoCAD

**Extra Practice:** Create a 3D solid model of <u>this drawing</u>. Start with the bottom profile and create a region, then extrude the region. Save this and other exercises for using in Lesson 3-9 (adding materials).

Extra Practice: Create a 3D solid model of this drawing.



## Take the Lesson 3-7 Quiz

What shape can you create a region from?

Line

Arc

Rectangle



## For Further Thought:

Here are the 'power tools' of 3D. You might have noticed that using these tools is fairly simple, but these tools are the foundation of much of your 3D work. As you work through more of my AutoCAD tutorials, you'll see how extruding plays a big part.

Learning how to create a basic 3D shape and then modifying it is very similar to the 2D process of drawing lines and rectangles and then modifying them.

Learn to see the steps ahead when building a 3D model and chart the best path. This takes a LOT of practice and experience, but learning to see the path will save you time, so take this approach even with simple models. Do you like what you just learned? Support this site with a Donation via Paypal:



## <u>CONTINUE TO THE NEXT LESSON: 3-</u> <u>7a</u>



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## Topics covered in this Lesson:

#### Extruding | Lofting

The purpose of this lesson is to look further at the **EXTRUDE** command. As you saw in **Lesson 3-7**, it can be used to create a 3D solid from a closed 2D shape. Two other ways you can extrude (which you may have seen as options on the command line) are to taper the extrusion and the extrude a shape along a path. If you need to, you can also combine the two options and extrude along a path while tapering the shape (shown below).



One example where you can use extruded paths is to represent pipes in a drawing. You may not use the tapered path option often, but at least you know it's there.

## EXTRUDING ALONG A PATH

Draw a **POLYLINE** from 0,0 to 120,0 to 120,120 to 240,120 to 240,0 and then press **<Enter>** to finish the command.

Do a **Zoom** > **Extents** to see the polyline and then zoom out a little more. Your line should look like this:



Next you will put a 24 unit radius on all the corners. The easiest way to do this is using the **P**olyline option of the **FILLET** command.

```
Command: FILLET <ENTER>
Current settings: Mode = TRIM, Radius = 0.0000
Select first object or [Polyline/Radius/Trim/mUltiple]: R
Specify fillet radius <0.0000>: 24
Select first object or [Polyline/Radius/Trim/mUltiple]: P
Select 2D polyline: <SELECT THE POLYLINE>
3 lines were filleted
```



What you're going to do next is extrude a circle a long the polyline - or to be more accurate, the path of the polyline. This would be one way of drawing pipes in 3D. For this example, you'll draw a pipeline with a diameter of 12 units.

Next draw **CIRCLE** at the bottom right end of polyline. Use a diameter of 12 (radius of 6). Once you have that, you need to rotate it in 3D. This is covered in the <u>next</u> <u>lesson</u> as well. To do this, you will select the circle, select the axis you want it rotated around and then choose the angle.

Command: ROTATE3D Current positive angle: ANGDIR=counterclockwise ANGBASE=0 Select objects: <SELECT THE CIRCLE> 1 found Select objects: <ENTER> Specify first point on axis or define axis by [Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: X Specify a point on the X axis <0,0,0>: <SELECT THE BOTTOM RIGHT END OF THE POLYLINE - Make sure your Osnaps are on for endpoints> Specify rotation angle or [Reference]: 90 <ENTER>

Your circle should have rotated 90 degrees and now you are looking at the side of it so the circle appears to be a line as shown in the image below.



Now comes the easy part. Next you will **EXTRUDE** the circle along the path of the polyline.

Command: EXT EXTRUDE Current wire frame density: ISOLINES=4 Select objects: <SELECT THE CIRCLE> 1 found Select objects: <ENTER> Specify height of extrusion or [Path]: P Select extrusion path or [Taper angle]: <SELECT THE PLINE> Note: After the Extrude command, the polyline will still be there. If you need to keep your drawing clean, remember to erase the path if you don't need it any more. To see how it looks, view the object in the **<u>SW Isometric view</u>**, and use the **HIDE** command. It should look like this:



This is just one option available with the Extrude command. Try it on other paths and see how it works. You will find that if your circle is too large, it may not be able to be extruded on polylines with tight corners. You might then see this error on the command line: **Unable to extrude the selected object.** 

Any object that can be extruded can be extruded along a path. A path can be any **open** object such as lines, arc, polylines, splines, etc.

For more practice, try to create a cord for your lamp (Lesson 3-8) using a Spline as the path. You can also extrude 2 circles along a path (make one one circle smaller) and then **SUBTRACT** the smaller diameter extrusion from the larger to create a hollow pipe.

### EXTRUDING WITH A TAPER

Extruding along a taper gives you another option in your 3D toolbox. Here is an example of how it is done:

Draw a **RECTANGLE** 100 units by 100 units.

**EXTRUDE** the Rectangle 50 units high with a taper angle of 45°. Here are the commands needed.

Command: EXT EXTRUDE Current wire frame density: ISOLINES=4 Select objects to extrude: <SELECT THE RECTANGLE> 1 found Select objects to extrude: <ENTER> Specify height of extrusion or [Direction/Path/Taper angle] <1.0000>: T Specify angle of taper for extrusion <0>: 45 Specify height of extrusion or [Direction/Path/Taper angle] <1.0000>: 50

Here is what you should have (a pyramid):



Use the **3DORBIT** command to view it at different angles (Click on the screen, hold the button down and move the cursor around the screen). Try extruding different shapes with various taper angles for more practice.

These options give you a lot of versatility within one command. You may not use these options very often in your everyday drafting, but they're good to know.

## LOFT COMMAND

New in AutoCAD 2007 (and long overdue) is the **LOFT** command. The loft command is similar to the extrude command, but much more versatile. Instead of extruding a

single shape, the loft command allows you to extrude several shapes and make one continuous object.

Here's an example of how it works, I have drawn several shapes and aligned them with the same center. Next, they were copied above and below the largest circle. The goal is to create a smooth object from the 7 profiles shown below.



Here is a front view of the objects before and after the loft command:



The example above is very simple, but think of how you can create complex shapes using this simple command.

#### USING THE LOFT COMMAND

The goal of this exercise is to create a "twisted Cube" - a 20x20x20 cube that looks like the top was twisted 45°. We're going to start with a simple step. Draw a square 20x20. Copy it in the same place. Rotated the 2nd square 45° using the midpoints of the square as the base point. Finally, move the 2nd square up 20 units. You should have something like this from the SW view:

From left to right, the original square, the copied and rotated square and the moved 2nd square. Copy the 2 rectangles over (out of the way) to use in another exercise.



Next you will create a lofted object from the 2 rectangles using the defaults.

```
Command: LOFT
Current wire frame density: ISOLINES=4, Closed profiles
creation mode = Solid
Select cross sections in lofting order or [POint/Join multiple
edges/MOde]: <PICK BOTTOM SQUARE> 1 found
Select cross sections in lofting order or [POint/Join multiple
edges/MOde]: <PICK TOP SQUARE> 1 found, 2 total
Select cross sections in lofting order or [POint/Join multiple
edges/MOde]: Specify opposite corner: <ENTER> 0 found
Select cross sections in lofting order or [POint/Join multiple
edges/MOde]: <ENTER>
2 cross sections selected
Enter an option [Guides/Path/Cross sections only/Settings]
<Cross sections only>: <ENTER>
```

After the command, your two squares will be turned into one lofted 3D Solid.

Now something doesn't look quite right to me. I would expect that corners of the squares would be even so that the lofted square looks like a 'twisted cube'. Instead, you can see lines going from the middle of the bottom square to a corner of the top square. Use the **HIDE** command to see how it looks. It looks a little strange, but you can make out that it has some odd shapes.



Take the pair of squares that you copied over earlier and **draw lines** from lower corner to upper corner as shown in the drawing below:



Now start the **LOFT** command again. This time you'll try something different. Select the squares again as your cross sections, then instead of accepting the default, use the Guides option G. Now **select the four lines** which will guide the loft to create a shape that better represents what you want.

Command: LOFT Select cross-sections in lofting order: <SELECT SQUARE> 1 found Select cross-sections in lofting order: <SELECT OTHER SQUARE> 1 found, 2 total Select cross-sections in lofting order: <ENTER Enter an option [Guides/Path/Cross-sections only] <Crosssections only>: G Select guide curves: <SELECT LINE> 1 found Select guide curves: <SELECT LINE> 1 found, 2 total Select guide curves: <SELECT LINE> 1 found, 3 total Select guide curves: <SELECT LINE> 1 found, 4 total Select guide curves: <SELECT LINE> 1 found, 4 total Select guide curves: <ENTER>

Compare the 2 objects, the 2nd one should look more like the goal of a twisted cube.



To change how you view things, use the **VISUALSTYLES** command to open the Visual Styles Manager palette. This command allows you to quickly change settings

for how you view 3D objects that previously needed the knowledge of several system variables.

Use the settings highlighted below and press the "Apply to Current Viewport" button (or double click on any of the colored squares at the top. You should now have a clear view that illustrated the differences between the first lofted object and the second one that used guides.





Try creating your own lofted 3D Solids. Anything that can be extruded can be lofted so any closed shape will work. Try lofting a rectangle to a circle. Try exploding a lofted object. There are many other options to this command, but in a effort to be brief and cover the basics, I will not go further into this command. (To explore further, select the Settings option in the command.) Remember to pick the cross sections in the correct order, add guides to better define the shape.

In review, you can see how extruding and lofting are two simple commands that give you a lot of power in the 3D world. Get familiar with extruding, as it is a great way to build things in 3D. Lofting will allow you to create more 'organic' shapes that previous versions of AutoCAD. Think of how you could use cross sections of a ship's hull to create a 3D model.

**Extra Practice:** Draw <u>this object</u>. Start with a 3DPolyline (3DPOLY) and then extrude the circle along the path of the polyline.

**Extra Practice:** Copy <u>this drawing</u> that shows the dimensions on the left and an isometric view for reference. Then create a lofted object (using the default setting) to look like <u>this</u>.

**Experiment:** What happens when you extrude a line along a line? Try this and use the LIST command see what type of object is created.

### Video: Extrude (Path & Taper) - Loft Command

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Take the Lesson 3-7a Quiz

Which object(s) can be extrdued?

Rectangle

All of these

Region



## For Further Thought:

These new tools open up a lot of options in your drafting. Using AutoCAD to make complex shapes is sometimes challenging, but it's good to know what your options are.

Think of how you can draw a series of profiles (slices) to create a shape for a boat or car. The more profiles you have, the more accurate your model. A lot of great 3D drafting takes a lot of time to get the details accurate. In some cases you might not have the time for complete accuracy (background items in a rendered scene, for example) so it's up to you as the designer to decide how much time gets spent on these things.

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## **CONTINUE TO THE NEXT LESSON: 3-8**



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Last Update: 02/25/2019 18:20:20

## Topics covered in this Lesson:

#### Revolving Objects | Revsurf | Revolve

Below is a sample drawing showing 2 different revolved objects (lamp and lampshade). It will also be your goal in this lesson to duplicate similar objects.



So far you've only worked with very basic blocks. Suppose that you need to draw something other than a rectangular cube. You did some new shapes in the previous lesson while extruding and lofting.

AutoCAD gives you two commands for those times when you need to draw cylindrical objects. One (**revsurf**) will give you a complex model comprised of a 3-D surface made up of many facets. The other (**revolve**) will give you a solid object. The method that you use will once again depend on what you need it for.

Take a look at the lamp on this page. This is an example of two different types of objects requiring two types of object construction. The lampshade is a 'hollow' object. Essentially it's just a surface. The lamp base is a solid object. The **REVSURF** command was used to create the lampshade, while the **REVOLVE** command was used to create the base. It's not the greatest looking lamp ever, so in this lesson, you'll be designing a lamp base and a lampshade. You'll start by defining half the profile of each object, then revolving them. This will also be good practice for viewing your 3-D model.

Begin a new drawing using the acad.dwt template.

Create 2 layers called **SHADE** and **BASE** and give them different colors.

Make **BASE** your current layer.

Start the **polyline** command. Begin the profile for your lamp base. Use Ortho mode to draw a a backwards "" shape with the vertical line 10 units up and the horizontal lines can be whatever you like (you're the designer now).

Next draw a spline to connect the two ends of your polyline using Osnaps.

Hint, when ending the spline command, you will be asked for the start and end tangency - choose the endpoint just to the right of your ends in the vertical line.

Draw a short vertical line as shown in step 3 (make sure Ortho mode is ON).

Make **SHADE** your current layer Draw a short angled **line** as shown in step 4 (this is the beginning of the shade.)

Make sure **BASE** your current layer. Start the **REGION** command and create a region from the lines that makeup the base. Don't include the vertical line at the top.



You have drawn everything you need for this lesson and will use modifying commands to complete the lesson.

First you'll create the lampshade. Before you do this, though, you'll have to set two of AutoCAD's system variables (SURFTAB1 & SURFTAB2). These variables control how many facets you'll have in your surface. The default is 6, which will give you a very chunky looking shade (like a hexagon instead of a circle). The number you pick will also influence how fast your computer can display the object as well as how round the shade will appear. The sample on the previous page had the SURFTAB1 variable set to 24 to give the lamp 24 sides.

To change these, type in **SURFTAB1** at the command prompt enter **24**. AutoCAD will show you the current setting and give you a chance to reset it. Set the variable and then set **SURFTAB2** enter **2** for this variable. Since the vertical shape is a straight line, you only a setting of 2 (the minimum).

Now you're ready to create the lampshade. Start the **REVSURF** (Revolve Surface) command. You will be asked to select the **object to revolve**. This is the line that represents the lampshade. Then you are asked to select the **axis of revolution**. Pick the vertical line that you drew up from the base profile. Accept the defaults of 0 for the start angle and 360 for the included angle. This will rotate your line a full 360 degrees.

```
Command: REVSURF
Current wire frame density: SURFTAB1=24 SURFTAB2=2
Select object to revolve: <pick the lampshade line>
Select object that defines the axis of revolution: <pick the
vertical line>
Specify start angle <0>: <ENTER>
Specify included angle (+=ccw, -=cw) <360>: <ENTER>
```

You'll see that the lampshade looks like a lampshade now. Use **3DOrbit** to view it.

### Revolve

Next you will create the base. Start the **REVOLVE** command. You'll be asked to select objects. Pick the region for the lamp base. Next you're asked to select the axis of rotation. With your endpoint Osnap on, pick the top and bottom of the vertical line. Accept the default of **<full circle>** for the angle of revolution. This will revolve the profile around the vertical line 360 degrees and create a **solid** object.

```
Command: REV REVOLVE
Current wire frame density: ISOLINES=4
Select objects: <SELECT THE REGION> 1 found
Select objects: <ENTER>
Specify start point for axis of revolution or
define axis by [Object/X (axis)/Y (axis)] <OBJECT>: <ENTER>
Select an object: <SELECT THE VERTICAL LINE ABOVE THE BASE>
Specify angle of revolution <360>: <ENTER>
```



## VIEWING YOUR LAMP

For the next few lessons, you should switch to the 3D basics workspace if you haven't already. Look for the icon in the bottom right of the AutoCAD screen.



Use the **HIDE** (**HI**) command to see that you really have 3D objects and that the shade blocks the top part of the base.

If you switch to your SW Isometric view, you'll see that the lamp is 'on its side'. To arrange it to be sitting on its base, you will have to use the ROTATE3D command. Begin the command by typing **ROTATE3D** and select the objects that you want to modify and press enter. By accepting to default of 2points, you are going to tell AutoCAD what axis you want the lamp rotated about. Pick the points as shown in the

example below. Make sure you have your Osnaps (Quadrant) on. Refer to the <u>right</u> <u>hand rule</u> for the correct rotation angle.

Command: ROTATE3D Current positive angle: ANGDIR=counterclockwise ANGBASE=0 Select objects: Specify opposite corner: <PICK OBJECTS> 2 found Select objects: <ENTER> Specify first point on axis or define axis by [Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: <PICK POINT 1> Specify second point on axis: <PICK POINT 2> Specify rotation angle or [Reference]: 90 <ENTER>



Now that you have your own 3D model, try some of the viewing options described in **Lesson 3-4**.

Save your drawing as **cool\_3d\_lamp.dwg** as you will need it in a later lesson. You'll be adding materials to it and rendering a visualization of it soon.

Extra Practice: Create this <u>wineglass</u> by drawing this <u>outline</u> and revolving it.

**Extra Practice** Create this <u>tire</u> by by drawing this <u>profile</u> and revolving it along the centre line.

Video: Revolve and Revsurf



## Take the Lesson 3-8 Quiz

Which command will give you a solid object?

Revolve		
Revsurf		
Revolution		

## For Further Thought:

When I taught this lesson in a classroom, it was always a favourite of the students as it started to give them some idea of what 3D was all about. It allowed them to use some simple tools to create a custom shape that didn't have square corners. Try to think of other shapes and objects you could draw using these tools. Then draw them for practice. Can you make a simple coffee mug with a handle? These are the type of problems you should start thinking about. Learning the tools is one thing, but applying them to create new designs is the trick.

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## **CONTINUE TO THE NEXT LESSON: 3-9**



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### Topics covered in this Lesson:

Introduction to Materials and Rendering

For the next few lessons, you should switch to the 3D modeling workspace. Look for the menu in the top left of the AutoCAD screen. Older versions may have this in the bottom right. You should also have your Ribbon in view if you don't already as there
are many commands that are easier to access this way instead of via the command line. A **video** of this tutorial is at the bottom of the page.



One of the more interesting aspects of working in 3-D is that you can visualize what your design will look like. You have so far used the hide and shade commands to give you some idea towards how the final piece will look. The next step is to learn about the **RENDER** command. This command is the most powerful one for viewing your 3D objects. Using **render**, you are able to add realistic lighting and materials to get the most realistic view of what you're designing.

You can render objects that don't have materials applied to them, but it won't look as realistic as if you have material added.

The first two blocks shown here are examples of the usual wire-frame view before and after the **HIDE** command.



These two blocks show how the render command can show the object two ways. The block on the left was rendered without a material applied, while the one on the right has a white ash material applied.



Applying the materials is a relatively easy process, getting them to look exactly the way you want them is skill in itself. Once the materials are added, getting the lights and shadows to look realistic is another task that many people prefer to complete in a program other than AutoCAD, although AutoCAD is getting better at this with each release.

Follow these steps to get an basic, accurate rendering:

- Draw the object using solids or surfaces
- Apply the materials
- Render the scene

Once your objects are drawn, you have to decide which materials you want to use. AutoCAD comes with a basic **materials library** that you can use to apply to your objects. This may have been an option when you installed the program, and this tutorial will assume that you have them. If you don't have them, jump ahead to the tutorial about <u>creating materials</u> and come back - I'll wait.

Later lessons will teach you how to <u>create new materials</u> and <u>map them to</u> <u>objects</u>. This lesson is designed to allow you to quickly and easily apply materials to your objects. Start by drawing a basic cube 20x20x20 - this will be your object for testing materials. Go to the Ribbon and find the Visualize Ribbon. Change the Visual Style to **Realistic** and then click on the **Material Browser** button.



Clicking on the Materials Browser (or typing **RMAT**) will open the Material Browser Palette (shown below).

Now that you have your object (the cube) and a list of materials, you're ready to go.

Materials Palette for AutoCAD 2016

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		Name   Type Category							
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					dium Blue	Ceramic	Ceramic: Tile		-11
				1.5in Squ	late Blue	Ceramic	Ceramic: Tile		- 1
		1.5in Square - Tan     12in NonBurgundy		iare - Tan	Ceramic	Ceramic: Tile		- 11	
				Burgundy	Masonry	Masonry: Brick		- 1	
				12in Run	Burgundy	Masonry	Masonry: Brick		- 11
				12in Unif	ng - Gray	Masonry	Masonry: Brick		- 1
SER				1 1in Squa	re - Ivory	Ceramic	Ceramic: Tile		- 11
SOW				1in Squa	saic Blue	Ceramic	Ceramic: Tile		- 1
LS BI			100	1in Squa	rsaic Gray	Ceramic	Ceramic: Tile		- 11
ERIA				2in Squa	re - Salmon	Ceramic	Ceramic: Tile		
MAT				2in Squa	res - Beige	Ceramic	Ceramic: Tile		-
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The easiest way to apply a material to an object is to click on the material you want to use and then drag it over to the object.

As you have set your visual style to "Realistic", you'll see the material cover the object like a skin. It might not be the right scale or exactly what you envisioned, but the material is applied to object now - and that's the easy part. Zoom in on your object to see the material a little closer if you need to.

# **BASIC RENDERING**

There are few ways to render, and for this exercise I will suggest this method. Look at the Render Ribbon again and under the "Materials Browser" button, you'll see this:



There are a few settings you can select before rendering, but for now, just set it to "**Medium**" and "**Render in Window**". This will create a decent rendering in a separate window from your main drawing space. Now type **RENDER** <enter> to start the rendering.



This window allows you to preview the rendering, zoom in and out, print and save the rendered image to a file. Click on the "Save" button to save your first rendering.

If you're going to be trying a lot of renderings, you should open the render settings palette by clicking on the small arrow shown below.

Kender   Kender   Render in:   Window   Render Size:   800 x 600 px - SVGA   Current Preset:   Medium   Preset Info   Name:   Medium   Description:   Applies 5 rendering levels   Render Duration   Until Satisfactory   Image: Second Participation   Value: Second Participation   Value: Second Participation   Medium   Description:   Applies 5 rendering levels   Render By Level:   Second Part   Ights and Materials   Render Accuracy:   Low   Draft   High	Render to Size	Medium Render in Wind Prender 👻	low	Render in Ren Cloud Gall	der
		×  + *	Render Size: Current Preset: Preset Info Name: Description: Render Duration Until Satisfa Render By L Render By T Lights and Mate	800 x 600 px - SVG/ Medium Medium Applies 5 rend a ctory evel: ime: irials cy: Cy: Draft	A

As you can see there are a few options in this palette, but for now I want to look at the two that are boxed in red. These settings allow you to crop to an area and then render the scene in your viewport. We'll look at more options in tutorial <u>3-17</u>.

One advantage of rendering a separate window is that you can continue working on your drawing while the render completes in the background. Depending upon how powerful your computer is, the rendering could slow everything down regardless.

### Review

This has just been an introduction to the rendering process. As you can see adding materials isn't needed unless you plan on rendering the scene. Using just the example above, you can get a nice looking presentation done using only a few simple steps. Next you would need to add lights, shadows and possibly custom materials. For me, this where AutoCAD gets fun and makes up for the steep learning curve when starting out.

**Extra Practice:** Add materials to any of the objects you created in Lessons 3-6, 3-7, 3-7a.

# Video: Adding Materials and Basic Rendering

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#### Take the Lesson 3-9 Quiz

Does the Materials Palette allow you to add materials to your objects?

No

Only with the Render Command

Yes



# For Further Thought:

I remember learning these techniques back in 1995 and staying long after class to keep working on them. It was fascinating to me. Think about the ultimate goal of this process. You create a 3D model, add a realistic material to it and then create an image that you can show someone. This person doesn't need the ability to read a blueprint or an orthographic projection drawing - they just look at the rendered model.

A majority of images in the Ikea furniture catalog are 3D renderings. It's easier for them to render the image (as they already have the drawing) than to hire photographers and deal with digital photos. That's the power of 3D.

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# <u>CONTINUE TO THE NEXT LESSON: 3-</u> <u>10</u>



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Last Update: 02/25/2019 18:20:51

### Topics covered in this Lesson:

#### Box | Sphere | Cylinder | Cone | Wedge | Torus | Pyramid | Polysolid

For the next few lessons, you should switch to the 3D modeling workspace. Look for the icon in the bottom right of the AutoCAD screen.



## What is a Primitive Solid?

A primitive solid is a 'building block' that you can use to work with in 3D. Rather than extruding or revolving an object, AutoCAD has some basic 3D shape commands at your disposal. From these basic primitives, you can start building your 3D models. In many cases, you get the same result from drawing circles and rectangles and then extruding them, but doing it one command is generally faster. Using these with Boolean operations can be a very effective way of drawing in 3D. There are eight different primitives that you can choose from and are on the Home > Modeling Tool Panel (when in the 3D workspace).

SHAPE	COMMAND	ICON	DESCRIPTION
BOX	вох	Box	Creates a solid box after you provide 2 opposite corners and a height.
SPHERE	SPHERE / SPH	Sphere	Creates a solid sphere from a center point and radius.
CYLINDER	CYLINDER / CYL	Cylinder	Creates a straight cylinder from a center point, radius and height.
CONE	CONE	Cone Cone	Creates a tapered cone from a center point, radius and height.
WEDGE	WEDGE / WED	Wedge	Creates a triangular wedge from 2 opposite points.
TORUS	torus / tor	O Torus	Creates a torus (donut shape) based on center point, radius and tube radius.
PYRAMID	PYRAMID / PYR	Pyramid	Draws a solid object with a polygon (3-32 sides) base that rises to a central point.
POLYSOLID	PSOLID	Polysolid	Draws a solid object with width and height as you would draw a polyline.

#### (Click on the ICON below to go to the command)

You can use primitives to either begin building a model or it can be a finished object on its own. Many of these commands are similar to 2D commands, except with an extra coordinate in the Z axis. Here is a summary of working with these commands.

BOX



Think of a box as an extruded rectangle. It has width, height and depth. It is created by establishing a starting corner and then establishing a second corner by either picking or giving relative coordinates.

Here is an example of this:

```
Command: BOX
Specify corner of box or [Center]: 2,3,4
Specify corner or [Cube/Length]: @5,7,10
```

This draws a box that is 5 units in the X-axis wide, 7 units long in the Y-axis and has a depth of 10 units in the positive Z-axis with one corner located at 2,3,4.

Here is another way of drawing that same box:

```
Command: BOX
Specify corner of box or [Center]: 2,3,4
Specify corner or [Cube/Length]: @5,7
Specify height: 10
```

Using this method, you establish the first corner as before, but only enter the X and Y coordinates of the opposite corner. AutoCAD will then prompt for the height.

Another way of drawing a box is to establish where the center of the box will be:

```
Command: BOX
Specify corner of box or [CEnter] <0,0,0>: C
Center of box <0,0,0>: <ENTER> or <PICK A POINT>
Specify corner or [Cube/Length]: @2,3,4
```

This draws a box that is 4x6x8 based about the center of 0,0,0.

If you want to draw a perfect cube, you can use this option:

```
Command: BOX
Specify corner of box or [CEnter]<0,0,0>: <pick point>
Specify corner or [Cube/Length]: C
Length: 4
```

This draws a cube with all sides equal to 4 units based of off a picked point.

The last way of drawing a cube allows you to enter the Length, Width and Height as separate distances, and not based on coordinate points.

```
Command: BOX
Center/<Corner of box> <0,0,0>: <pick point>
Cube/Length/<other corner>: L
Length: 5 <X AXIS>
Width: 4 <Y AXIS>
Height: 6 <Y AXIS>
```

Of course you can also draw a box by picking two opposite corners with your mouse. This is useful for filling in areas and can be very quick. Make sure to use your Osnaps.

#### **SPHERE**



A Sphere is a globe-like shape. It is very similar to drawing a circle in that you pick a center point and then input either the radius (default) or diameter. Both methods draw the same sphere in the following examples:

```
Command: SPHERE
Specify center point or [3P/2P/Ttr]: <PICK POINT>
Specify radius or [Diameter] <2.3756>: 6 <ENTER>
```

Command: SPHERE Specify center point or [3P/2P/Ttr]: <PICK POINT> Specify radius of sphere or [Diameter]: D Specify Diameter: 12

Both of the above methods will give you the same result.

You also have the options of selecting 3 Points, 2 Points, or using 2 tangents and radius (TTR).

### CYLINDER



A cylinder is just like an extruded circle. Creating one is very similar to creating a circle, except that you are giving depth to it.

These two examples would draw the same cylinder:

```
Command: CYLINDER
Current wire frame density: ISOLINES=4
Specify center point for base of cylinder or [Elliptical]
<0,0,0>: <PICK POINT>
Diameter/<Radius>: 2.5
Center of other end/<Height>: 1.2
```

Command: CYLINDER Current wire frame density: ISOLINES=4 Specify center point for base of cylinder or [Elliptical] <0,0,0>: <PICK POINT> Specify radius for base of cylinder or [Diameter]: D Diameter: 5 Specify height of cylinder or [Center of other end]: 1.2

CONE



Drawing a cone is the same as drawing a cylinder, except that the resulting object tapers smoothly from the bottom to a point at the top.

Command: CONE Current wire frame density: ISOLINES=4 Specify center point for base of cone or [Elliptical] <0,0,0>: Specify radius for base of cone or [Diameter]: 4 Specify height of cone or [Apex]: 8

Command: CONE Current wire frame density: ISOLINES=4 Specify center point for base of cone or [Elliptical] <0,0,0>: Specify radius for base of cone or [Diameter]: D Specify diameter for base of cone: 8 Apex/<Height>: 8

Another way of drawing a cone is to enter in the center point, the radius (or diameter) and then establish where you want the apex (point of cone) to be. You can either type in coordinate points or pick a point with your cursor.



Command: CONE Current wire frame density: ISOLINES=4

```
Specify center point for base of cone or [Elliptical]
<0,0,0>: <PICK>
Specify radius for base of cone or [Diameter]: 5
Specify height of cone or [Apex]: A
Specify apex point: @5,5,6
```

#### WEDGE



One of the trickier primitives to draw is the wedge. You have to be careful which coordinates you input to make the wedge lie in the position you want. If it doesn't turn out just as you hoped for, you can always rotate it.

Here is an example of drawing a wedge:

Command: WEDGE Specify first corner of wedge or [CEnter]<0,0,0>: <PICK> Specify corner or [Cube/Length]: @5,2,4

Once the first corner is established, you can either enter points, or pick a spot. AutoCAD will draw the shape like it would draw a cube except it is slice in half along the length starting at the point above the first corner. There are other options to drawing wedges and see the box examples for these.





A torus is a donut-like shape or something resembling an inner tube. When drawing one you have to enter the center point, a radius to the center of the tube and the radius of the tube itself.

```
Command: TORUS
Current wire frame density: ISOLINES=4
Specify center of torus <0,0,0>: <PICK>
Specify radius of torus or [Diameter]: 3
Specify radius of tube or [Diameter]:.25
```

The diagram above shows the resulting diameters from the input above. A center mark indicates the picked center of the torus.

#### PYRAMID



To draw a pyramid, you need to know the diameter of the base and the height. The diameter can either be inscribed (inside the circle) or circumscribed (outside the circle). You can define the number of edges to from 3 to 32.

```
Command: PYR

PYRAMID

5 sides Inscribed

Specify center point of base or [Edge/Sides]:

Specify base radius or [Circumscribed] <4.5655>: C

Specify base radius or [Inscribed] <4.5655>: I

Specify base radius or [Circumscribed] <4.5655>: 4

Specify height or [2Point/Axis endpoint/Top radius]

<10.0108>: 9
```

In the above example, I showed how to switch between Inscribed or Circumscribed as the options change depending upon the method selected.

Try creating some pyramids using a variety of methods.

#### POLYSOLID



This is a new command since AutoCAD 2007. A polysolid allows you to draw a solid object while defining the height and the width. I think this command is aimed at the architects, who will enjoy the ability to quickly draw solid walls.

```
Command: PSOLID
POLYSOLID Specify start point or [Object/Height/Width/Justify]
<Object>: H
Specify height <96.0000>: 96
Specify start point or [Object/Height/Width/Justify]
<Object>: W
Specify width <6.0000>: 6
Specify start point or [Object/Height/Width/Justify]
<Object>: J
Enter justification [Left/Center/Right] <Center>: L
Specify start point or [Object/Height/Width/Justify]
<Object>: <Pick 1st point>
```

#### REVIEW

As mentioned in previous lessons, solids are usually the way to go with 3D CAD. Depending upon your chosen field, you may use 3D meshes in Civil Drafting, Isometric in HVAC, solids in mechanical, etc.

I recommend getting used to solids if you are doing any kind of mechanical drafting or architectural drafting if you are using base AutoCAD software. When using solids, you will usually take a shape and extrude it - then use **boolean commands** and others to edit it. Here is a basic building that was quickly drawn using a variety of primitive solids.



**Extra Practice:** Draw some primitive solids using the dimensions shown above or make up your own. Try to create a building out primitive solids.

# Video: Creating Primitive Solids



### Take the Lesson 3-10 Quiz

Does an object created with the Box command have 6 identical sides?

Only if you draw it that way

Never

Always



# For Further Thought:

Here's a set of tools that come in a separate case that fits perfectly in your 3D toolbox. When do you choose to use them? When you draw a box instead of extruding a rectangle? When do you use a wedge? I can't tell you that. You need to learn the tools, remember how to use them and then pull them out at the right time.

Many designers will use these tools at the early design stage to visualize space and work with concepts of how these shape might work in the environment. Think of how your could quickly assemble some 3D blocks to visualize the shape of an apartment block and then place it in a photo. This tutorials isn't about how to draw a cube, it's about how to visualize.

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# <u>CONTINUE TO THE NEXT LESSON: 3-</u> <u>11</u>



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### Topics covered in this Lesson:

#### Union | Subtract | Intersect | Extrude Face | Slice | 3D Align

First, to explain the funny name : "It (Boolean operations) was named after George Boole, who first defined an algebraic system of logic in the mid 19th century."

Working in 3D usually involves the use of solid objects. At times you may need to combine multiple parts into one, or remove sections from a solid. AutoCAD has

some commands that make this easy for you. These are the boolean operations as well as some other helpful commands for solids editing.:

COMMAND	INPUT	ICON	DESCRIPTION
UNION (Boolean)	UNION / UNI	0	Joins two or more solids into creating one based on the total geometry of all.
SUBTRACT (Boolean)	SUBTRACT / SU	0	Subtracts one or more solids from another creating a solid based on the remaining geometry.
INTERSECT (Boolean)	INTERSECT / IN	0	Creates a single solid from one more solids based on the intersected geometry.
EXTRUDE FACE	SOLIDEDIT	Extrude Faces ▼	Allows you to increase the size of a solid by extruding out one of its faces.
SLICE	SLICE	2	Slices a solid along a cutting plane.
3D ALIGN	3DALIGN	05	Aligns 2 3D Objects in 3D space.

The boolean commands work only on solids or regions. They are easy to work with *IF* you follow the command line prompts. Here is an example of each.

Start these exercises by drawing a box **5W X 7L X 3H** and a cylinder 3 units in diameter so that the center of the circle is on the midpoint of the block.

## UNION

Shown below (left) there is a box and a cylinder. These are two separate objects. If you want to combine them into one object (right), you have to use the union command.



The UNION command combines one or more solid objects into one object.

Here are the command line prompts and the resulting object:

Command: UNION <ENTER> Select objects: <SELECT THE BLOCK> 1 found Select objects: <SELECT THE CYLINDER> 1 found Select objects: <ENTER>

NOTE: The object that you select **first** will determine the properties of the unioned object when it is created.

## SUBTRACT

The subtract command is used to cut away, or remove the volume of one object from another. It is important to check the command line when using this command. Remember that AutoCAD always asks for the object that you are subtracting **FROM** first, then it asks for the objects to subtract. Here is an example:



The SUBTRACT command removes the volume of one or more solid objects from an object.

```
Command: SUBTRACT
Select solids and regions to subtract from...
Select objects: <SELECT THE BLOCK> 1 found <ENTER>
Select objects: Select solids and regions to subtract...
Select objects: <SELECT THE CYLINDER> 1 found <ENTER>
Select objects: <ENTER>
```

You can also Subtract any number of solids from a number of solids. If you do this, the solids that you subtracted from will become ONE object - even if they are not touching. Be careful with this (although you can always slice the object if you need to).

#### INTERSECT

This command creates a new solid from the intersecting volume of two or more solids or regions. AutoCAD will find where the two objects have an volume of interference and retain that area and discard the rest. Here is an example of this command shown below:



The INTERSECT command combines the volume of one or more solid objects at the areas of interference to create one solid object.

```
Command: INTERSECT
Select objects: <SELECT THE BLOCK> 1 found
Select objects: <SELECT THE CYLINDER> 1 found
Select objects: <ENTER>
```

You can intersect solids that are not touching - they will become one object.

Try these 3 boolean commands with various 3D solid objects to get familiar with them. Draw the block in <u>Lesson 3-2</u>. Draw the outline of the block, extrude it - then draw the circle and extrude it. Then subtract the circle from the block.

These commands will allow you do a lot of 3D work, using only the extrude and these boolean commands. Of course, there are some other ways to edit 3D solids.

# SLICE

This command does exactly what the name implies. You can slice a 3D solid just like you were using a knife.

Start with the basic block and cylinder shape you used in the examples above.



The INTERSECT command combines the volume of one or more solid objects at the areas of interference to create one solid object.

```
Command: SLICE
Select objects: 1 found
Select objects: Specify first point on slicing plane by
[Object/Zaxis/View/XY/YZ/ZX/3points] <3points>: <PICK POINT 1>
Specify second point on plane: <PICK POINT 2>
Specify third point on plane: <PICK POINT 3>
Specify a point on desired side of the plane or [keep Both
sides]: <Pick on the side towards the cylinder>
```

This is a very useful command - think of it as a trim in 3D. Make sure you have your Osnaps on for this command and that you pick the correct points. In a complex 3D drawing, this can be tough to see. Undo this slice and try picking 3 other points. See if the results match what you thought they would do.

# EXTRUDE FACE

Just as there is a "trim"-like command in 3D - there is also a "stretch". This is a new command in recent versions.

I usually start this command by clicking on the Ribbon item Home > Solids Editing > Extrude Faces.

The command is quite easy to use, but you need to be careful on which face you select.

Try to extend one edge of the block by 1 inch. Start the command and pick the face on the side (on the bottom line). You'll notice that the bottom face highlights as well. Next type  $\mathbf{R}$  and pick the bottom face to remove it. Then follow the command line to finish the command.

Command: solidedit Solids editing automatic checking: SOLIDCHECK=1 Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: F Enter a face editing option [Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/eXit] <eXit>: E Select faces or [Undo/Remove]: <PICK BOTTOM LINE OF SIDE FACE> 2 faces found. Select faces or [Undo/Remove/ALL]: R Remove faces or [Undo/Add/ALL]: <PICK AN EDGE ON THE BOTTOM FACE> 2 faces found, 1 removed. Remove faces or [Undo/Add/ALL]: <ENTER> Specify height of extrusion or [Path]: 1 <ENTER> Specify angle of taper for extrusion <0>: <ENTER> Solid validation started. Solid validation completed. Enter a face editing option [Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/eXit] <eXit>: <ENTER>

You should end up with this:



Another way of editing faces in AutoCAD 2007 and newer is to use grips to extrude the faces, just like you would on a 2D object. Here is an image below that shows some of the grips available. This option is only available on the basic shapes shown in lesson <u>3-10</u>.



# **3D ALIGN**

Sometimes, you may find it faster or easier to draw something separately and then move and align it into place. The command to use this in 3D is (funnily enough) **3DALIGN**.

This is a simple example, but will show you the method.

Draw a box that is  $5 \times 5 \times 6$  tall. Next draw a cylinder that is 3 in diameter and 1 tall. It should look like this:



The goal will be to align the cylinder on the front face of the box where the dotted line is.

Turn on your quadrant Osnaps. Start the **3DALIGN** command. You will first be asked to select the objects - select the cylinder and press enter.

Now you will be asked to select the 3 points as indicated below: the center and 2 quadrants. Now the cylinder will be "stuck" to your cursor as AutoCAD asks where it needs to go.

Line the cylinder up with the box by using object tracking to locate the center of the face on the box first. Then pick on the midpoints to line up the cylinder to the box. After you pick the 3rd point, the cylinder should move into place and end the command.

Here's a view of the points that were picked in case you had trouble.



The finished 'alignment job' should look like this after using the hide command.



#### Review:

After learning how to draw some basic 3D solids, you can see that using equally basic editing commands you can have a lot of options. Before advancing, review these commands by drawing simple 3D shapes and editing them.

With the commands explained on these pages, you will be able draw most of the shapes you will need in 3D. There are other options, but get very familiar with these 3D editing options before moving on. The drawings done in the sample drawing section were done almost exclusively with the commands taught in Lessons 3-7 to 3-11. Your approach will make the project either easy or difficult. Think of the various ways to draw an object before starting. You could save days with some forethought.

**Extra Practice:** Draw **this object** as a 3D solid using primitive solids and boolean operations.

**Extra Practice:** Draw **this object** as a 3D solid using primitive solids and boolean operations.

## Video: Boolean Operations and 3D Editing



## Take the Lesson 3-11 Quiz

After using the Union command, the resulting object will have the properties of...

The last selected object

It will be on Layer 0

The first selected object



## For Further Thought:

By now you should start being able to plan some simple 3D projects. You should be able to start visualizing your models and 'seeing' in 3D. Using AutoCAD to make 3D models is fairly easy but at the same time a little limited. I find that if you are using it for mechanical parts, architecture and similar fields, then it will work well. If you want to draw an organic shape like a dinosaur or exotic car, then you have some challenges ahead. There are some highly advanced 3D modeling programs that work great for organic models (3DS MAX, Maya, etc) and depending upon your goals, you may need to learn more than one program.

Even within the AutoCAD family, there are specific 3D program for architecture (Revit), civil drafting (AutoCAD Civil 3D) and others.

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### Topics covered in this Lesson:

Setting and Using the UCS | Dynamic UCS

When working in 3D, it sometimes necessary to change the plane that you are drawing on. For example, if you need to add some detail to the side of a wall, you

would want to draw on that plane. It's like taking a sheet of paper up off the floor (WCS) and taping it onto the wall (UCS).

The WCS is the World Co-ordinate System. This is the way that the default X,Y and Z axes are directed when you begin a new drawing (X to the right, Y pointing up and Z pointing towards you). The UCS is the User Co-ordinate System. This is a 'redirection' of the WCS based on parameters set by the AutoCAD user.

There are several ways of doing this, and we'll look at an example here.



This is a simple shape drawn on the **WCS** with one corner located at **0,0,0**.

Here is the same object shown a new UCS based on the side of the object, so that you are able to draw on the side of the building - maybe to put a door in? Note the direction that the X and Y axes are now pointing.



Notice that the positive X Axis is now pointing along side of the structure.

Here's how it was changed:

First start the **UCS** command. You first choose the 3 point option by typing 3 at the prompt. Next you have to pick three points to define the plane. The first point is the new origin. The second point is where you want the positive X-axis to be positioned. The last point is for the positive Y-Axis.

```
Command: UCS
Current ucs name: *NO NAME*
Enter an option
[New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]
<World>: 3
Specify new origin point <0,0,0>: <P1>
Specify point on positive portion of X-axis
<1.0000,7.0000,0.0000>: <P2>
Specify point on positive-Y portion of the UCS XY plane
<1.0000,7.0000,0.0000>: <P3>
```



This is the '3-Point' option of the UCS Command. It is one of the most useful, because you control exactly where the new drawing plane will be. You must also be extremely careful when picking the 3 points, or your plane can be shifted and cause some major problems. I would recommend using this method for most of your UCS work - or at least get very comfortable with it before moving on to other methods. Make sure you are using Osnaps to select your points accurately.

NOTE: Pressing enter immediately after entering the UCS command accepts the default of returning to the WCS.

Here are examples of the other options of the UCS command.

**ORIGIN:** This option moves the UCS based on a newly picked origin point. It does not shift the drawing plane at all as you only pick one point.

ORIGIN:

Command: UCS Origin/ZAxis/3point/OBject/View/X/Y/Z/ Prev/Restore/Save/Del/?/<World>: O Origin point <0,0,0>: <PICK A POINT>
**Z-AXIS:** This choice allows you to pick two points to define the direction of the Z-axis. First you pick the new origin, then you pick a point for the positive Z-Axis.

Command: UCS Origin/ZAxis/3point/OBject/View/X/Y/Z/ Prev/Restore/Save/Del/?/<World>: ZA Origin point <0,0,0>: <PICK A POINT> Point on positive portion of Z-axis <-8.0000,0.0000,1.0000>: <PICK A POINT>

**OBJECT:** Using this method, you have to pick on a 2D object that is lying on a particular drawing plane. This gets tricky, as you have to be aware how your positive X and Y axes end up. You also need to know how the object was drawn originally as this defines its plane.

Command: UCS Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/ Save/Del/?/<World>: OB Select object to align UCS: <SELECT AN OBJECT>

**VIEW:** By choosing the view option, AutoCAD will automatically reset the UCS to be aligned with your current view, keeping the origin where it was previously located.

Command: UCS Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/ Save/Del/?/<World>: V

**X / Y / Z:** By selecting either the X, Y or Z options, you need to first pick a point along the axis you have chosen, then provide a rotation angle based on the right hand rule mentioned earlier. This example shows how you would rotate the UCS -90 around the positive X axis.

```
Command: UCS
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/
Save/Del/?/<World>: X
Rotation angle about X axis <0>: -90
```

**PREVIOUS:** This option returns you to the last setting you had for the UCS. You'll use this one a lot in 3D. You might need to change your UCS to draw one object, then go back to what you had it set at before.

```
Command: UCS
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/
Save/Del/?/<World>: P
```

**Restore/Save/Del/?/**: The next options are used in conjunction with each other. You have the option of saving a particular UCS with a name. You can then restore that named UCS or delete if you no longer will be using it. Here are examples of these options:

Command: UCS Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/ Save/Del/?/<World>: S (Save a UCS Setting) ?/Desired UCS name: VIEW1 Command: UCS Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/ Save/Del/?/<World>: R (Restore a UCS setting) ?/Name of UCS to restore: VIEW1 Command: UCS Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/ Save/Del/?/<World>: ? (List your UCS settings) UCS name(s) to list <\*>: Current UCS: VIEW1 Saved coordinate systems: VIEW1 Origin = <0.0000,0.0000,0.0000>, X Axis = <1.0000,0.0000,0.0000> Y Axis = <0.0000,1.0000,0.0000>, Z Axis = <0.0000,0.0000,1.0000> Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/ Save/Del/?/<World>: D UCS name(s) to delete <none>: VIEW1 Deleted 1 UCS name.

This was a series where a view named **VIEW1** was saved, restored, listed, and then deleted.

#### DYNAMIC UCS

Another option to try is using Dynamic UCS, but I don't recommend this technique for new users. Here it is anyway. Down on the status bar, you have an icon that turns this setting on or off. You can also toggle this on and off with the **F6** key.



With Dynamic UCS turned on, you can easily draw on any surface you want. If you have a drawing like the basic building shown at the top of the page, you can draw a rectangle or box on the roof without manually setting a new UCS. The key is to look and see which face is highlighted as you move your cursor on the screen.

In the images below, you can see how this works. The left side shows the 'roof' face is highlighted and the cursor has changed to reflect the new UCS. The right shows how this looks when the right side face is active. Not the change in the colored cursor while the UCS Icon remains the same.



#### Other notes about the UCS:

Be careful when choosing a UCS. Look to the UCS icon and see that it is aligned the way you want it to be. Look for a clean vertical lines if it should be aligned along a vertical plane. Most of the time I will use the 3 Point UCS option and then Previous to return and then back to a different 3 point option and so on. 3 Point gives me a lot of versatility without having to think about it too much (and concentrate on how I'm going to draw the project.

Always be aware of where your UCS is located. Make sure that positive X is where you expect it to be.

**Extra Practice:** Create this **drawing** by switching the UCS to the correct plane to draw the roof and skylights. For the geometry, use a combination of extruding and boolean operations to build it.

**Extra Practice:** Create this **drawing** by setting the UCS to achieve what you need. Tip: Extrude a rectangle, set the UCS and extrude another rectangle.

# Video: Setting the UCS / Dynamic UCS

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### Take the Lesson 3-12 Quiz

Which key toggles Dynamic UCS?



## For Further Thought:

Remember when you were learning 2D AutoCAD and found that the zoom and pan tools were useful? Now that you are dealing with a 3rd dimension, you can see how changing your USC is very useful. The more complex your model gets, the more you

will use the UCS.

I've seen people fight with drawing something and trying to move it into the correct position rather than just change the UCS and draw it in place to begin with. At the risk of repeating myself - you need to learn 'when' to use these tools as much as 'how' to use them.

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# Topics covered in this Lesson:

### Mapping materials on Objects

Whenever you are attaching materials to an object in AutoCAD, you are effectively 'stretching' the image of the material around the object. For many objects, this can be acceptable. For a lot of others, though, you may want to adjust how the material is displayed on the object. This process is called mapping. The commands in this lesson are similar to the **SETUV** command used in previous versions of AutoCAD.

Here is an example of the same object with the same material, but with the mapping of the material adjusted on the right-side object.



By default, AutoCAD will apply the material as it sees fit. In this example above, the scale of the material is too small. To properly render the block, mapping is used to adjust and fine tune the material so that it looks the way you want it to appear. With a little knowledge of this command, you can make your renderings more realistic.

COMMAND OR INPUT	ICON	LOCATION	DESCRIPTION
MATERIALMAP / SETUV / MAP	● Material Mapping ▼	Home > Visualize > Material Mapping	Enter this on the command line to select mapping options via keyboard or icons.
Planar Mapping	Planar	Home > Visualize > Material Mapping > Planar	Maps individual faces of an object.
Box Mapping	Вох	Home > Visualize > Material Mapping > Box	Maps any solid object with controls for width, depth and height as well as rotation on all sides.
Spherical Mapping	Spherical	Home > Visualize > Material Mapping > Spherical	Allows you to map any solid object, but uses rotation only.
Cylindrical mapping	Cylindrical	Home > Visualize > Material Mapping > Cylindrical	Maps a solid object with height and rotation only.

Here are the commands needed for mapping your materials:

Start by drawing a box that is 120x120x240 high and do a **Zoom** > **Extents**. Set your visual style to Realistic (Visualize > Visual Styles > Realistic). Switch to the SW ISO view.

As shown in Lesson 3-9 apply the material called "Masonry > Common" to the box.



Depending upon your settings it may look something like this in a default view.



In this view, you can see the bricks, but we want to make them bigger - this is done via mapping.

We'll start with the **BOX MAPPING** icon and then select the box and press enter.

You should now see some new grips on your object (this are different from the grips that allow you to adjust the size of an object.



The above image shows the added options when the rotate option <**R**> is selected on the command line.

With the grips active, you will need to make the map larger by selecting the mapping grips one at time to make the material bounding box (in yellow) larger. Once you have finished moving a grip, click in the drawing space to release it. Note that there is one on the top as well (for height) as well as the four at the bottom. When you are happy with the look, press enter to end the command. Your new and improved box could look something like this:



As you can see, the one on the right looks better than the default on the left.

Now start the Box Mapping option again, and type in R **<ENTER>** to invoke the Rotate option. You should now see the circular grips as shown two images above. To rotate the map, move your mouse over one of the circles and it will change color. Click and you will be able to rotate the material around the axis you chose.



From these simple options, you can control the look of any material on any object. Ultimately, the look of your final rendering is the choice of you - the designer.

The image below shows the options below for the cylindrical and spherical mapping commands:



The left image shows the options available with the spherical mapping command. Note that it is limited to Rotation and Moving.

The right image shows the options of the cylindrical mapping command. It is limited to rotation and height adjustment.

Try the two mapping methods shown above on a sphere and cylinder. Then try using the Box Mapping command on those objects. You may find that in common usage, the Box Mapping method will provide you with the most versatile options to achieve the look you want.

#### PLANAR MAPPING

The other option not discussed yet is the PLANAR MAPPING command. It works a little differently than the others in that adjusts the material on one face only instead of the whole object.

Draw a cube 120x120x120. Apply a material to it that has some contrast to it so you can see the results clearly.

Start the Planar Mapping Command using this icon:



Instead of just selecting an object, press the Control Key as you click the box and you will see that only one face is highlighted. Click on the face that you want to map. From there, you will see that you have the same resize and rotate grips available to you, but will only affect the one face you selected.

**Extra Practice:** Try mapping several different materials onto various objects of different sizes. You will need to get very familiar with this concept to produce better renderings.

# Video: Mapping Material in AutoCAD 2016

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### Take the Lesson 3-13 Quiz

Which key toggles Dynamic UCS?

F8		
CTRL+U		
F6		

# For Further Thought:

One of the hardest things in 3D drafting and rendering is getting the final model to look as great as you expect it to. The key to this is getting the best material mapped properly and lit correctly. This process is a complete skill-set unto itself. Depending upon your goals and job description, you may have to only get a rough concept across or provide an exact rendering showing the actual material. Mapping can make all the difference between a good rendering and a poor one. Do you like what you just learned? Support this site with a Donation via Paypal:



# <u>CONTINUE TO THE NEXT LESSON: 3-</u> <u>14</u>



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Last Update: 02/25/2019 18:22:00

## Topics covered in this Lesson:

#### Creating your own custom materials

Once you start working with AutoCAD's default materials, you'll soon realize that you don't really have a large selection. What if you need a white stucco material for a wall? Or grass for the lawn, or brushed aluminum, or... well, you get the picture. What you need to do is create your own materials.

The first step is to locate an image that represents the material you want in your drawing. There are a number of sources available on the internet. Below are 3 samples you can try for this lesson. Click on the images to download the samples.



Pick on any of the images above and when the new window opens, right click on the large image and select "Save image as..." and save it in a folder where you can find it easily.

Now start the **MATERIALS** command to open the Materials Palette.



Look for the button shown in the red box above - this will let you create your own custom materials.

I will take you through the simple process of creating a material from an image file and applying it to an an object in your drawing. Then you will learn how to modify the material. The previous lesson will give you more information on **mapping** the material to an object.

To create a new material from one of the images from above (or another texture) open the materials palette and click New Material button. This will give you a droplist to select the type of material you are creating. Select **New Generic Material** and we'll use the metal image you downloaded at the top of the tutorial.



Next you'll see the palette for creating new materials. Click on the **Information** tab and fill in the three fields as shown below.

Appea	rance	Inform	nation		X
Info	rmati	on			*
		Name	Rusty I	Vletal	
	Desc	ription	Custor	n Rusty Metal Material	
	Ke	ywords	materi	als,metal, rust	
Abo	ut				
		Туре	Generic		
					ĸ
					EDITO
					IALS
					AATER
					炎 MATERIALS EDITOR

Click on the Appearance tab and add your image by selecting in the box shown below. The will open up window so that you can navigate to the image's location. Load the image and you'll see it shown in the window.



For now we'll just use the default settings so that you can see how the material looks "straight out of the box". Apply your new material to an object and see how it looks. Perform a render if you want a better view.



Your object might look a little different from what I show above, but it should be close. Now that you've seen your material in action, you can adjust some setting and see how it changes the appearance. If you move the slider for transparency for example, you'll see the changes live on your screen.

× 1.1 **	Appearance Information			
	▼ Generic			
	Color	RGB 80 80 80		•
	Image		1	
		metal33t.jpg		
	Image Fade		100	
	Glossiness	Non-Metallic	50	
	Highlights	Non-Metallic		_
	▼ ✓ Reflectivity		47	
	Direct Oblique		17 15	-
			15	
	<ul> <li>Transparency</li> <li>Amount</li> </ul>		20	
	Image		30	
	inage			-
		(no image selected)		
	Image Fade		0	
	Translucency		34	•
	Refraction	Water 👻	1.33	
ITOR	Cutouts			
MATERIALS EDITOR	Self Illumination			
RIAL	Bump			-1
ATE	► Tint			
⊻ ⊗				

Try some of the different settings to see what they do. You have endless possibilities to make your materials do exactly what you want it to do.

Although the example above is very basic, it shows how you can quickly create some materials for a custom rendering. Keep this in mind when you are just starting your rendering process or when you don't have any advanced materials to work with.

You can edit the image you're using if you click on the small sample image in the palette.

Create another new material like you did above. Instead of editing the material, this time you will use on of the images you saved (from above) to create a more realistic material.

×	Appearance Informat	ion			
⊧• ₩	6		×		
			н ж		
	Rusty Metal			5.5000000	12 1
	▼ Generic			5.50	Search State
	Color	RGB 80 80 80			F F0000000
	Image		Image		5.50000000
		metal33t.jpg			metal33t.jpg
	Image Fade Glossiness		Brigh	tness	86
	Highlights	Non-Metallic			Invert Image
	▼ ✓ Reflectivity		▼ Transfor	ms	
	Direct				Link texture transforms
	Oblique		▼ Positi	<b>on</b> Offset	0.00000000 💠 X
	▼ ✓ Transparency		Ì	JIISEL	0.00000000 ÷ Y
	Amount		Rot	tation	0.00°
	Image		▼ Scale		
			Sampl	e Size	5.50000000 💠 Width 📾
	Image Fade	(no image selected			5.50000000 💠 Height
	Translucency		♥ Repeating		
	Refraction	Water	E	ontal	Tile •
TOR	Cutouts			ertical	Tile 🔻
S EDI	Self Illumination		TEXTURE		
MATERIALS EDITO	Bump				
AATE	► Tint		»		
~ ?	Q • 📃				

A couple of things to look at here are the Brightness and Scale settings. Adjust the brightness and you can darken your material easily. The one I use quite often is Scale. This allows me to quickly change how big the material appears on the objects. This can be done using MAPMATERIALS, but it's also easy to do in this palette. The scale you use will depend upon two things: the size of you image (in pixels) and the size of your object. You'll have to play with this one to get it looking the way you want

it. As a rule, larger image files are better as you can scale them down and retain clarity

This is just an introduction to the world of creating new materials. If you have a photo editing program like Photoshop or **<u>PixBuilder Studio</u>** (it's free), you will be able to adjust the image files to fit your needs.

For practice, here is good way to get graphics into your renderings. Draw an octagon (**POLYGON** command) and turn it into a **REGION**. Then create a new material using this image of a **<u>stop sign</u>** and apply it to the region. You will most likely have to adjust map settings to make it look good (or use the scale to object option shown above). For even more fun, add a wooden pole to it and use **3DROTATE** to make it stand up properly.

For more practice, try downloading some images or using ones that you already have and apply them to objects. Use the mapping tools to adjust them.

### Video: Custom Materials n AutoCAD 2016



### Take the Lesson 3-14 Quiz

Can you make any any material transparent?

No, only the default AutoCAD materials

Only the ones you create

You can make any material transparent



# For Further Thought:

AutoCAD has come a long way in their rendering capabilities and material options. Even so, they can't cover every possible material. If you work in furniture design, you'll have a complete library of custom fabrics to use. If you work with wood products like cabinet work, you will have a different library. Designing stone counter tops? There's another library.

Knowing how to create and use custom materials is one way to make your work look accurate and unique.

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# <u>CONTINUE TO THE NEXT LESSON: 3-</u> <u>15</u>



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Last Update: 02/25/2019 18:22:14

## Topics covered in this Lesson:

#### **Review and Practice of 3D Concepts**

There is only one way to get better at AutoCAD and that is to practice. In 3D, you not only need to know the commands, but also how best to use them. As you start drawing in 3D, you may start looking at everyday objects and think about how they would be drawn. Look at some 3D drawings online and figure out how they were done.

### EXTRA PROJECT - A

If you have finished the previous exercises and would like more practice, try this:

Sketch out a simple coffee table on paper. Add the basic dimensions. Draw this in AutoCAD using any 3-D method. Once you've done this, draw another lamp and place it on the table. Use rotate3d if you need to turn it right-side up. Find a good view to display what you have drawn. Add materials.

**EXTRA PROJECT - B** 

Take your file from <u>Lesson 3-7</u> that gave you a solid model and use the fillet and chamfer commands to add some smoother corners to your chair.



### EXTRA PROJECT - C

Draw the object that you draw in the **isometric** lesson in 3-D using extruded regions. You will have to first create the outline, then the circle. Turn both objects into regions. Subtract the circle from the outline to have one remaining region. Extrude this to get your solid object.

### TUTORIAL - CREATING A 3D OBJECT FROM A 2D PROFILE

For this tutorial, you should be familiar with these 3D commands: region, extrude, union, subtract.

This tutorial shows you a method for quickly turning 2D shapes into 3D objects. One of biggest problems with 2D objects is that they are not always drawn properly. This

is something that should have been addressed when it was first drawn, but here is a way of working around it.

First lets start with a basic shape (remember the **array** command). This is the profile for a gasket. It is drawn in 2D with lines, arc and circles.

Let's hope it was well drawn, meaning no overlapping lines, gaps, etc. Copy the objects over to the side for the next section.

Here is how you turn it into a 3D object easily. Start the **REGION** command. Draw a crossing window around the whole area and press enter. Once you have your regions created, you can extrude them. Start the **EXTRUDE** command and select all objects. Press enter and enter your extrusion height and taper angle. When these are extruded, begin your **SUBTRACT** command and first select the larger object first (to be subtracted from), press enter then select all the other objects and press enter. When you change to your SW Isometric view, type in **HIDE** (enter) and you should see the object as shown below.



Remember that you can always work in your SW Isometric view to see the progress of your 3D work.

There is one problem when using the **REGION** command. If have two lines that are overlapping, AutoCAD can not make a region out of the objects. You could search until find the offending line, but there is an easier way.

Here's this approach:

Now look at the copy of the shape that you made you earlier. Instead of using the region command, use the **BPOLY** command. This command works similar to the **HATCH** command in that it finds the boundaries for you. Start the command and you will see this dialog box come up:

An Boundary Creation
Pick Points
✓ Island detection
Boundary retention
✓ Retain boundaries
Object type:  Polyline  ▼
Boundary set Current viewport    New
OK Cancel Help

Accept the defaults and press on the **Pick Points** button. In your drawing screen, pick an internal point as if you were hatching it and then press enter.

Now start your **EXTRUDE** command and select the last (L) object created and extrude it. Now begin the **SUBTRACT** command and select the outside object, (press enter) then use a crossing box to select everything else and press enter. Use the **HIDE** command to ensure that everything is as it should be.

## Video: Review of exercises in Lesson 3-15

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### Take the Lesson 3-15 Quiz

Which object(s) can be extrdued?

Rectangle All of these

Region



## For Further Thought:

When I taught 3D AutoCAD in a classroom, I let each student come up with their own idea for a final project. Some had specific ideas what they wanted to draw, others needed help. But the main goal was to think of something and then figure out how to draw it. Some projects I remember were a boat, a keyboard and a guitar. Each one used some of the tools in this level. All of them had one thing in common. They all required planning on the part of the student to make the project happen. We discussed which tools might work best, which ones wouldn't work (and why) and then we came up with a plan. It was great experience for the student and I had a lot of fun thinking about designs I hadn't considered before. Look around your 3D world and start thinking about how you would draw the world around you. Do you like what you just learned? Support this site with a Donation via Paypal:



# <u>CONTINUE TO THE NEXT LESSON: 3-</u> <u>16</u>



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Last Update: 02/25/2019 18:22:32

## Topics covered in this Lesson:

### Create a 3D Model of a Building

If you have completed the previous lessons in this level, you now have the tools to create a large variety of objects in 3D. This lesson is designed to show how to create simple building in 3D from start to finish. The same process can be used to build more complex buildings as well. Start by creating this building and then try drawing your home.

This method follows my usual approach of "modeling it like you would build it". I'll start with a floor plan, then build the walls, make openings for the doors and windows, add doors and windows, then put a roof on it. It just seems like the most logical approach to me.

Below is an image of the final building that is drawn in this lesson (with rendering and lighting explained in the next lesson).



Start by setting your units to architectural (DDUNITS). I apologize to everyone in advance that uses the metric system, but most of the viewers and revenue come from the USA, which still refuses to learn the metric system. These techniques will work with any floor plan, so if you have one with metric measurements, feel free to use it.

Remember to create **new layers** for each type of objects you will draw (windows, doors, walls, roof, etc).

Now draw the bases of the walls using the dimensions are shown below. You don't need to dimension it. Optionally, you can use the (new in AutoCAD 2007) **POLYSOLID** command.



Now you will create regions out of all your wall lines.

Command: REG REGION
Select objects: <SELECT ALL THE LINES> Specify opposite
corner: 7 found
Select objects: <ENTER>
3 loops extracted.
3 Regions created.

You should now have 3 regions. If you don't then it usually means the your lines don't meet.

Next you are going to **EXTRUDE** the walls 9' high.

Command: EXT EXTRUDE Current wire frame density: ISOLINES=4 Select objects: <SELECT THE 3 REGIONS> Specify opposite corner: 3 found Select objects: <ENTER> Specify height of extrusion or [Path]: 9' Specify angle of taper for extrusion <0>: <ENTER>

It should appear that nothing has happened. You will need to go to the SW Isometric view to see how the wall were extruded. Do this and then use the **HIDE**Command. Something doesn't look right. What you will need to do is **Subtract** the 2 smaller regions from the larger region. You should see this result after the Subtract and Hide commands.



Now it's starting to look like something. Your walls are almost complete.

Next you will add a door or two. The first door, on the outside will be on the bottom left wall. Start by drawing a rectangle from the middle of the outside wall and the other corner @6,36 - this will create the opening. Your rectangle should look like this:



Now you want to **EXTRUDE** the rectangle up 6'8" and **SUBTRACT** it from the wall.

Create a door layer (or make it active if you already have one) and draw a rectangle in the door opening that is 3"x 3' - put it in the middle of the door opening. Extrude it 6'8".

Now create a second door (and opening) on the inside wall parallel to the outside wall that is 32" wide. After you do that, the drawing should look like this from the SW Isometric view:


If you haven't saved your drawing yet, now would be a good time.

Now you're going to add some windows. This will be done using a similar process as the door. You will create openings, then add the window.

Create 2 rectangles in the position shown below.



**EXTRUDE** the window 36 inches. Them move them up 3'8" in the Z axis. Remember your relative coordinates from Lesson 1-1?

```
Command: M
MOVE
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: Specify base point or displacement: <PICK
ANYWHERE ON SCREEN>
Specify second point of
displacement or <use first point as displacement>: @0,0,3'8
```

Now mirror the bottom left box to the opposite wall (Note: It will be easier if you revert to the top view for the mirror command) and the upper box to the inside wall so you have the 4 boxes looking like this :



Next, subtract the four window openings from the walls. After switching to SW Isometric view and using the hide command, your drawing should look like this:



Now the window panes have to go in. Since this is a simple drawing, you just be putting in a plane of glass to represent the window.

Create a new layer for windows. The easiest way to do this is just like you created the doors. Make the window 1" deep. Once again, start with the rectangle at the base of the window opening, extrude it the height of the window opening and move it into the centre.

Once you have one window made, copy it to the other openings. Save your drawing and then type in **SHADE** and choose the **G** option. Your building should look like this:



Wow - now you're getting there. Next you will add to the walls to make them the right pitch for the roof.

First you will have to change your <u>UCS</u>. Start he **UCS** command and choose the 3Point option by typing 3 <ENTER>. When prompted (watch the command line) select the points shown below:



After you pick the third point, the cursor should turn on its side. Now you can draw on the side of the building. In this case, you will be drawing a triangular shape to represent the rest of the wall up to the roof.

Draw a **LINE** from the middle of the top outside of the wall up 6'. Then draw a **POLYLINE** from one corner, to the next, to the top of the line and use the Coption to close the polyline.

Finally **EXTRUDE** the triangle -6" and then **COPY** the extruded object to the other side of the building. Erase the vertical line you drew. You should have something like this now:



To finish the wall, perform a UNION (<u>Lesson 3-11</u>) between the triangular sections and the bottom wall.

You're almost done now. You will need to draw the roof to complete the drawing.

First you will need to change your **UCS** to align it with the slant of the roof. Use the points shown below - and be careful which endpoints you select.



Now draw a **RECTANGLE** from the top left corner of the triangular shape to the bottom of the opposite corner of the other triangular shape. It should like the magenta rectangle below:



New **EXTRUDE** the rectangle 6". Then copy 12" down in the Y Axis and then copy it 12" up in the Y Axis - when you have the 3 sections **UNION** them into one object.

Change back to the WCS (UCS <ENTER> ) and mirror the roof object to the other side. Check that it looks correct by viewing the model from the left and from the front.



The last thing to do is trim the excess pieces off the top of the roof where the two slabs intersect. This will be done using the **SLICE** command (<u>Lesson 3-11</u>).

Go to the SW Isometric view.

Start the **SLICE** command and select the roof section towards the back. Then when you are asked to pick three points on the slicing plane, select any three points on the top of the roof section at the front.

Switch to the NW Isometric view and slice other roof section in the same manner. Check to see that your roof looks correct, and if it is, union the two roof objects now.

Go to the WCS and copy the roof 12" in the Positive X Axis and 12 " in the Negative X Axis and then union all 3 roof objects. Your building should now look like the one below:



Your building is complete. To add a lawn , draw a very large rectangle around the house at the same Z level as the bottom of the house. Then turn it into a region so that materials can be added later.

Save the drawing, then if you like, add materials and render the final image. In the image at the top of the lesson, the door is detailed material. More detailed instructions for rendering will be found in <u>Lesson 3-17</u>.

Here are the materials used for the rendering at the top of the page. You can download them and add them to your Textures folder. The door texture would be used like the stop sign you may have rendered in **lesson 3-14**.

#### door.jpg | grey\_roof\_slate.jpg | grass.jpg | stucco\_white.jpg

You can also try to add more detail to the building if you like. But now you can see that using a few commands (in this case about 12), you can draw decent looking models.

**Extra Practice:** Build a 3D model of the Cabin Drawing from <u>Lesson 2-1</u>. How much detail can you add? Door knobs? Window panes?

Video: Model a Building in 3D

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### Take the Lesson 3-16 Quiz

What should you model first in a building?

The walls The doors The windows

### For Further Thought:

Even if you have no plans to create architectural drawings I hope you complete this project. It is a simple building, but the concepts shown can be used in a number of different fields. It also shows how a 3D model in AutoCAD is created from nothing. If you're a mechanical drafter, you can use these skills to create assembly drawings. You can continue to add as much detail as you want to this model. You can add furniture, cabinets or anything that a house would have.

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# <u>CONTINUE TO THE NEXT LESSON: 3-</u> <u>17</u>



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Last Update: 02/25/2019 18:22:44

Topics covered in this Lesson:

#### An introduction to Making an Animation / Walkthrough in AutoCAD

One of the great things about AutoCAD is that you have a number of ways to display your design to other people. If you draw in 3D, you can show it from any angle, in any material and any lighting. It's amazing what you can do. This tutorial will show one more option - animation.

Animation - or exporting a video - is a great way to communicate your idea to someone less technically minded. It could make the difference between selling a product, or just getting a blank stare from a client. Almost everyone can look at a video and imagine what the final product might look like.

Here is a very simple video I made to show what you can do with AutoCAD's animation tools:

#### Video: Sample Animation in AutoCAD

First we'll start with a simple animation of a small model. This is something you might use for a presentation of a mechanical part design.

<u>Here is the file</u> that you will need, for now we won't worry about a realistic rendering, I'll just show the steps I used to create the video shown above. Download and open this file and you should see something like this:



What you see is the main object that we want to show (the block with holes drilled in it) and two circles. The outer circle will be the path that the camera follows as it 'shoots the video' and the smaller circle is the path the camera will look at (target) as it creates the video.

You'll need to see the 'Visualize' ribbon for this lesson, so make sure your workspace is set to "3D Modeling" otherwise you won't see it. Then look for the Animation Panel. If you don't see the Animation Panel you will have to turn it on. Right click on the palette and click on "Show Panels" and select "Animations" so that there will be a check mark beside it.



Now you can select the icon to start the setup of the animation.

Of course, the easiest thing to do is just type in **ANIPATH** to start the command - remember how in Level 1 I said that typing commands is faster?

No matter how you start the command, you will see a new dialog box appear:

A Motion Path Animation				
Camera Link camera to:	Animation settings <u>F</u> rame rate (FPS): 30			
⊘ <u>P</u> oint	Number of frames: 300			
Path1 🔻	Duration (seconds):			
Target Link target to:	Visual style: Realistic			
⊘ P <u>o</u> int ⊚ Pa <u>t</u> h 🔶	Format:         Resolution:           MOV         ▼           800 x 600         ▼			
Path2 🔻	Corner deceleration			
When previewing show camera preview				
Previe <u>w</u>	OK Cancel <u>H</u> elp			

Like any new dialog box, start at the top and work your way down. First link the **camera** to a path (the outer circle) by picking on the crosshairs icon at the top. Pick the outer circle and then accept the default name in "Path Name" dialog that appears after you select it.

Now link the **target** to the inner circle using the same method.

You're ready to do a quick preview, so click on the "**Preview...**" button. This will bring up a simple video player to show you the preview:



Now that you know what the animation will look like, you can set the video settings on the right side of the dialog box. Make the adjustments you need as shown below. This will create a 10 second video at 800x600 with a 'realistic' rendering style as shown in the sample video at the top of the page.

Motion Path Animation				
Camera Link camera to:	Animation settings <u>F</u> rame rate (FPS):	30		
© <u>P</u> oint ⊚ <u>Pa</u> th +	Number of frames:	300		
Path1 👻	Duration (seconds):	10.00		
Target Link target to:	<u>V</u> isual style: Realistic	<b></b>		
⊘ P <u>o</u> int		olution: 0 x 600		
Path2 🔻	Corn <u>e</u> r deceleration	Reverse		
When previewing show camera preview				
Preview OK Cancel Help				

When you have it all ready to go, press the **OK** button and save the video in a place you can easily find it. After it renders, open the video in whatever video player you have on your computer and check it out! If your computer can not play the chosen output format (AVI in this example) try a different format.

If you're paying attention, you noticed a rectangle near the inner circle in the sample drawing. I added this in so that you could see how selecting a different target path would affect your video. Create another video using the same process as above, but link the target to the path of the rectangle. Render the video and see the difference.

Now that you know how the process works, try some variations of your own. See what the different "Visual Styles" look like and think about how one style might be preferred over the other depending upon the design you are showing.

Create a spline to use as your camera path. This can be and open or closed spline. Think about what angle might be best for showing this simple block, then try to imagine what you might do for a house (which we'll work on next).

This next animation will use the same process, but with a more complex model. If you draw the small cabin in the last tutorial and added materials, then you're ready. If you didn't (and you should have drawn it!!) then you can **download the basic model here** without materials.

## Video: Sample Rendered Animation in AutoCAD

If you have the materials added to your drawing, you can choose the "Render" option for Visual Style - but be prepared that you will keep AutoCAD busy for up to an hour (or longer depending upon your computer). If you don't want AutoCAD tied that long, try a different Visual Style.

The purpose of using this building model is to show you the computing power needed to create animations. The simple block object we used at the beginning of lesson was selected for a reason. It allowed you to test different settings without having to wait very long for results.

#### Creating a Walkthrough Video

For some of you a walkthrough video might come very naturally. If you've ever played a 'first person' type video game, then this is almost the same thing. Here's a sample video of a recorded walkthrough in AutoCAD.

#### Video: Sample Rendered Walkthrough in AutoCAD

As you can see, my video game skills are a lacking, but you get the idea. All of the camera movement in that video was controlled by my mouse. You can also use the 'arrow' keys to control the camera.

Here's how to set up a simple walkthrough. If you don't already have the 'cabin' file open, get it open again and open the Animation panel if needed. Look for the 'feet' icon that will start up a new palette for you.



When you click this icon, you'll see the "Position Locator" palette appear. What this shows is the overall model space you're working with. At the top, you'll see zoom icons - try these out first.



If you click in the window at the top, you can use the scroll wheel of your mouse to zoom in and out as well. You will also see a green triangle - this represents your camera with the camera at the red dot in the triangle, and the 'cone' representing the view you will see. You can practice dragging this around and see the result in your main drawing screen.

### Video: Creating a Walkthrough in AutoCAD

## Learn to Fly

There is one other option you have for animating your model, and that is the 'Fly' mode. The process is the same as a walkthrough, but the difference is that you can change your elevation. In walkthroughs your elevation, or Z axis location remains the same. You can look up or down, but you stay at the same 'height' - just like walking.

When you are in Fly mode, you can point the camera up or down and then move towards that point and change your Z axis location. Use the Fly icon and setting just like you would a walkthrough, but see how your elevation can change.



#### Take the Lesson 3-18 Quiz

What do you set for a motion path animation?

Target and Path

Camera and Target

Camera and Path



#### For Further Thought:

In many of my tutorials I've tried to explain the advantage of using AutoCAD as more than just another way to draw things. What you have learned just now is a valuable tool that will let you communicate with clients who may have trouble visualizing what you have designed or drawn. A simple 10 second animation video can save hours of tedious explanation. The bottom line here is to learn what tools you have available and when to use them to best get your point across. As you have seen, video is not a difficult concept to learn, but it can be a very powerful tool in your toolbox. Learn to master it if you are drawing in 3D.

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### Topics covered in this Lesson:

#### An introduction to Lighting and Rendering in AutoCAD

If you have successfully worked your way through this level, you should be able to create some decent looking models in AutoCAD. The next step is learning to render them. This lesson will include a look at the new lighting and rendering options first introduced in AutoCAD 2007.

Looking at the image below (model from <u>Lesson 3-16</u>) you'll notice that it has a background of clouds, shadows and looks real (if you squint). You should already know how to create the model, view in a perspective view, add materials, map them and look at them in the viewport. The next step is to create lighting, add a background and render the scene out to a file.



This lesson will use the house model as an example. If you didn't draw it (yes, you should have) here is file that you can download as well as the textures used.

FILE	FILENAME	APPLY TO
Building drawing file	building.dwg	-
Roof material	grey_roof_slate.jpg	Roof
Grass material	grass.jpg	Lawn Region
Wall material	stucco_white.jpg	Building Walls
Door material	<u>door.jpg</u>	Front Door
Sky background	<u>sky.jpg</u>	Background

Create the materials as shown in <u>Lesson 3-14</u>. Add a glass material to the windows from AutoCAD's material library, or create a new generic glass material. Use or

create a concrete material for the sidewalk if you drew one. Alternately, you can use the materials from the default material library.

Apply the materials (<u>Lesson 3-9</u>) to the appropriate objects and map them as shown in <u>Lesson 3-13</u>.

Here's the drawing 'as-is' with the Visual Style set to Realistic - with materials added, but not mapped.



Note that it looks very rough at this point

Now with the materials mapped, it's starting to look more realistic:



Looking better, but not there yet.

## Background

Next step is to add a background to the scene. To do this, you'll have to create a new view. Set up a perspective view similar to what's shown above (see <u>Lesson 3-</u><u>4</u>) and name it using the **VIEW** command. The comes with a ready made perspective view (PERS). Click on this. Under the General settings, you will see one for Background. Select this and choose Image. This overrides other background settings used in other views.

🔺 View Manager	22		. Die		×	
Current View: Current		General		_		
Model Views	<u> </u>	Name			Set <u>C</u> urrent	
ISO			pers	- 1	<u>N</u> ew	
r pers		Category	<none></none>	- 1	<u>IN</u> EW	
Layout views		UCS	Front	- 1	Update Layers	
🖃 🗍 Preset Views		Layer snapshot				
Тор	E		Annotation sca	1:1		Edit Boundaries
		Visual Style	3dWireframe			
- 🗾 Left		Background ov	Image	-	Delete	
Right		Live Section	<none></none>			
Front Back		Animation		-		
SW Isometric		View type	Still			
SE Isometric		Transition type	Cut to shot			
NE Isometric	-	Playback durat	1/2"			
		]				
		ОК	Cancel	Ar	pply <u>H</u> elp	

In the next dialog box, click on Browse and find the sky.jpg file and select it in the next dialog, then select the "Adjust Image" button.

🔺 Backg	round
Type:	[Image -
Preview	~
Image	options
E:\w	e-R-here-FULL-BACKUP\cad-16\t Browse
Adju	ust Image
	OK Cancel Help

This will give you yet another dialog that allows to scale the image for the background. This is similar to setting your Windows desktop background. Choose **Stretch** in this example.

🔺 Adjust Background	d Image	
Background preview	Stretch 💌	
Sliders adjust:		
Offset	© <u>S</u> cale	
	<u>R</u> eset	
<u>M</u> ainta	in aspect ratio when scaling	
	OK Cancel <u>H</u> elp	

Press OK and OK and Apply and OK to get back to the drawing screen. If all went well, then it should look something like this:



Ok, not the best background, but you now know the concept and can try different image if you like.

So now you have the model, the materials, the mapping and the background. All that is left is some light and shadows. There are three types of lights in AutoCAD: **Point**, **Spotlight** and **Distance**. I'll show how each one is created and adjusted.

One form of a distance light is the Sun (most distance of all light).



To do this, type in **SUNPROPERTIES** or (menu: Render > Sun and Location). It will give you this palette:



This palette allows you to control the Sun. Yes, it sounds very powerful, and it is.

First you must turn on the Sun, if it isn't already. Then select the time and day sometime you like (in daylight).

To render the scene, type in **RENDER**. You should get something like this:



Now that you see how it works, you can try different settings. Earlier in the day should produce longer shadows than at midday. You can also control the sun from the ribbon (Visualize > Lights)



The button the left for Sun Status will turn the sun On or Off (in the image above, it is on as indicated by the blue shading surrounding the icon). This is a quick way to work with the sun. You can also type in **SUNSTATUS** and give the variable 1 (on) or Zero (off).

Before adding the lights you want to change the background from the sky image to night. Instead of using another image, choose the gradient option in the View Manager > Select View > Background Option. Try to get a black to dark blue gradient. Try selecting the Three color checkbox for more versatility.

View Manager Current View: Current <u>V</u> iews		X
Background Type: Gradient Preview	X3         -           ne>         it           ireframe	Set <u>Q</u> urrent <u>N</u> ew Update <u>L</u> ayers Edit <u>B</u> oundaries <u>D</u> elete
Gradient options Top color: Rotation: O Bottom color: OK Cancel	Select Color  Index Color  True Color  Red: 25  Green:  Green:  I28  Color: 25,34,128  OK	Color Books Color model: RGB True Color Stored as RGB: Red: 25 Green: 34 Blue: 128 Cancel <u>H</u> elp

If you perform a render now, you will get a nice twilight background and generic lighting. The scene is still too bright for dusk, but we'll change that soon.



## Spotlight

The next type of light to add is a spot light. We're going to add a safety light above the door for night time.

For something a little more dramatic, it's time to add a spotlight. This will be placed under the peak of the roof and point straight down.

Look for the Lights Tool Panel on the Render ribbon.



This is easy enough - just click on the button and you will have the choice of where you want your light and what direction you want to point it. In this example, pick the endpoint (osnaps) under the peak of the roof and point straight down to the ground (Ortho).


Once your light is placed, press **<ENTER>** to accept the default settings. Now select the light glyph and you will see the default spread of the beam that the light will cast. (You might want to switch to your SW ISO view first). Right click on your new light and choose properties so you can fine tune your light for rendering.

×			
ļ.	Light 🔹 🖬 🔶		
×	General	-	
	Name	Spotlight1	
	Туре	Spotlight	
	On/Off Status	On	
	Hotspot angle	45	
	Falloff angle	50	
	Intensity factor	300.0000	
	Filter color	221,240,95	
	Plot glyph	No	
	Glyph display	Auto	
	Geometry	-	
	Position X	1 1/8"	
	Position Y	15'	
	Position Z	-5'-6 3/8"	
	Target X	1 1/8"	
	Target Y	1 1/2"	
	Target Z	-5'-6 3/8"	
	Targeted	Yes	
	Attenuation	-	
S	Туре	Inverse Square	
RTI	Use limits	No	
PROPERTIES	Start limit offset		
PR(	End limit offset	10"	
티			

You change these later, but for now, set the **Intensity Factor** to 300 (higher number makes it brighter). Change the **Filter Color** to a pale yellow (this will simulate the yellow glow of a tungsten light). I like to set the **Attenuation** to "Inverse Square" as this is how lights work in the real world.

**Note:** When you have a drawing with a lot of lights, you should give each light a unique **name** to keep track of them. This is also done in the properties palette. Use the **LIGHTLIST** command to see a list off all lights in your current drawing.

Before you try a render, you'll want to turn off the Sun (if you haven't already) and then lower the Default Lighting (this is the ambient light that lights your scene). We want the Default Lighting to be low so it simulates night time and allows us to see our spotlight better. Default lighting is the ambient light that surrounds your scene. Try the settings shown below. **Note:** With Default Lighting, the lower the number you set, the brighter your scene will be.



Try a render now and you should see something like this:



If you click on your light and see the two cones radiating out from the light icon, you'll see the Hotspot (inner cone) and the Falloff (outer cone). You can change these settings to control your light even further. These are also in your Properties palette. These settings control how wide of an area is lit by your spot light. Try some various settings and render each time to see the effect of your settings. Note: Older versions provide grips on the spotlight object that let you adjust these settings.

×			
ļi I	Light 🔹 📌 🛧		
×	General	-	
	Name	Spotlight1	
	Туре	Spotlight	
	On/Off Status	On	
	Hotspot angle	40	
	Falloff angle	60 🔳	
	Intensity factor	300.0000	
	Filter color	221,240,95	
	Plot glyph	No	
	Glyph display	Auto	
	Geometry	-	
	Position X	1 1/8"	
	Position Y	15'	
	Position Z	-5'-6 3/8"	
	Target X	1 1/8"	
	Target Y	1 1/2"	
	Target Z	-5'-6 3/8"	
	Targeted	Yes	
	Attenuation	-	
ដ	Туре	Inverse Square	
RTI	Use limits	No	
OPE	Start limit offset		
PR	End limit offset	10"	
PROPERTIES			

## Point Light

Another light option is a point light. Think of this a light bulb you would have in a lamp. Point lights radiate light in all directions evenly (unlike the directional spotlight). In this example, we're going to add a point light inside the building and let its light shine out the windows. We'll keep the twilight look.

From the same light tool panel that you used to place the spotlight, select Point Light and place inside the building in the larger room. Check from other views to make sure that you placed it correctly. You may find that you need to move it. Once you have it placed, perform a render.

You will probably find that the light isn't very bright. In fact it needs to be brighter to show through the windows. Click on the point light glyph and press CTRL+1 to bring up the properties palette. Start with an Intensity Factor of 6 and then start experimenting from there. Remember to render after each time you set it.



In the image above, you can see the I have changed the **Intensity Factor** to 6. This should make the light bright enough to be seen. Change this setting in your drawing and try another render.

Below is what I have after moving the view slightly (with 3DO) to show more of the ground where the light from the window hits.



Now that you have a decent looking image, you can try some more settings. try increasing or decreasing the Intensity Factor for the Spotlight. Try adding another point light inside. Remove the interior door to get light into the small room. Try adding a street light to light the exterior of the building a little. Add a Distance light off to the side and try different intensities. Yes, I know I didn't show you how to add a Distance Light, but I believe that you can figure it out now (distance light is the easiest).

To see how many lights you have in your drawing and access them quickly for changing settings, click on the bottom right corner of the Lights tool panel or use the LIGHTLIST command.



Draw a light in the yard and add a point light where the bulb should go. Add some window panes to get more dramatic lighting.

The most important thing with lighting is to **experiment**. Try different locations and settings. Now that you know the basics, it's up to you to explore.

## **Render Settings**

Now when you render a final image, there are many options available to you. Mostly it depends on the end use of the rendering. To adjust these settings, type in **RPREF** (Render > Advanced Render Settings...) to get this palette full of options.

×	Render in:	Winc		-	
×	Render Size:	1920	x 1080 px - Full HDT	ν -	
	Current Preset:	Lunc	h Quality	-	* ×
	Preset Info				<del>, -</del> 0
	Name:		Lunch Quality		
	Description:		Renders for 60 minu	utes	
	Render Duration	i			-
	🔿 Until Satisfa	ctory			
GER	🔿 Render By L	evel:			
RENDER PRESETS MANAGER	💿 Render By T	ime:		60 📜	minutes
SETS	Lights and Mate	rials			=
PRES	Render Accurac	y:	O Low		
IDER			🔘 Draft		
REP			💿 High		
-					

To get a good quality rendering like the one above try the "Lunch" setting (it won't really take 60 minutes). Make sure that you have materials and shadows on. Render out to the window and choose the size that you want it to be. Once you have the setting you want, type **RENDER** again and view your results. If you like what you have done, you can save an image file from the render window.



Note the difference between the two settings. Looks a the grass under the spotlight for a great example of how you can use "Low" to get a quick representation, but you want to use "Lunch" or better for a final rendering. The more complex your model and more lights you have, the longer your render will take.

Once you have a rendering that you are happy with, you should save it out to a file. Just click the Disk icon on the top left of your rendering window and save the image as a JPG.



That's the basics of rendering, practice with different settings and even experiment with light.

Once again, you have learned some easy tools that will allow to produce a good rendering of an object. Of course there are many variations and a lot of detail would go into a larger corporate presentation, but it only takes time, practice and experience.

**Extra Practice:** Many people will measure out their home or a room and model it and then render it. This is a great exercise. The more detail you add, the better your model will be. Also consider taking a photo of your wall or desk finish to make custom materials.

Extra Practice: Create a scene using your Lamp from Lesson 3-8 and the solid chair you made. Add walls, a floor and end table for your lamp. Create materials, map them and then add a Point Light to your lamp. Try to render the scene to be as realistic as possible.

#### Video: Lighting and Rendering



## Take the Lesson 3-17 Quiz

Can you set a background in the View Manager?

No Only Gradients Yes

## For Further Thought:

As I mentioned in a previous summary, adding the correct materials and mapping them properly is very important. Just like in photography, the lighting in your 3D models is also very important. If you're trying to sell a design, the proper lighting will help. You might need to put a model in a photo - like a building rendering in a photo of the property it is designed for. Everything from the model, the materials to the lighting all come together in the final presentation. It is up to you to make it look its best. Have high standards.

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## **4-1: AutoCAD Template Files**

LESSON 4-2	USING OSNAPS IN AUTOCAD
LESSON 4-3	LINETYPES AND LINETYPE SCALE
LESSON 4-4	AUTOCAD FILE FORMATS
LESSON 4-5	GRIP EDITING
LESSON 4-6	AUTOCAD'S INQUIRY TOOLS
LESSON 4-7	XREF'S (EXTERNAL REFERENCES)
LESSON 4-8	SYSTEM VARIABLES
LESSON 4-9	TABLES
LESSON 4-10	INTRODUCTION TO SHEET SETS
LESSON 4-11	MODIFYING AND CREATING DIMENSIONS
LESSON 4-11a	TYPES OF DIMENSIONS
LESSON 4-12	INTRODUCTION TO CUSTOMIZING

#### AUTOCAD

### Topics covered in this Lesson:

#### Creating and Using AutoCAD Template Files

Whether you know it not, you have already used templates to begin a new drawing. AutoCAD uses a template every time it starts up. If you do not designate your own template, AutoCAD will use a default one.

A template is a drawing file that includes some of the following settings:

Unit type and precision (DDRMODES)

Drawing limits (LIMITS)

Snap, Grid, and Ortho settings (Status Bar)

Layer organization (LA)

Title blocks (Tutorial)

Dimension and text styles (DDIM, ST)

Linetypes (LT)

Common blocks (Tutorial)

In most cases, you do not want to set these things every time you begin a drawing. By having a template with all of these parameters pre-set, you can work more efficiently, faster and consistently. There is no difference between a template file (DWT) and regular DWG other than the extension.

Most CAD businesses use a company-wide template that is updated from time to time. This is done to maintain CAD standards through the company. Imagine if 50 different designers each used a different set of layers and blocks.

Occasionally, you will use a client's template. When you start a project, you will be told which template you are to use.

## Creating a Template

To create a template drawing, you first have to set up any parameters (listed above) that you feel you would need in a regular drawing. Once you have this, you can save your drawing as a template. Do to this, press **CTRL+SHIFT+S** to get the 'Save As' option.

You will see this dialog box:



You have to change the 'Files of type' setting from a DWG file to a DWT (template) file.

Once you've changed this, make sure you save it in the folder where you can load it later. You'll then be asked to add a description and other information in the next small dialog box. Remember - you might not be the only using this file - so give as much information as you can. It's a great habit to learn.

👗 Template Options	×
Description Decimal drawing template. Used for example drawings at myCADsite.com	OK Cancel Help
Measurement Metric  New Layer Notification  Save all layers as <u>unreconciled</u> Save all layers as <u>reconciled</u>	

You can also use the Saveas icon at the top of the AutoCAD window.



To use the template for a new drawing, type **NEW** or press **CTRL+N** to start a new file and select the template you want.

## Video: AutoCAD Template Files



## Take the Lesson 4-1 Quiz

DWT DWG TEM

What is the file extension for a template?

### For Further Thought:

As you work through this level you'll learn a number of concepts that you will need to know about in your daily drafting. These are subjects that didn't fit in with the 'flow' of the other levels, so I have grouped them into a separate level.

This doesn't mean that they aren't as important as some of the other tools you've learned. In fact, these last lessons are the ones that will give you a more rounded AutoCAD eduction. As you work through them, try to think of the bigger picture of how they might fit into the type of drafting you plan to do.

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# Topics covered in this Lesson:

Using Object Snaps (Osnaps) in AutoCAD

Object snaps (Osnaps) are one of the handiest tools in AutoCAD. Without them, you would have a lot of trouble doing even the simplest dimension accurately. Before discussing how they are used, first think about what they are. Their name, "object snaps" means that they allow you to snap to objects, or more precisely, you snap to specific parts of an object. A line for example has 3 points that you can snap to: a midpoint and two endpoints. A circle has 5 points: a center and 4 quadrants. To use Osnaps



effectively, you must know what points an object has that you are able to snap to.

There are also some Osnaps that are not particular points. These allow you to snap to (sometimes) arbitrary points along an object. These would include nearest, tangent, perpendicular and to some extent, intersection.

There are two different ways of working with Osnaps; in running mode or invoking them individually as required. Running Osnaps mean that certain ones are turned on and running in the background. This means that they are available when needed in the middle of a command. Generally, you'll leave some running that you use commonly like endpoint, midpoint and center.

Sometimes you may need one Osnap that isn't one you have selected in running mode. In this case you can turn it on for one-time use. There are a few ways of doing this: you can select the icon for the Osnap you need, you can type in the 3 letter shortcut, or you can bring up the Osnap dialog box and turn it on ('os), or even shift+right-click and pick from the menu. I recommend learning the 3 letter shortcuts and typing them in when required. This is generally the fastest way to get your Osnap on, plus it doesn't leave it on if you don't need it afterwards. Having the icons up your screen works, but takes up valuable screen area. Try each method and see which one works best for you.

In most cases you do not want to turn on all the osnaps while drawing as you could accidentally snap to a point you don't want. For example, the endpoint and a perpendicular point could be very close - so you need to be careful when using perpendicular.

Below is the Osnap dialog (**os**) box followed by a detailed listing of how each one is used.

👗 Dr	afting Settings			
Sna	ap and Grid Polar Tracking Object Sna	ap 3D Object Snap Dynamic Input Quic 4 >		
	Object Snap On (F3)	☑ Object Snap Trac <u>k</u> ing On (F11)		
	Object Snap modes			
	Endpoint	Extension Select All		
	△ Midpoint C	Clear All		
	⊖ <u>I C</u> enter E	_ Perpendicular		
	C Geometric Center C	5 🔲 Ta <u>ng</u> ent		
	⊠ No <u>d</u> e ∑	∠ Nearest		
	🛇 📝 Quadrant	Apparent intersection		
	X Intersection	🖉 🖻 Paraļlel		
	To track from an Osnap point, pause over the point while in a command. A tracking vector appears when you move the cursor. To stop tracking, pause over the point again.			
0	ptjons	OK Cancel <u>H</u> elp		

Here is a table which lists all available Osnaps in AutoCAD:

ICON	SHORTCUT	SETTING
₽0	TT	TEMPORARY TRACK POINT
i.	FROM	SNAP FROM
P	END	ENDPOINT
ø	MID	MIDPOINT
	EXT	EXTENTION
•	CEN	<u>CENTER</u>
0	CE	GEOMETRIC CENTER

•	NOD	NODE
$\diamond$	QUA	QUADRANT
×	INT	INTERSECTION
<mark>~</mark> ~	INS	<b>INSERTION POINT</b>
*	PER	PERPENDICULAR
Ó	TAN	TANGENT
≁	NEA	<u>NEAREST</u>
×	APP	APPARENT INTERSECTION
ſ <b>č</b>	NON	NONE
<b>n</b> .	OS	OSNAP SETTINGS
-	M2P	Middle of 2 points

To see the icons, **Shift+Right-Click** while in a command like Line where an Osnap would be used.



The 3 letter shortcut is shown in bold in the table above. Notice that it is the first 3 letters of each Osnap.

Also note the Osnap symbol beside each setting in the image of the dialog box above. These will be visible on your screen as the osnap becomes active.

#### LISTING OF OSNAPS MODES





**OSNAP TRACKING** - While not really an Object Snap, this option allows you to pick any point on the screen and track from there. This point can be on an object or not. Toggle on and off with **F11**.

**FROM** - This option allows you to select an Osnap and use it as a reference point. Very similar to Temporary Tracking.



**ENDPOINT** – This snap is used to get to the exact endpoint of a line, arc or other object that has a definite ending to it. This should be used for joining lines, and dimensioning. In general, you would want this on as a running Osnap in your drawing. The endpoint Osnap is indicated by a small square.



Center

**MIDPOINT** – This is used to find the exact middle of any object that has a beginning and an end. All lines and arcs have a midpoint. (Circle have a center, not a midpoint.) This is also a good Osnap to have in running mode. The midpoint Osnap is indicated by a small triangle.

**CENTER** – The center Osnap is used to find the exact center of circles, arc and ellipses. This is also a good Osnap to leave in running mode. To pick the center point, you have to select the object itself, and not an area in the middle of the object. The center Osnap is indicated by a small circle.

**GEOMETRIC CENTER** – This is a new Osnap that is

really handy. Use it when you need to snap to the 'geometric' center of a rectangle rather than trying to





find the two midpoints. Works on closed polylines, rectangles, splines **NODE** – Nodes are points. The are created using the point command. They are also created

when an object is divided. A node Osnap is indicated by a circle with angled crosshairs in side.













**QUADRANT** – Quadrants are the 'corners' of circles and ellipses. Arcs can also have quadrants. The quadrant Osnap is indicated by a diamond. The are four quadrants on every circle.

**INTERSECTION** – The intersection Osnap appears wherever two objects cross. If you select one object you'll see a cross with three dots indicating that you have to select a second object to find an intersection. Pick on the second object and AutoCAD will find the intersection for you. This is handy when you have a lot of lines in the same area. An 'X' indicates the intersection Osnap.

**INSERTION** – The insertion Osnap will snap to the insertion of points of objects such as blocks, text and attributes. Each object will be different depending on how it was created. The insertion Osnap is shown as two small, offset squares.

**PERPENDICULAR** – The perpendicular Osnap is used to draw a line from one point to another point at a right angle (90degrees) to an object. This Osnap can save you having to trim or extend afterwards. If you use the perpendicular Osnap for starting a line, you will get the indicator with 3 dots telling you that the next point is needed to establish the starting point. The perpendicular Osnap is indicated by a square with two extending lines.

**TANGENT** – Tangent Osnaps are used on circles, arcs, ellipses and splines. A line tangent to an arc gives you a smooth line off the curve. If you select the tangent Osnap as the first point, your indicator will have three dots following it. This means that the tangential point can not be established until the second point is given. The tangent Osnap is shown as a circle with a line across the top of it.

**NEAREST** – The nearest Osnap does just as the name implies: it will find the closest point on an object relative to where you started. This is handy for measuring distance and drawing quick lines, but can lead to erroneous results when dimensioning or drawing. This Osnap should rarely be used in running mode, and never for dimensioning. The nearest Osnap is indicated by a triangular hourglass.



**APPARENT INTERSECTION** – The Apparent Intersection Osnap is used when two objects appear to intersect on the screen, but do not truly intersect in 3-D space. It also works when any two objects do not intersect, but you need to find the point where they would. This Osnap will also find the true intersection of two crossing objects. This Osnaps has different indicators depending on which mode is needed at the time.

**NONE** – This command is used to turn off any Osnaps that may be on in running mode. You can also disable your Osnaps by hitting the F3 key, which would be a faster way of doing this.

Added In AutoCAD 2005, there is a new Osnap "Midpoint Between 2 objects" - it's not in the Osnap dialog box, but can be invoked when needed by typing "MTP" or "M2P" or using the shift+right-click option.

Command: L LINE Specify first point: MTP First point of mid: <PICK> Second point of mid: <PICK> Specify next point or [Undo]: (Select point or enter to end)

## OTHER NOTES ABOUT OSNAPS:

Osnaps are only available when you are currently in a command.

Toggle Osnaps on and off by using the **F3** key. If you do not have any Osnaps on in running mode, the F3 key will invoke the Osnap dialog box. Toggle Object Snap Tracking with **F11**.

Get into the habit of working with the same 2 or 3 Osnaps in running mode whenever you are drawing. The more consistent you are, the easier it is to keep track off while working.

When dimensioning, you must be VERY careful where you are snapping to. Make sure that you are not using the nearest, perpendicular or intersection Osnaps unless you specifically require them. Below is an image that shows a common mistake of dimensioning to another dimension line and not the object. Often you will catch this if your measurement is an odd number (should be 11 in the example below).



If you have a lot of objects in one area, and need a particular Osnap, use the **TAB** key to cycle through all available Osnaps in the area. As you hit the TAB key, you will see that the Osnap location changes and AutoCAD highlights the object you would snap to.

Zoom in when in doubt. Make sure that you are snapping where you need to. Snapping to the wrong part can cause catastrophic events.

If you want to invoke the Osnap dialog box while in the middle of a command, type in **'OS** at the command prompt. You can then make your changes and then continue with the command.

#### CHANGING THE OSNAP'S APPEARANCE

If you want to change the appearance of the Osnap indicators, such as size or color, bring up the Osnap dialog box and pick on the Options button on the Osnap Dialog box).

A Options	
Current profile: <>	Current drawing: BASIC-TEMDECIMAL.dwt
Files Display Open and Save Plot and Publish System User	Preferences Drafting 3D Modeling Selection Profiles Online
AutoSnap Settings          Image:         Image: <tr< td=""><td>Auto Track Settings</td></tr<>	Auto Track Settings
Object Snap Options          Ignore hatch objects         Ignore dimension extension lines         Ignore negative Z object snaps for Dynamic UCS         Replace Z value with current elevation	Drafting Tooltip S <u>e</u> ttings Lights Glyph Settings C <u>a</u> meras Glyph Settings
	OK Cancel Apply Help

On the left side of the dialog box, you can set the 'look' of the Osnaps when you are using them.

The Marker, Magnet and Snap Tip should be left on. Marker shows the active osnap. Magnet 'snaps' you into position when you are near. The Snap Tip is a small text box telling you which osnap is current. Leaving the aperture box off, makes it easier to see the snap points.

The Marker Size is a matter of personal preference, but the default seems good to me.

The color of your OSnap might need to be changed depending on the color of the objects you are working with or screen background color.

## Video: Using OSnaps in AutoCAD

🕒 Share / Save 🖪 🈏 🖻 ...

### Take the Lesson 4-2 Quiz

Which object doesn't have an endpoint?

Circle			
Line			
Arc			



## For Further Thought:

Osnaps are one of the most powerful tools you have in AutoCAD. You will use them all the time, so it's important that you understand how they work and how to use them efficiently.

Using Osnaps will save you a lot of time, but using them incorrectly can seriously mess up your drawing. Make sure you are comfortable using them and although it's tempting to turn them all on when you are starting out - don't. Just don't. Have the few active that you will be using and activate the others as you need them.

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## **CONTINUE TO THE NEXT LESSON: 4-3**



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Last Update: 02/25/2019 18:25:17

#### Topics covered in this Lesson:

Linetypes | Linetype Scale | Lineweights

When you have been drawing in AutoCAD during these tutorials, almost all of the lines have been **continuous**. This is a particular linetype. Most simple drawings can be drawn with just this one linetype. More advanced drawings will require different linetypes such as center lines, hidden lines, phantom lines and others.

When you first start AutoCAD, the default template has only one linetype is available. This is the continuous linetype. AutoCAD has many more available, but only loads in one to start with in order to keep the drawing file size smaller. If you need a different linetype, you must **load** it into your current drawing. As is usually the case in AutoCAD, there are a few ways to do this.

Here are some of the commands that you'll you be learning in this tutorial:

Command	Keystroke	Location	Result
Linetype	LINETYPE / LT	Home > Properties > Li <u>n</u> etype	Opens the LINETYPE dialog box
<u>Linetype</u> <u>scale</u>	LTSCALE / LTS	None	Allows you to set the linetype scale globally.
Lineweight	LINEWEIGHT / LW	None	Set line weight globally

## Linetype

Invoking the **LINETYPE** command brings up the Linetype Manager dialog box as shown below.

Linetype Manager	X			
Linetype filters				
Show all linetypes   Invert filter  Current Show details				
Current Linetype: ByLa	yer			
Linetype	Appearance Description			
ByLayer ByBloc CENT Contin HIDDE	Reload Linetypes			
PHAN Available Lin	etypes			
Linetype	Description			
BATTING BORDER BORDER2	Batting SSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSSS			
BORDERX2 CENTER	Center Center			
CENTER2 CENTERX2	Center (.5x)			
•				
	OK Cancel Help			
C				
	OK Cancel <u>H</u> elp			

You'll notice that it is similar to the layer dialog box.

Instead of creating a new linetype (like you would a layer), you will have to **LOAD** it into your drawing. AutoCAD has many different linetypes that you can load, as well as giving you the option to create your own. (This will not be covered in this lesson.) To load a linetype, press the **Load...** button that is towards the top-right corner. When you do this, you'll see another dialog box appear (shown above).

Notice that AutoCAD is giving you a choice of different linetypes as defined in the **acad.lin** file. (The acad.lin file is the default set of linetype definitions that come with AutoCAD when you install it. Your company may have different .lin file that it uses for custom linetypes.) Scroll through the list to see what options you have available. You'll see the most common ones (hidden, center, etc) as well as some

that are only for certain applications. Click on the **Hidden** linetype and then press **OK**.

Adding a linetype to your drawing is that easy. Notice that there are three different hidden linetype options: HIDDEN, HIDDEN2, HIDDENX2. All of these are valid linetypes, but as you can see from the samples shown in the right side of the dialog box, they are slightly different. Which one you choose is up to you. **Remember one thing though.** If you pick HIDDEN2, then you should also pick CENTER2, if you need a center line. This will keep your **linetype scale** consistent. Only change your linetype scale separately on objects if you have a specific reason to.

Here is an example of 3 different linetypes:

HIDDEN HIDDEN2 HIDDEN22

## LINETYPE SCALE

Your linetype scale determines how the linetype is displayed and plotted. Depending on your linetype or original area you set up, you may have to change it. This is one more reason, why you should set up your drawing properly from the beginning. If you need to change your linetype scale, type in LTSCALE (LTS) and try different values to get the look you want.

Your linetype scale will depend upon the size of the objects in the drawing. A larger floor plan will require a larger Linetype Scale than a small mechanical part drawing.

You can also change you linetype scale from the LINETYPE dialog box by changing the value in the Global Scale Factor box.

🔺 Linetype Manager				×
Linetype filters Show all linetypes Current Linetype: ByLayer	Load Curren			
Linetype	Appearance	Description		
ByLayer ByBlock BORDER2 CENTER2 Continuous HIDDEN2 PHANTOM2		Center (.5x) Continuous Hidden (.5x) _	·	
Details Name:			<u>G</u> lobal scale factor:	1.0000
D <u>e</u> scription:			Current object scale:	1.0000
✓ Use paper space units	for scaling		ISO gen width:	1.0 mm 💌
			OK Cance	l <u>H</u> elp

**Note:** you can change the linetype scale separately on each object, but this is not recommended as it can be very difficult to keep track of, and therefore lose consistency.

Here is a single linetype (hidden) with 3 different linetype scales applied. Notice that the line with a LTS of .5 has lines and dashes that are 1/2 the size of line above it. The line with a LTS of 5 has lines and dashes that are 5 times longer.

HIDDEN - LINETYPE SCALE =1

HIDDEN - LINETYPE SCALE = .5

HIDDEN - LINETYPE SCALE = 5

To change the linetype of an existing object, just right-click on it and choose "Quick Properties" from the context menu.



Sometimes you'll want to create a layer just for certain linetypes. This can help you manage a drawing that have a lot of different linetypes and allows you to make quick easy global changes.

Current	layer: BORDER					Sea	rch for lay	er 🔍	X
	i 🕯 🚳 🗐	\$ <b>5</b> \$ 5						C 🛱	×
_	Image: select Linetype   Loaded linetypes		Color white white red gree y		Default Default Default Default	0 0 0	Plot Styl Color_7 Color_7 Color_7 Color_3 Color_7 Color_2 Color_7	LAVER PROPERTIES MANAGER	
	CENTER2 Continuous HIDDEN2 PHANTOM2		_		Center (.5 Solid line Hidden (.9 Phantom	5x)	•		TA)

Just like an object's colour, this process allows to set the linetype using the **ByLayer** option. For example, you could have all of your hidden lines on one layer so you can turn them off all at once or give them a light lineweight.

## LINEWEIGHTS

Another property of AutoCAD objects is their lineweight, or how wide they display on the screen and when printed. One common example of a heavier lineweight would be a border around a title block. Some times, you may use a lighter lineweight for hatching. But whatever you use them for, they are powerful display options.

Here are the range of lineweights available (There are more options in between):


The methods for changing the lineweight for objects is similar to the ones for linetypes (above).

Cur	rrent	layer: BOR	DER						Sea	rch for lay	er	٩	X
Ē		- <b>-</b>	<b>9</b>	• <b>E</b>	Ð						C	✿	×
<b>»</b>	S	Name		0	Fre	L	Color	Linetype	Lineweight	Trans	Plo	t Styl	
	-	0		1	<u> </u>	ц,	white	Continuous	Default	0		or_7	
	$\checkmark$	BORDER		<b>?</b>		6	white	BORDER2	<u> </u>		_	or_7	
	_	Defpoints		1	- <u></u>	۰,	white	Continuou	Deiauit			or_7	
		DIMS OBJECT		X	- <u></u>	۰	red	Continuo	Default			or_1	GER
	_	TB		1		9	green white	Continue s Continue us	Default Default			or_3 or_7	NA
	-	TEXT						Contine as		0		or_2	MA
	_	TITLE_BLO	)		🛕 Line	weigh	nt		? X	0		or_2 or_7	TIES
» \$	<ul> <li>A</li> </ul>	ll: 8 layers	disp	lay	New	nal: (	Defaul 0.00 m 0.05 m 0.09 m 0.13 m 0.15 m 0.20 m 0.20 m 0.25 m 0.35 mm 0.35 mm 0.35 mm	im im im im im im im im	F F Help			F	IN LAVER PROPERTIES MANAGER

There is also the option of turning the lineweight display on or off. Just click on the LWT icon of the status bar. Below the icon indicates that lineweights are shown in the drawing. As you can see below, the keyboard command for toggling this is **LWDISPLAY** (1=on / 0=off).



Type in **Lw** to access the Lineweight Settings Dialog box.

🔺 Lineweight Settings					
Lineweights ByLayer ByBlock Default 0.00 mm 0.05 mm	Units for Listing <ul> <li>Millimeters (mm)</li> <li>Inches (in)</li> </ul> <li>Display Lineweight <ul> <li>Default</li> <li>0.25 mm</li> </ul> </li>				
0.09 mm	Adjust Display Scale				
Current Lineweight: ByLayer					

The options are quite straightforward, but you also have a third option of displaying lineweights or not (Display Lineweight) and changing the default display lineweight.

By working with different linetypes and lineweights you can make a drawing clearer. Some people will make their objects have a heavier lineweight than the dimensions to make them stand out better.



**Extra Practice:** Make an **isometric** drawing that uses a heavier lineweight for the main objects and a lighter lineweight for hidden lines.

## Video: Linetypes and Lineweights in AutoCAD



#### Take the Lesson 4-3 Quiz

Which linetype has the longest lines and breaks?

Hidden

Hidden X2

Hidden 2



## For Further Thought:

Having your linetypes looks correct is a small part of making sure that your AutoCAD drawings look professional. It's all part of the overall communication from you as the drafter to the person reading the drawing. Imagine a hidden line that looks solid when your print it. How could that affect the production of the part you're asking someone to build? How about if you miss some centre lines? Does that mean the drawing has to come back to you? If so, you've just wasted valuable time. Remember, these are all tools for communication. Make sure your drawing speaks a clear language.

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# RETURN TO THE TOP OF THE LESSON | RETURN TO THE TABLE OF CONTENTS

# **CONTINUE TO THE NEXT LESSON: 4-4**



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## Topics covered in this Lesson:

#### File Formats | Images | Recovery

Every file in a computer is created as a particular type or format. For example, the file you're looking at is a HTML file used on the web. A standard AutoCAD drawing is created as a DWG file. The difference is the way that the application program recognizes the file. Each file type also has a distinct structure so that the program that created it can read it correctly. Your operating system knows which file types are associated with which programs.

In the 'olden' days when DOS was the common operating system, all files had a maximum of 8 characters in their name, followed by a period, followed by the extension. The extension was a 3 letter 'code' indicating the type of file.

Example: DRAWING1.DWG

DRAWING1

Name of the file

DWG

After the introduction of Windows 95, you could have more than 8 characters in a name (up to 256), but the 3 or 4 letter extension remains. Windows is even set up to hide these extensions from you in its default configuration, so you might want to turn them on to make things easier for you.

While working with AutoCAD, it is important to know that there are several types of files you'll be working with. Here are some of the common filetypes you will come across:

#### DWG - BAK - DWT - DWF -SV\$ - DXF - WMF - 3DS - JPG

EXTENSION	USAGE
DWG	The standard AutoCAD drawing file extension. This has been consistently used for decades, but the extension does not denote the version the file was created in.
BAK	A Backup File that can be automatically created each time you save your drawing file.
DWT	Used for AutoCAD Template files.
DWF	The Drawing Web Format is used for sharing files either online or through a DWF viewer.
SV\$	The extension used for Autosave files created automatically at a set interval
DXF	The Drawing Exchange Format is used to export files for use in other software programs.
PDF	The Portable Document Format is used in Adobe Acrobat files and can be easily viewed.

These are only a few of the file types that AutoCAD uses. There are many others that would be considered support files. Those listed above deal with various forms of drawing files. There are many other ways to save your drawing, but these are the common ones.

**DWG** – This is the standard AutoCAD drawing file format. The thing to remember is that older versions of AutoCAD cannot read files created on newer versions. The newest version can read any of the older files. If you are exchanging files with other companies, do not assume that they are using the same version you are. Some coworkers will also have older or newer versions than the one on your computer.

Below you can see many of the options available when you save an AutoCAD file. Sometimes I'll put the version number in the file name for later reference.

🛕 Save Drawing As						×
Save <u>i</u> n:	길 cad-16		•	•	🜊 💥 📮 Views	▼ Too <u>l</u> s ▼
	Name	*		Тур	Preview	
2 North Carlor C	notes			File		
History	autocad-	tests		File		
R	I flex-slide	r		File		
	퉬 images			File		
Documents	퉬 other			File		
	scripts			File	Options	
	search			File		
Favorites	styles			File	Update sheet and vie thumbnails now	ew
	tutorials			File File	and non-	
	Videos			File	Copy Design Feed (or to enable)	og in to A360
Template					to enable)	
(Canada and Canada and						
Desktop						
	•	III		•		
cad-16						
12	File <u>n</u> ame:	Awesome-Drawing-R1-	2007.dwg		•	<u>S</u> ave
Buzzsaw 🚽	Files of type:	AutoCAD 2010/LT201	0 Drawing (* dwg	)	•	Cancel
	Thes of type.	AutoCAD 2013 Drawin				
		AutoCAD 2010/LT201	0 Drawing (*.dwg			
		AutoCAD 2007/LT200 AutoCAD 2004/LT200	4 Drawing (*.dwg	)		
		AutoCAD 2000/LT200	0 Drawing (*.dwg	)		
		AutoCAD R14/LT98/LT97 Drawing (*.dwg) AutoCAD Drawing Standards (*.dws)				
		AutoCAD Drawing Tem AutoCAD 2013 DXF (*.				
		AutoCAD 2010/LT201	0 DXF (*.dxf)			
		AutoCAD 2007/LT200 AutoCAD 2004/LT200	/DXF(".dxf) 4DXF(*.dxf)			
		AutoCAD 2000/LT200 AutoCAD R12/LT2 DX	0 DXF (*.dxf)			

When you save an AutoCAD drawing you can save it for earlier versions. For example, you might have coworker who is working in AutoCAD 2008, so you could save it as an AutoCAD 2007 file for them. You can not save to a newer version than the version you are working on. Here is a handy table that shows which option to choose depending upon the version that is being used.

Save As:	For AutoCAD Version:
----------	----------------------

AutoCAD 2013	AutoCAD 2013 - AutoCAD 2016
AutoCAD 2010	AutoCAD 2010 - AutoCAD 2012
AutoCAD 2007	AutoCAD 2007 - AutoCAD 2009
AutoCAD 2004	AutoCAD 2004 - AutoCAD 2006
AutoCAD 2000	AutoCAD 2000 - AutoCAD 2002
AutoCAD R14	AutoCAD R14
AutoCAD R12 DXF	AutoCAD R12

**BAK** – This is AutoCAD's backup file format. Whenever you save a drawing, AutoCAD will automatically create a duplicate backup file. This file has all the same information as the original, but a different extension. If your original file becomes corrupt, or unusable for any reason, you can rename the BAK file to a DWG file and open it as you would any other drawing file. You can set the interval for backing up your file in the Options (OP) > **Open and Save tab** set for as short as possible with interrupting your workflow.

A Options	
Current profile: < <unnamed profile="">&gt;</unnamed>	🐴 Current drawing:
Files Display Open and Save Plot and Publish System User P	references Drafting 3D Model
File Save   Save as:   AutoCAD 2010/LT2010 Drawing (*.dwg)   Maintain visual fidelity for annotative objects   Maintain drawing size compatibility   Inumbnail Preview Settings   50   Incremental save percentage	File Open 9 <u>N</u> umber of recently- Display <u>f</u> ull path in title Application Menu 9 Number of recently- External References (Xrefs) Demand load <u>X</u> refs: Enabled with copy
<ul> <li>Automatic save</li> <li>10 Minutes between saves</li> <li>Create backup copy with each save</li> <li>Full-time CRC validation</li> <li>Maintain a log file</li> <li>ac\$ File extension for temporary files</li> <li>Digital Signatures</li> <li>Display digital signature information</li> </ul>	<ul> <li>Retain changes to X</li> <li>Retain changes to X</li> <li>Allow other users to f</li> <li>ObjectARX Applications</li> <li>Demand load ObjectARX apps</li> <li>Object detect and command i</li> <li>Proxy images for custom object</li> <li>Show proxy graphics</li> <li>Show Proxy Information dia</li> </ul>
	OK Cance

You also have the option of saving a .BAK file with your drawings as well. You can check this in the dialog box shown above or use the **ISAVEBAK** system variable (1=backup is saved - 0=NO backup is saved).

**DWT** – This is a filetype that is used for drawing templates. Templates are the same as prototype drawings in that they contain all the setup parameters for certain types of drawings. They are essentially the same as a DWG file. You change the extension from .DWT to .DWG and open the files as a regular drawing file. Learn more about template files in Lesson 4-1.

**DWF** – This is the **Drawing Web Format**. This type of file contains features that allow users to view drawings on the Web (Internet Explorer) without having AutoCAD on their computer. You can also view DWF files with the free Autodesk Design Review program (<u>download here</u>). Apps are available for Android and iOS to view DWF files as well.

Use the **DWFOUT** command to open the Plot dialog box pre-set to DWF setting.

Using DWF files is a good way to share drawings without having to send someone an editable CAD file. For example, you might send the file to a manager for review. He can then view the file, make measurements and add comments and send it back to you for revisions. Below is how the drawing file might look in the Autodesk Design Review program.



**SV\$** - This is the file format used whenever AutoCAD performs an automatic save. AutoCAD will save the file automatically within a pre-determined time frame. This is set in your Options > Files dialog box (see the Options dialog box above). You can also tell AutoCAD where you want your autosave files to go. Most computers are set to save to the Windows/Temp folder. If you need to recover a drawing from the autosave file, just rename it to a DWG format.

You might consider changing the location of your autosave folder to make it easier to locate the files when you need them.

A Options	
Current profile: < <unnamed profile="">&gt;</unnamed>	🛅 Current drawing:
Files Display Open and Save Plot and Publis	sh System User Preferences Drafting 3D Modeli
Search paths, file names, and file locations:	
🕀 Working Support File Search Path	
🗄 🕀 Trusted Locations	
🗄 🖷 🔁 Device Driver File Search Path	
🕀 🐨 📳 Project Files Search Path	
🗄 🕀 Customization Files	
🕂 🕂 Help and Miscellaneous File Names	
🕂 🕀 Text Editor, Dictionary, and Font File N	lames
Print File, Spooler, and Prolog Section	Names
Printer Support File Path	_
Automatic Save File Location	
C:\Users\Me\appdata\local\temp	>\
🕂 🖓 🔁 Color Book Locations	
	Contains the drive and path for automatic sav
Template Settings	
🗄 ···· 🍋 Tool Palettes File Locations	
🗄 ···· 🍋 Authoring Palette File Locations	
🗄 🕀 Eog File Location	
Action Recorder Settinas	
	OK Cance

DXF – This is not really an AutoCAD format but an industry standard, but one that you should be aware of. DXF stands for D rawing e**X** change **F** ormat. This is a very standard format that is used but many different CAD and graphics programs. This

allows users to exchange drawings even if they don't have the same program. When you use the DXF format, some objects may change their appearance when reopened. As with DWG formats, DXF formats vary from different releases. You have the option of saving the files as a DXF or you can use the **DXFOUT** command, conversely DXF Files can be imported using the **DXFIN**command.

Just as with the Save as command, you can save for earlier version of AutoCAD as well.

**PDF** - Although not a AutoCAD specific file format, you can now save your drawing as a PDF (Portable Document Format) so that others can read it with the common Adobe Acrobat Reader. Use the Saveas... PDF option.

Here's a look at how a file appears in the DWF viewer (left) and the Acrobat PDF viewer (right) when zoomed in. Note the sharpness of the text and the thinner lines in the DWF viewer.



**WMF** – This is a Windows Meta File. This file can be used if you need to use this drawing in a non-CAD program. For example, you may need to add a drawing of section of a drawing to a Microsoft Word document. You would use this format to do that. You can export only a portion of your drawing with the **WMFOUT** command. Use the **WMFIN** command to import a WMF file.

In Word, you can use the Insert > Picture menu option to bring the WMF file into your document. You can also Copy and Paste from AutoCAD into Word.

**3DS** - This is an extension for 3D Studio - a high level animation and rendering program. You can import to AutoCAD by using the **3DSIN** command. AutoCAD used to contain a 3DSOUT command, but removed it. If you want to use this command in newer version (2007 and later) you will have to install an extension (**download**).

**OTHER CAD FILES** - There are a lot of CAD programs out there and each has its own way of saving files. AutoCAD allows you to import many of these using the **IMPORT** command. I won't cover all of the formats here, but if someone sends you a CAD file that you can't open, you can try using the Import command.

🛦 Import File			<b>X</b>
Look in: [	My Documents	- 🖛 🖳 🎕 🗶 📮	<u>V</u> iews <b>v</b> Too <u>l</u> s <b>v</b>
	Vame 🇳	Date modified	Туре
2 A A A A A A A A A A A A A A A A A A A	Add-in Express	6/19/2013 12:42 PM	File folder
History	Adobe .	4/11/2015 11:19 AM	File folder
	📙 Aiseesoft Studio	3/24/2015 4:32 PM	File folder
	📕 Amazon MP3	5/4/2013 2:42 PM	File folder ≡
Documents	]] Any Video Converter	5/30/2013 3:40 PM	File folder
	📙 Arduino	8/11/2015 8:27 PM	File folder
	🕌 Autodesk Application Manager	4/7/2014 9:08 PM	File folder
Favorites	🕌 AutoPlay Media Studio 6.0 Projects	5/12/2011 3:50 PM	File folder
	📙 Camtasia Studio	1/7/2016 8:26 PM	File folder
	🕌 DxO Optics Pro v7 logs	2/7/2013 4:22 PM	File folder
Template	📙 Fax	6/20/2012 1:43 PM	File folder
rempiace	Feature-Presentation-test	7/1/2014 6:13 PM	File folder
	🕌 Fragments	2/21/2011 1:51 PM	File folder
Desktop	🕌 FXhome	5/6/2013 8:02 PM	File folder
Desktop	🍌 Garmin	11/17/2013 2:26 PM	File folder
	🍌 gegl-0.0	6/6/2011 2:32 PM	File folder 👻
cad-16			•
Fil Fil	e <u>n</u> ame:		▼ Open ▼
Buzzsaw 🔻 🗖	les of type: FBX (*.fbx)		Cancel
Fi	les of type: FBX (*.fbx) 3D Studio (*.3ds)		
	ACIS (*.sat) CATIA V4 (*.model;*.session;*.e CATIA V5 (*.CATPart;*.CATPro FBX (*.fbx) IGES (*.igs;*.iges) Inventor (*.ipt;*.iam) JT (*.jt) Metafile (*.wmf) MicroStation DGN (*.dgn) NX (*.prt) Parasolid Binary (*.x_b) Parasolid Binary (*.x_b) Parasolid Binary (*.x_b) Parasolid Text (*.x_t) Pro/ENGINEER (*.pt;*.asm*) Pro/ENGINEER Granite (*.g) Pro/ENGINEER Neutral (*.neu' Rhino (*.3dm) SolidWorks (*.prt;*.sldprt;*.asm; STEP (*.ste;*stp;*.step) All DGN Files (*.*)	duct)	

Image Files

Image files can be used in AutoCAD as a reference, for sketching over or for files to create materials in 3D rendering. There are a few common image files used with AutoCAD. You can also Copy and Paste most images files into AutoCAD.

Image files can be brought into AutoCAD using the **IMAGE** command (makes sense, doesn't it?). Start the **IMAGE** command you'll see the External References palette open.



Once you click the "Attach Image..." menu item, you can select your image and then you'll see another dialog box that lets you select some options similar to what you would see when you insert a block.

🛕 Attach Image			
Name: cabin_floor_plan2	Browse		
Preview	Path type Full path ▼ Insertion point  Specify on-screen  X: 0"  Y: 0"  Z: 0"  O"	Scale Scale Specify on-screen 1.0000 Rotation Specify on-screen Angle: 0	
Show Details	ОК С	Cancel <u>H</u> elp	

**JPG** - Most people know JPGs as the common format on the internet for images. You can also export your drawing easily to a JPG now. Use the **JPGOUT**command and then select your objects. If you select a viewport, the objects shown int he viewport will not be shown in the resulting JPG image. It's usually best to work from Model Space when you do this. You may want to change your background color to white for this.

**GIF** - A gif is generally lower resolution image file as it is limited to 256 colours. It is good for showing things like the dialog boxes that I use in these tutorials, or high contrast images like a line drawing. GIF files can also have a transparent background.

**PNG** - A PNG is a nice compromised between the JPG which have have millions of colours and a GIF which can have a transparent background. Using the PNGOUT command from AutoCAD will create a PNG file, but it will not have a transparent background. PNGs are good images for custom materials that have an odd shape.

**BMP** - This is similar to a JPG, but exports a raster BMP file instead of a JPG. Use the **BMPOUT** command. Generally JPGs are smaller and better for e-mail and websites. Not many people use BMPs any more as the files are much larger and lower resolution than a JPG. For example, exporting the cabin floor plan results in a

114kB JPG, a 53kB PNG and a 2,592mB BMP! There may some older programs out there you need to deal with that only open BMPs, but you should really avoid BMP files these days.

These are the common file types used in AutoCAD. The main thing to remember is that you have the option of sharing your drawing with people who don't have AutoCAD installed on their machines.

#### Recovery

If you see this when you open AutoCAD - don't panic!

Drawing Recovery	<b>x</b>
In your previous work session, the prog terminated unexpectedly. You can resto unsaved changes from a backup file by Drawing Recovery Manager.	ore
<ul> <li>Show details</li> <li>Do not show me this message again</li> </ul>	Close

Sometimes you'll have a file that gets corrupted. Or you might need quick access to an Autosave file. This can happen when AutoCAD crashes, or the power goes out in the middle of a drawing session or the Operating System locks up. Hopefully this is rare, and these days it doesn't seem to happen to me as much.

But when these problems arise, it's good to know that you have some tools in AutoCAD to help you out. One of the best ways to go about restoring a file is to use the Drawing Recovery Manager. To open, click on the Application Manager (Top left icon on the window) and navigate to Drawing Utilities (if needed) and click on Recover (look for the bandaid!)



This will open a sidebar on your drawing screen and show you all of the recent files that may have had problems. Click to open the folder and see what options you have.



Now you have the files you need, you can just double-click on the file you need like the highlighted .SV\$ shown above and AutoCAD will open the file for you. When you do this, SAVE the file immediately with a new name (something like Building\_RECOVERED) so that you know it's a recovered file and may not be as up to date as you think it is. You can also open the Drawing Recovery Manager by using the DRAWINGRECOVERY command.

Between Autosave files and drawing recovery you will likely be able to salvage any drawing mishap.

#### Video: File Types and Recovery

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### Take the Lesson 4-4 Quiz

What AutoCAD filetype can be best used with Microsoft Office?

DWF			
WMF			
DWG			

## For Further Thought:

Is your toolbox full yet? Knowing how to draft in AutoCAD is just part of the job description when you are expected to handle all of your drawings and files. Knowing which AutoCAD files do what job is very important as you delve deeper into the CAD world. Make sure you are aware of these different files types and especially ensure that you are familiar with the recovery process - it could save your job one day.

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# RETURN TO THE TOP OF THE LESSON | RETURN TO THE TABLE OF CONTENTS

# **CONTINUE TO THE NEXT LESSON: 4-5**



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Last Update: 02/25/2019 18:25:52

## Topics covered in this Lesson:

Working with Grips | Points/Nodes

By now you should be familiar with grips. These are the 'blue squares' that get highlighted when you select an object outside of a command.



Grips are handles that allow you to manipulate the object without entering a command directly. Be aware that grip editing can lead to undesired results if you are not careful.

Usually these settings do not need to be changed, but the look of your grips can be modified by entering the **DDGRIPS** command. You'll see this dialog box (it's the Options dialog box: Selection tab):

A Options		
Current profile: <>	Current drawing: cabin_floor_plan2.dwg	
Files Display Open and Save Plot and Publish System User	Preferences Drafting 3D Modeling Selection Frofiles Online	
Pickbox size	Grip size	
Selection modes	Grips	
✓ Noun/verb selection	Grip <u>C</u> olors	
Use Shift to add to selection	Show grips	
✓ Object grouping	Show grips within <u>b</u> locks	
Associati <u>v</u> e Hatch	✓ Show grip tips	
Implied windowing	Show dynamic grip men <u>u</u>	
Allow press and drag on object	Allow Ctrl+cycling behavior	
Allow press and drag for Lasso	Show single grip on groups	
Window selection method:	Show bounding box on groups	
Both - Automatic detection 👻	100 Object selection li <u>m</u> it for display of grips	
25000 Object limit for Properties palette		
Selection effect color:	Preview Selection preview When a command is active	
	When no command is active	
Ribbon options	Visual Effect Settings	
Contextual Tab States	Command preview	
	Property preview	
	E Licked Frage	
	OK Cancel <u>Apply H</u> elp	

Notice that you can enable or disable grips, as well as control whether they are enabled within blocks or not. The default settings should work for most situations.

You can change the colors so that they are more visible depending on the particular drawing or background color you are working in. You can also change the size of the grip if you need.

Notice that there are two different colors for grips, one when they are unselected (but visible) and another color when they are selected, or hot.

When you first start working with grip editing, I recommend that you use it for modifying your dimensions. This is an easy way to get started, as it won't affect your

geometry. Once you have mastered grip editing with dimensions, try editing your geometry and you'll find that in many cases this can be quicker and easier than using the regular AutoCAD commands.

When you first select a grip and make it hot, it turns red and you are able to stretch it to where you need it to go. Always read your command line when grip editing.

Command: <select an object> \*\* STRETCH \*\* <Stretch to point>/Base point/Copy/Undo/eXit:

You'll see that there are few options, ignore these for now and try the stretch option. You can now move that point anywhere and snap to any Osnap that you have active. This can be a quick easy way to connect lines rather than extend them. Once you have made your modification, press the escape key to deselect the grips. (Pressing the enter key will repeat the last command, which you don't want to do). You may have to press escape a few times to totally deselect the object.

The stretch option is only the first of several commands that you have available for grip editing. By pressing the **space bar**, you will be able to cycle through all available options for grip editing.

Command: <Pick and Object and Select a Grip>
\*\* STRETCH \*\*
Specify stretch point or [Base point/Copy/Undo/eXit]: <space>
\*\* MOVE \*\*
Specify move point or [Base point/Copy/Undo/eXit]:<space>
\*\* ROTATE \*\*
Specify rotation angle or [Base
point/Copy/Undo/Reference/eXit]: <space>
\*\* SCALE \*\*
Specify scale factor or [Base
point/Copy/Undo/Reference/eXit]: <space>
\*\* MIRROR \*\*
Specify second point or [Base point/Copy/Undo/eXit]:

The previous lines of text show what the command line would look like as you cycle through the options.

# To use any of these options, you should be very comfortable using the regular commands first.

You'll notice that these other options work a bit differently than the stretch option. If you have many objects selected, you can stretch one point, but can move all the objects highlighted with grip editing. This applies to the rotate, scale and mirror options as well.

Different objects have different grips available to them. A line has 3 grips: 2 endpoints and a midpoint. You can stretch the endpoints, but if you 'stretch' the midpoint, you move the line. You can stretch quadrants of a circle to change the radius, but stretching the center grip will move the circle. Single line text has one or two grips depending on the justification used. You can use their grips to align the text.

You can also use grips to adjust points in 3D objects. Just like a 2D object, select the solid with no commands are currently running and the grips will appear. You can then stretch grips to new points, giving you more power when editing solids.

In the example below the Center grip was used to shorten the 3D solid.



### WORKING WITH NODES AND POINTS

One last area to cover is the function and display of points, or nodes. If you have used the divide command, you have worked with the Node Osnaps. First make sure your **Node Osnap** is turned on. If you haven't, just draw a line and then enter the **DIVIDE** command.

Command: DIV DIVIDE Select object to divide: <Select the line> Enter the number of segments or [Block]: 3

With default Point Style settings, it will look like nothing has changed. But if you start the LINE command again, you'll see that 2 more Osnaps have been added.



If you divide a line and add 2 nodes, it's fairly easy to find then. If you divide a line into 20 segments, you'll start having some trouble locating them. Lucky for you, AutoCAD has a way of changing the display of your node Osnap. To do this, use the DDPTYPE command.

A Point Style		
Point <u>S</u> ize: 5.0000 %		
<ul> <li>Set Size <u>R</u>elative to Screen</li> <li>Set Size in <u>A</u>bsolute Units</li> <li>OK Cancel <u>H</u>elp</li> </ul>		

This is an easy dialog box to figure out. The first section allows you to select what you want your point to look like. I usually select the fourth one in second row as it is very easy to see, but you can select whichever one you want. Try a few out.



The next section defines what size the point will be.

Leave the value of 5 for now and look and the radio button options. The default, "Set Size Relative to Screen" will make the point appear 5% of your display. This makes it very easy to find in any drawing. The drawback is that if you have drawing with a lot points, the screen gets very cluttered. Use this setting in smaller drawings, or if you have few points. If you choose this setting, zoom out and then refresh the screen (**RE**) to see how they stay the same size.

The second choice is "Set Size in Absolute Units". This keeps the points at the same size consistently. If you zoom and refresh, the point size stays relative to your other drawing objects and not the screen. Use this if you have a lot of nodes or points in your drawing.

The divide command is very handy, and I recommend that once you have divided the object and made your edits, erase the nodes. Having extra nodes on a line can confuse you (or anyone else working on the drawing) at a later point.

**Conclusion:** In review, if you can master grips, you will be able to be much quicker at editing your drawing. I use grip edits a LOT in my drafting and it saves me a ton of time. Practice this technique until it becomes second nature to you and you'll be happier for the experience.

### Video: Grips & Divide / Nodes

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#### Take the Lesson 4-5 Quiz

Which objects don't have grips?

All objects have grips

**Blocks** 

**3D Objects** 



#### For Further Thought:

I love working with grips. They are such as time saver. I can't imagine using AutoCAD without them. These are in the top drawer of my tool box. When you are still learning AutoCAD, you need to be very careful with grips. Just like using a power tool like a drill press, you should use caution when using powerful CAD tools like grips. They're a quick way of editing objects, and in the wrong hands can cause trouble if used incorrectly.

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# **CONTINUE TO THE NEXT LESSON: 4-6**



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Last Update: 02/25/2019 18:26:11

### Topics covered in this Lesson:

#### Distance | Area | Mass Properties | List | ID

AutoCAD has many tools to help you as a designer. Some of these you may have already used. The properties command gives you a lot of information about an object you have selected and allows you change most of them. AutoCAD also has some commands used specifically for pulling information from objects.

Most of these commands can be accessed from the Home > Utilities tool panel. Here is a list of those commands (click on the icon for more information) :

COMMAND	KEYBOARD	ICON	DESCRIPTION
Distance	DISTANCE / DI	<b>└───</b> I Distance	Distance measures the distance and angle between two points
Radius	MEASUREGEOM	Radius	Measures the radius of an arc or a circle.
Angle	MEASUREGEOM	Angle	Measures the angle between two lines.
Area	AREA	Area	Calculates the area and perimeter of objects or of defined areas.
Volume	MEA / MEASUREGEOM	Volume	Calculates the area and perimeter of objects or of defined areas.
Mass Properties	MAS / VOL MASSPROP	None	Mass Properties calculates and displays the mass properties of regions or solids.
List	LI	None	Displays database information for selected objects.
ID	ID	🕒 ID Point	ID Point displays the coordinate values of a location.

**DISTANCE** (**DI**) is quite straight forward. It allows you to select two points on the screen and AutoCAD will tell you the total distance as well as the changes in X,Y & Z. This is a common 2D measuring tool. You can use cut and paste to take the results from the Text Window and put them into another document if needed.

```
Command: DI DIST
Specify first point: <SELECT ANY POINT>
Specify second point or [Multiple points]: <SELECT ANY POINT>
Distance = 12.5107, Angle in XY Plane = 25, Angle from XY
Plane = 0
Delta X = 11.3228, Delta Y = 5.3208, Delta Z = 0.0000
```

**MEASUREGEOM** is used to for a few options, among them Radius which will measure the radius of a circle or arc. Enter the command, right-click and then choose radius from the list on the screen. You can also use the command line options shown here:

```
Command: MEASUREGEOM
```





If you have Dynamic Input turned on, it will look a little different, but work the same way. With Radius selected, just pick on any arc or circle to have the radius displayed on the drawing screen and on the command line.



AutoCAD is kind enough to give you both the radius and the diameter on the screen - and on the command line:

Command: measuregeom

```
Enter an option [Distance/Radius/Angle/ARea/Volume]
<Distance>: R
```

Select arc or circle: <SELECT THE OBJECT> Radius = 3.3780 Diameter = 6.7559

You are left in the command and can select another option like Angle. Now you're asked to select 2 lines and the screen will display the angle between them.



**Volume** measurements can be used on 3D objects or 2D objects (if you enter a height). You can either select and object or pick points. Object is easiest for any single object, but if you are trying to measure a void, or other space not defined by objects, you can pick points. Note that this option is NOT the same as the **VOLUME** command.


**AREA** is a command that can be very useful. This can used to calculate square footage for floors, property lots, or even something like sheet metal parts. To use this command, you can type in **AREA** and see this on your command line:

```
Command: AREA

Specify first corner point or [Object/Add area/Subtract area]

<Object>: <pick 1st corner>

Specify next point or [Arc/Length/Undo]: <pick 2nd corner>

Specify next point or [Arc/Length/Undo]: <pick 3rd corner>

Specify next point or [Arc/Length/Undo/Total] <Total>: <pick

4th corner>

Specify next point or [Arc/Length/Undo/Total] <Total>: <enter>

Area = 1114.38 square in. (7.7388 square ft.), Perimeter =

11'-1 15/16"
```



The default option is to pick a point on the screen. This allows you to select the four corners of a room to find the total area in square drawing units (usually inches). You can pick as many points as you need and then press enter when you're done. AutoCAD returns the area and the perimeter length. If you select the next option (object), you can select a circle, rectangle, polyline or some other closed object. You also have the option of adding to or subtracting from the initial area. You may need to find the carpeting area of a room, but exclude where you have tile. Or you may need to add several rooms together.

**MASS PROPERTIES** is an option for finding information about 3D solid objects. You can find out the total volume and from there calculate the weight based on volume. You can also use the **VOLUME** (VOL) command to achieve the same results. AutoCAD also figures out some important engineering figures for you such as the center of gravity for an object. This listing shows you all the information you can get from **MASSPROP**:

Command: MASSPROP Select objects: 1 found Select objects: ----- SOLIDS ------Mass: 26693.3540 lb Volume: 26693.3540 cu in Bounding box: X: 449.5643 -- 480.3642 in Y: 172.2430 -- 208.4244 in Z: 0.0000 -- 23.9535 in Centroid: X: 464.9643 in Y: 190.3337 in Z: 11.9768 in Moments of inertia: X: 975034871.0602 lb sq in Y: 5778098589.0930 lb sq in Z: 6742922856.4689 lb sq in Products of inertia: XY: -2362317788.2337 lb sq in YZ: -60849711.7117 lb sq in ZX: -148649167.9674 lb sq in Radii of gyration: X: 191.1210 in Y: 465.2548 in Z: 502.6000 in Principal moments (lb sq in) and X-Y-Z directions about centroid: I: 4188330.0936 along [1.0000 0.0000 0.0000] J: 3386500.5727 along [0.0000 1.0000 0.0000] K: 5022179.7452 along [0.0000 0.0000 1.0000] Write analysis to a file? [Yes/No] <N>: N

This is a lot of information, so AutoCAD gives you the option of saving it to a file after it is displayed. It will be saved as an .MPR file, which can be opened in a text editor like Notepad.

**LIST** is an inquiry that gives you some basic information about a command. This is usually the information that AutoCAD needs to know about it to keep track of it in its database. Here is a listing of the same box that was selected for mass properties above:

Command: LI LIST Select objects: 1 found Select objects: 3DSOLID Layer: "DECK"

```
Space: Model space
Handle = 1469
History = None
Show History = No
Solid type = Box
Position, X = 38'-8 15/16" Y = 15'-10 5/16" Z = 0'-0"
Length: 2'-6 13/16"
Width: 3'-0 3/16"
Height: 1'-11 15/16"
Rotation: 0
```

Here are the properties (CTRL+1) for the same line:



Notice that you can get the same information, but also have the option of changing anything that you need to.

After List, the next option is ID POINT. This allows you to select anywhere on the screen and have AutoCAD return the X,Y&Z coordinates for that point. Here is a listing of ID POINT:



(You can also get this information by selecting a grip and looking at the status bar.)



#### **OTHER INQUIRY TOOLS:**

Time: Displays the date and time statistics of a drawing. Type **<u>TIME</u>**.

Status Displays drawing statistics, modes, and extents. Type **<u>STATUS</u>**.

Set Variable Lists or changes the values of system variables. Type **SETVAR**.

TIME and STATUS are two inquiries that give you information about the system, as opposed to objects.

Here is a listing of **TIME**:

Command: TIME Current time: Thursday, January 28, 2016 12:31:38:374 PM Times for this drawing: Created: Monday, May 17, 1999 10:04:22:260 AM Last updated: Monday, January 25, 2016 7:21:25:530 PM Total editing time: 0 days 22:24:16:169 Elapsed timer (on): 0 days 22:24:16:171 Next automatic save in: 0 days 00:03:49:794 Enter option [Display/ON/OFF/Reset]: <ENTER>

You can see the current date, when you last saved and when AutoCAD will do its next automatic save. And yes, that file has been with me for 8 years.

Here is a listing of **STATUS**:

Command: STATUS 1608 objects in Z:\we-rhere\cad\tutorials\level 2\links\cabin floor plan2.dwg Undo file size: 60024 bytes Model space limits are X: 0'-0" Y: 0'-0" (Off) X: 1'-0" Y: 0'-9" Model space uses X: -13'-11 3/4" Y: -5'-11 7/8" \*\*Over X: 40'-0 3/8" Y: 37'-9 15/16" \*\*Over Display shows X: -10'-3 15/16" Y: 14'-0 7/16" X: 7'-9 7/8" Y: 25'-0 1/8" Insertion base is X: 0'-0" Y: 0'-0" Z: 0'-0" Snap resolution is X: 0'-0 1/2" Y: 0'-0 1/2" Grid spacing is X: 0'-0 1/2" Y: 0'-0 1/2" Current space: Model space Current layout: Model Current layer: "DECK" Current color: BYLAYER -- 17 Current linetype: BYLAYER -- "CONTINUOUS" Current material: BYLAYER -- "Global" Current lineweight: BYLAYER Current elevation: 0'-0" thickness: 0'-0" Fill on Grid off Ortho off Qtext off Snap off Tablet off Object snap modes: Center, Endpoint, Gcenter, Intersection, Midpoint, Node, Quadrant Free dwg disk (Z:) space: 225265.5 MBytes Free temp disk (C:) space: 27095.2 MBytes Free physical memory: 1667.9 Mbytes (out of 8191.2M). Free swap file space: 8758.6 Mbytes (out of 17380.7M).

As you can see, there is a lot of information here, not only about your drawing, but about the computer you are working on.

The last inquiry, SET VARIABLE (**SETVAR**), allows you to change any system variables, or if you like AutoCAD will list them for you. The details of this command are not covered in this lesson. You must be VERY careful when working with system variables. A brief overview is given in **Lesson 4-8**.

### Video: AutoCAD Inquiry Tools

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### Take the Lesson 4-6 Quiz

Which objects don't have grips?

All objects have grips

Blocks

**3D Objects** 



### For Further Thought:

These AutoCAD tools won't help your drawing skills, but they are great for checking your work and ensuring accuracy in your drawing. Within this set of tools, the Distance tool is the one you might be using the most. Many Civil drafters will use the ID to locate a specific point. Mechanical drafters may use the Massprop tool to work with engineering design. Each tool has a specific use, and you might only need one or two, but there are times when you need to check your work and these tools come in very handy.

They can also be used by people who aren't familiar with AutoCAD to measure distances in a drawing, or get areas for estimation work.

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## **CONTINUE TO THE NEXT LESSON: 4-7**



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Last Update: 02/25/2019 18:26:27

Topics covered in this Lesson:

#### XRefs (External Reference Files in AutoCAD)

An **XRef** is an '**external reference**' to another AutoCAD drawing file. One file can reference many other files and display them as if they were one. These are used in larger projects for many reasons:

They keep the file sizes down.

They allow many users to work on individual components of a project.

Every time an XRef is loaded, it is the most recent version of the drawing.

XRefs can be updated, added, or unattached from the main drawing at any time.

You can XRef drawings that they themselves XRef other drawings (nesting).

For this project you can (should) use these two files. One is a master drawing file of a simple hotel layout. The second file is a typical hotel room and will be referenced by the master file. This is great example of XRefs are used in the real world.

Download the two files below to use in this lesson:

XRef Tutorial Drawing Files				
Master Drawing Motel-Floor-Plan.dwg				
XRef File	Typical-Motel-Room.dwg			

When a team is designing the motel, one person can work on the room and possibly be changing door, wall locations. Another person could be doing a furniture plan (possibly as a nested XRef within the room file), and then one person can oversee the whole project and use the updates on the 'room' drawing to keep a live view of the overall project.

In these days of networks and the Internet, many projects are produced this way. People anywhere in the world can collaborate on a project in real time. Here is an overview of the master file, note that it has the lobby, hallway, reception area but no guest rooms:

The first thing to do is to **ATTACH** the X-ref. This means that you are linking another drawing to your current one. Do this by starting the **XFEF** (**XR**) command. When you start the command, you will see the palette shown below. To attach an XRef,click on the "Attach DWG" button on the top left. (Note that you can also XRef Image files and DWF's)



The "Select Reference File" window will open: select a file and press Open. When you do this, you will see another dialog box appear.

Attach External Reference	1 10 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1 1	
Name: typical-motel-room	▼ <u>B</u> rowse.	
Preview	Scale         Specify On-screen         X:       1.00         Y:       1.00         Z:       1.00         I:       0         I:       0 <th>Path type Full path  Rotation  Specify On-screen  Angle: 0</th>	Path type Full path  Rotation  Specify On-screen  Angle: 0
Reference Type <ul> <li>Attachment</li> <li>Qverlay</li> </ul> Locate using Geographic Data	Specify On-screen           ☆:         0.00           ☆:         0.00           ☆:         0.00	Block Unit Unit: Inches Factor: 1.0000
Sho <u>w</u> Details	ОК	Cancel <u>H</u> elp

Once you find the file, decide whether you want it to be an attachment or an overlay. An overlay will not include any of the nested X-ref's that the file may contain. Below that section are the insertion parameters. You have a choice of either defining them on-screen, or using standard defaults in the dialog box. Once you make you decisions, press OK.

Finish the Attachment by following the prompts on the command. You'll see that it is similar to inserting a block. Insert the room to the point shown below at the top of the red triangle:



If you still have the XRef Palette open (if you don't, type **xR** to open it), you should see this:

Fi	le Referen	es				IE 1
	Reference	Name		Status	Size	Туре
*	Motel-Floo	or-Plan*		Opened	78.2 KB	Curren
owe	typical-ho	tel-room		Loaded	82.2 KB	Attach
•		III				
D	etails					6 🖬
Re	ference	typical-hotel-roo	m			
		Loaded				
	ze	82.2 KB				
	pe	Attach				
Da		1/28/2016 4:32:56	PM			
		E:\we-R-here-FUI		ACKUP\cad-1	6\tutorials'	
	ved Path	E:\we-R-here-FUI	LL-B	ACKUP\cad-1	6\tutorials	\level_4\l

Now that you have an X-ref, there are more options for you if you right-click the filename of the XRef.

ATTACH – Attaches another X-ref.

**DETATCH** – Detaches the selected X-ref.

RELOAD – Updates the selected XRef - use this if the XRef was changed.

**UNLOAD** – Removes the X-ref, but retains the reference for future use.

**BIND** – Permanently attaches a loaded X-ref, so that it is part of the drawing.

To fill up the Motel, use the **MIRROR** command to create the extra rooms. Mirror along the top half, then mirror those to the bottom using the middle of the hallway door on the right as your mirror line.

Working with X-ref's is usually quite straightforward, but there are some things to keep in mind.

The insertion point is very important. Everyone on a project must be working on the same set of coordinates and insert points.

When you attach and X-ref, you insert that file's layers and blocks. AutoCAD creates new layer names based on the filename and layer name of the X-ref. (In AutoCAD R14, you have a limit of 31 characters for these names. If you exceed that number, your XRef will not load.)

Cur	rrent	layer: WALLS						Se	arch for layer	•	х н
									e	×	*
<b>&gt;&gt;</b>	S	Name		o	Fre	L	Color	Linetype	Lineweight	Trans.	
	_	0		•	×	<b>e</b>	white	Continuous	Default	0	
	_	BORDER		•	×.	<b>B</b>	white	Continuous	Default	0	
	_	DIMS		•	×.	<b>e</b>	red	Continuous	Default	0	
	-	DOORS		<b>†</b>	×	E.	<b>b</b> lue	Continuous	Default	0	
	-	TEXT		<b>†</b>	×.	<b>e</b>	yellow	Continuous	Default	0	
	-	typical-motel-room BORDER		1	- X	Ē	white	Continuous	Default	0	
	-	typical-motel-room DIMS		1	×.	E.	red	Continuous	Default	0	
	-	typical-motel-room DOORS		1	***	E.	blue	Continuous	Default	0	
	-	typical-motel-room FURN		1	× .	E.	102	Continuous	Default	0	~
	-	typical-motel-room hidden		1	<u> </u>	•	green	Continuous	Default	0	PROPERTIES MANAGER
	-	typical-motel-room PLUMB		<b>?</b>	×.	•	mag	Continuous	Default	0	MA
	-	typical-motel-room TEXT		<b>?</b>	×.	•	blue	Continuous	Default	0	MM
	-	typical-motel-room WALLS		<b>?</b>	×.	•	green	Continuous	Default	0	IES
	-	typical-motel-room/WINDOWS	S	<u></u>	×	<b>F</b>	blue	Continuous	Default	0	ERT
	$\checkmark$	WALLS		<b>?</b>	- <u>è</u>	6	🔄 green	Continuous	Default	0	OP
											R PR
<b>&gt;&gt;</b>	•									•	LAYER
	Ъ,										LA
	_ A	II: 15 layers displayed of 15 tota	T lay	/ers							Ð

When you print, AutoCAD will automatically reload the attached X-refs.

If a file is moved from a location that AutoCAD expects to find it, you can browse for a new path, without having to re-attach the XRef.

Recent versions have added more functionality to XRefs. You can edit them in the drawing (as opposed to having to open the original file). To do this, just select the XRef in the drawing and right-click.



As you can see, there are other options to Open the XRef, Clip it (crop it down) or open the XRef Manager.

One problem in the past was the if a co-worker changed an XRef that was attached to your drawing, you didn't know and could keep working on the wrong information. Later versions of AutoCAD alert you when an XRef in your drawing has been modified by someone else.



To check this out, make some changes to the furniture in the room drawing and **SAVE** the file. Open up or switch to the Motel-Floor-Plan drawing and you'll the message appear in the bottom right corner.

Click on the blue link to reload, or use the XRef palette (right-click on the file name and choose reload).



Keep in mind that XRefs are simple to use but can be **VERY** important and used in many major projects. Practice working with them by attaching some of your previous drawings.

#### Video: External References in AutoCAD



### Take the Lesson 4-7 Quiz

If an Xref in your drawing gets changed, what should you do?

Bind it			
Reload it			
Detach it			



For Further Thought:

I hope you can see the power of XRefs in AutoCAD. If you have only been using CAD for a short while, then you may not be able to envision large, complex projects that contain many drawings. Try to imagine how many drawings it might take for a 10 story building with electrical, plumbing, structural and more contained in a set of drawings. Each portion of the plan might have different specialists working on it while a project manager references these drawings into a main file that allows him to oversee the design and look for conflicts.

This is the power of AutoCAD. Communication between many designers to create an amazing project together.

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# RETURN TO THE TOP OF THE LESSON | RETURN TO THE TABLE OF CONTENTS

## **CONTINUE TO THE NEXT LESSON: 4-8**



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Last Update: 02/25/2019 18:26:44

#### Topics covered in this Lesson:

#### **AutoCAD System Variables**

AutoCAD is a complex program and one of its major selling points is that it is highly customizable. Every user will use AutoCAD differently. Some use icons, some use command line prompts. Each drawing is also different. You may have different text heights, or dimension styles from one drawing to the next. To keep track of these different parameters, AutoCAD uses a large number of system variables to do this. You have recently used one system variable: LTSCALE. This could have been changed in the Linetype dialog box, but typing LTS is easier and quicker.

A system variable is a configuration setting that can be changed by the user. You just type in the name of the variable you want to change and enter the new value.

Command: TEXTSIZE

Enter new value for TEXTSIZE <0'-6">: 10 <ENTER>

Some system variables can not be changed, these are known as **read-only** variables. Here is an example of each:

SYSTEM VARIABLE	CURRENT VALUE	TYPE
TEXTSIZE	0.200	USER CONFIGURABLE
ACADVER	20.1s (LMS Tech)	READ ONLY

Most of AutoCAD's system variables are changeable. This allows for quick customization as well as easier programming.

Once you have used AutoCAD for a while, you may find that it is easier to redefine a system variable than to go through a dialog box to get the same result. In some cases, the only way to make these changes is through the system variable itself. Unfortunately, there are no shortcuts for entering these, you have to either remember them, or have a list nearby. Some of them are quite straightforward as to what they do, but some are rather cryptic and make it difficult for memorization.

If you start to learn programming for AutoCAD (LISP, VBA, C++), you will likely use system variables a lot to have your program control AutoCAD.

## Warning: Don't change system variables unless you know exactly what you're doing.

To see a list of all of the AutoCAD system variables, follow these steps:

Command: setvar <ENTER> Variable name or ?: ?<ENTER> Variable(s) to list <\*>: <ENTER>

It shows you what the settings were for a particular drawing at a specific time. The list also shows you which system variables are read only. For a listing of these in AutoCAD 2016, <u>click here</u>.

There is no video for this tutorial.



#### Take the Lesson 4-8 Quiz

Can you set system variables yourself?



#### For Further Thought:

This might seem like a tutorial you want to skip over, but it is very important to understand the concept of system variables in AutoCAD. If you are having problems with AutoCAD, a lot times it can be fixed by changing a system variable. You might not know which one, but after Googling for a solution, you need to know to change these - or at the very least, know that they exist and may be the solution you're looking for. Do you like what you just learned? Support this site with a Donation via Paypal:



## **CONTINUE TO THE NEXT LESSON: 4-9**



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Last Update: 02/25/2019 18:27:01

Topics covered in this Lesson:

#### Creating Tables in AutoCAD

Tables are an AutoCAD object that is can be created and customized by the user. Tables can be used for parts lists, revision history and any time you need to organize text and data. If you have used a spreadsheet before, you are already familiar with tables.

COMMAND	KEYBOARD	ICON	LOCATION	DESCRIPTION
Table	TABLE	Table	Annotation > Tables	Creates and inserts a new table.

Before starting this tutorial, open a drawing that has three text styles or start a new drawing and define them as shown in <u>Tutorial 2-6</u>. We'll need one style each for Title, Header and Data (name your text styles for easier reference later).

Start the command **TABLE** and you will see this dialog box:

🗼 Insert Table	
Table style	Insertion behavior
Insert options Start from empty table From a data link No data links found	Column & row settings <u>C</u> olumns: Column width: 5 2 1/2*
<ul> <li>From object data in the drawing (Data Extraction)</li> <li>Preview</li> </ul>	Data rows: Row height: 1 Line(s)
TitleHeaderHeaderHeaderData	Set cell style: First row cell style: Second row cell style: All other row cell styles: Data ▼
Data Data Data	OK Cancel <u>H</u> elp

The labels look pretty straightforward - you can define the number of rows, columns, row height, column width and press OK. *Or*.....you could press on the button shown in red above to open up the **Table Style** dialog box.

🔺 Table Style	23
Current Tablestyle: Standard	
Styles:	Preview of: Standard
Standard	Set Current New
Create New Table Style           New Style Name:           MY FIRST TABLE           Start With:           Standard	Continue Cancel Help
	Close Help

This allows you to create your own table style - sorry, just had to state the obvious. Press the "New..." button and give your new style a name and press "Continue".

A New Table Style: MY FIRST TABLE	
Starting table Select table to start from:	Cell styles
General 2 Table direction: Down -	General Text Borders Properties Fill color: None
Title Header Header Header	Alignment:     Middle Left       Format:     Text       Type:     Data
	Margins           Horizontal:         0'-0 1/16''           Vertical:         0'-0 1/16''
Image: second	Merge cells on row/column creation Cell style preview
	OK Cancel Help

Here, you can define the parameters (color, text height, borders, etc.) in your new custom table style. We're not going to delve very deep into this as it is something you can explore on your own. the concepts are quite easy to follow along.

First look at the menu shown above in red (1) and allows you select a section of the table: **Data, Header or Title** - and begin formatting that area.

The second section to look at is the tabs for **General**, **Text and Border**.

To fully format a table, you would potentially change all three tabs for all three sections. Make the following changes:



Text > Text Style	Use your lightest text	Make sure the text is easy to read, there could be a lot of it.
Borders > Lineweight	0.30mm	We'll give this a more visible border.
HEADER		
General > Fill Color	Yellow	Provides a background color to the header text.
Text > Text Style	Use your medium weight text	This can just be a 'bold' version of your data text.
Borders > Lineweight	0.40mm	We'll give this a thicker border than the one for the data.
TITLE		
General > Fill Color	Yellow	Provides a background color to the title text.
Text > Text Style	Use your heaviest weight text	This text style should stand out from the rest.
Borders > Lineweight	0.40mm	We'll give this a thicker border than the one for the data.

This is just a demonstration of how to style your table, and when you have a style that you like, you can save it as part of your template file. Having a consistent look throughout your drawings helps the people viewing your drawing and gives the drawing a more professional appearance.

When you are done formatting, press OK to close the dialog box and you will see your new style added to the list on the left in the Table Style dialog box (which you can now close). If you want to see how your table looks, you can press OK in the Insert Table dialog box - just be sure that "Start from an empty table" is selected.

🛕 Insert Table				×
Table style		Insertion behavior		
<ul> <li>Insert options</li> <li><u>S</u>tart from en</li> </ul>	npty table		Column & row settings	
C From a data	From a data link     No data links found     T		<u>C</u> olumns:	Column wi <u>d</u> th:
From object of Preview	data in the drawing (l	Data E <u>x</u> traction)	Data <u>r</u> ows:	Row height:
	Title		Set cell styles	
Header	Header	Header	First row cell style:	Title 🔻
Data	Data	Data	Second row cell style:	Header 🔻
Data	Data	Data	All other row cell styles:	Data 🔻
Data	Data	Data		
Data	Data	Data		
Pata	Data	Data		
Data	Data	Data		
Pata	Data	Data		
Data	Data	Data		
			OK	Cancel <u>H</u> elp

So now that you have defined your table style, you can insert one into the drawing and enter data using the Mtext editor and tabbing through the cells. Anyone who has used a simple spreadsheet will be able to use this. One major drawback is that there is no formula function in the table.

	-	A201155-1_2	2ND FLOOR	CORRIDOF	REVISED_F	INAL_03191	1.dwg	-		_ 0	X
H	ome Ins	ert Annot	ate Parar	netric V	iew Mar	iage Out	put Ad	d-ins	A360	» •	-
	<b>+</b>	A	<i>5</i>			₿	<b>⊬</b> +	Ê			
on Draw of	Modify	Annotati	Layers	Block	Properties	Groups	Utilities	Clipbo	ard	View	
Start		A20115	5-1_2ND FL	OSED_FI	VAL_031911	• × (+					
										-	đΧ
	TILE (D)										
HEADER 1		HEADER	2	HEADE	ta 👘	HEAD	er 4 👘	ŀ	IEADE	R 5	8
DATA		DATA		DATA		DATA		C	ATA		
DATA		DATA		DATA		DATA	DATA		DATA		
× Speci	Enorify apparite company										
🔩 🖭 - Type a command											
Model 0.Cover 1.FP 2.RCP 3.Above + 🎽 📶 🕇 📰 🕇 🕄 🕇 🖄 🔹 🕼 🔹											
	<u>%</u>	1/8" =	1'-0" / 1.04	% - 🌣	- + 1	Architect	ural 🔻 🗉	⊑r ∙	₽ (	2 😓	₽≡

You can always go back and edit your table style at any time.

Once the table is in the drawing, you can double-click in any cell to edit it or right click (below) it for more options:

	*		e c	D	E
1			TITLE	-	
_	HEADER 1	HEADER 2	HEADER 3	HEADER 4	HEADER 5
3	DATA	ATA	Пата	DATA	DATA
data dat		DAT	Cut Copy Paste Recent Input Cell Style Background Fill Alignment Borders Locking Data Format	•	DATA
			Match Cell Remove All Property Overric Data Link	des	
			Insert Edit Text Manage Content Delete Content Delete All Contents	ŀ	
			Columns Rows	<b>b</b>	
			Merge Unmerge	Þ	
		Ē	Properties Quick Properties		

What once used to be a very tedious task in AutoCAD has become easy with this new tool. You also have the ability to copy styles from one table to another to add consistency to your drawing. Practice creating a table or two.

In case you missed it, you can also extract your data from attributes to create a table that is dynamically linked to your drawing's blocks. This tutorial is in **Lesson 2-4**.

**Extra Practice:** Use <u>this drawing</u> (which contains blocks with attributes) and create a new table style and extract the data from the blocks as shown Lesson 2-4 and create a new table.

### Video: Formatting Tables in AutoCAD



#### Take the Lesson 4-9 Quiz

Can a table be linked to objects in your drawing?

Yes No

**Only Blocks** 



### For Further Thought:

Here's another lesson that is designed to help you create more professional drawings. Remember that your goal as a drafter is communication, so making a table that is easy to read will be important. If you work for a company, they may have specific standards for layout. but if you need to create a new style make sure it is clear and easy to read.

In most cases, simple will work. Remember that your table may be printed in black and white, so you have to consider different types of output and at different sizes. Do you like what you just learned? Support this site with a Donation via Paypal:



## <u>CONTINUE TO THE NEXT LESSON: 4-</u> <u>10</u>



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### Topics covered in this Lesson:

#### Creating Sheet Sets in AutoCAD | Publishing a Sheet Set

#### Concept:

Sheet sets are the digital version of the rolled up groups of drawings shoved in the back corner behind you. Actually they are the next progression from old Pack 'n' Go (now **ETRANSMIT**). This time though, you are just sending the layouts and not your actual CAD file. There are a lot of variables involved when you create these, but this tutorial will give you an idea of what's involved. Sheet sets can be published as DWF's or opened up within AutoCAD (.dst extension).

COMMAND	KEYBOARD	ICON	LOCATION	DESCRIPTION
Sheetset	SHEETSET	Sheet Set Manager	View > Palettes > Sheet Set Manager	Opens the Sheet set Manager

#### **Procedure:**

The first thing you would do is check that you have all your 'ducks in a row' concerning the drawings and layouts that you plan to publish. It sounds obvious, but you don't want to keep creating Sheet Sets after you find the layouts are scaled wrong, duplicated or whatever.

In this tutorial, I'll be using some of the example drawings that come with a standard AutoCAD installation. This is easier than creating a complex set of drawings and layouts just for this exercise. To follow along, make sure you have this folder available (or use a similar location for your version of AutoCAD):

#### C:\Program Files\Autodesk\AutoCAD 2016\Sample\Sheet Sets\Architectural

Start the command, **SHEETSET**, or use the **View > Sheet Set Manager** ribbon option to invoke the command (for those that like icons, it looks like the rolled up sheets

shoved in the corner behind you. The Sheet Set Manager palette will open as shown below - select the New Sheetset option from the top drop menu.



After that , you'll see what could possible be the largest dialog box you've ever seen with so little info:

Create Sheet Set - Begin		×
<ul> <li>Begin</li> <li>Sheet Set Details</li> <li>Choose Layouts</li> <li>Confirm</li> </ul>	Create a sheet set using An example sheet set Susting drawings This option lets you specify one or more folders that contain drawing files. The layouts from these drawings can be imported into the sheet set automatically.	
	< <u>B</u> ack Ca	ancel

Press "Next" and get taken to Step 2.

Create Sheet Set - Sheet Set Details						
Begin Sheet Set Details Choose Layouts Confirm	1 2	Name of new sheet set:         My First Sheet Set         Description (optional):         Practice, practice, practice				
	3	Sheet Set Properties				
		< <u>B</u> ack <u>N</u> ext > Cancel				

This seems fairly simple, give your set a (1) name, a (2) description and tell AutoCAD (3) where to save the file. There is a button for "Sheet Set Properties" that gives you some more options on the set (below) - you can even create your own custom fields here (or make changes to what you just entered).



For the sake of simplicity, this lesson will not cover the 'Sheet Set Properties', but it is straightforward to create your own fields.

The next tab is where you can select the "folders" where your drawings are located and add them to the Set. In your AutoCAD installation folder, select the sample folder shown below.



You now can turn on or off the layouts as you need to them. In this example I have excluded the section layouts.


The last step is to confirm what you just did in the final tab of the wizard.

Create Sheet Set - Confirm		X
Begin Sheet Set Details Choose Layouts Confirm	Sheet Set Preview: My First Sheet Set 1 A-01 - MAIN AND SECOND FLOOR PLAN 2 A-02 - ELEVATIONS 3 A-03 - DOORS WINDOWS AND ROOMS 4 A-04 - REFLECTED CEILING PLANS 5 AS-01 - ARCH SITE PLAN 6 Building Section - Layout1 7 Exterior Elevations - Layout1 9 Pile Detail - Layout1 9 Pile Detail - Layout1 9 Pile Detail - Layout1 11 Section 2 - Layout1 12 Sections - Layout1 13 STAIR - Layout1 13 STAIR - Layout1 14 STAIR1 - Layout1 15 STAIR2 - Layout1 16 Structural Base 1 - Layout1 17 Structural Base 2 - Layout1 18 Wall Base - Layout1 19 S-01 - FOUNDATION PLAN	
		< Back Finish Cancel

You see a list of layouts that will be part of this Sheet Set - if it's what you want, press "Finish".

After finishing the wizard, you are back in AutoCAD and ready to work with the Sheet Set. Any changes you make now are automatically saved to the .DST file in the location you selected at the beginning.

If you double-click on a sheet name, it opens up the drawing file in the selected layout tab.

Below, I have done a 'click and drag' to bring the "Title Sheet" layout to the top. As I mouse over the name of that layout, I get a preview image as well as some information about this layout.



The Sheet View list (second tab down) allows you to zoom directly to any saved view that was part of the layout (a new, great use for **views**).



As with anything in AutoCAD, there is more to these, but this lesson is aimed at explaining the concepts behind the command.

### Publishing a Sheet Set

Now you that you have created a great Sheet Set you need to publish it. This can be to PDF files, to a printer or as DWFx files - you have options.

Make sure you are back at the "Sheet List" tab in the Sheet Set Manage palette.



In this example, we'll plot the Sheet Set to DWFx. Click on the Publish button (above in red) and select "Publish to DWFx". Select a location to save your DWF and choose "Select". You will likely get a message telling you that AutoCAD will publish the DWFx in the background so that you can continue working.

Once the publishing is complete, you can view the DWF in a viewer like <u>this one</u> <u>online</u> or any DWF viewers you have on your computer.

**Conclusion:** Sheet Sets are a great way to share your drawings. As your drawings get more complex, you may have more than one file connected with a project and to share all of the information easily, you can create a Sheet Set that you client requires. Once you have this, you can arrange and edit the layouts that you want to share and publish them in a format that is easy to view for someone without AutoCAD.

### Video: Creating Sheet Sets in AutoCAD

🕒 Share / Save 🖪 🈏 🖻 ...

Take the Lesson 4-10 Quiz

What do you send with a Sheet Set?

Your fonts

**DWF Files** 

Your AutoCAD files



### For Further Thought:

This is another tool that you might not think is important at this stage of your CAD career. Remember this level of tutorials is all about filling in the gaps of your AutoCAD knowledge. You may not use Sheet Sets for a few years, but you might be asked to compile a set someday. Having a basic knowledge of them will be very handy when the time comes.

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# <u>CONTINUE TO THE NEXT LESSON: 4-</u> <u>11</u>



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Last Update: 02/25/2019 18:27:31

### Topics covered in this Lesson:

Dimensions | Dimension Styles in AutoCAD | Dimscale | Layout Dimensions

Concept:

A drawing needs to contain certain information. Most drawings will show you what

the object is (as clearly as possible) but won't tell you everything unless you dimension it. Would you agree to buy a house just by looking at the floor plan if it wasn't dimensioned? Would manufacturers know how to build your product if you didn't dimension it? You can see how important it is that a drawing used to convey information must have clear, accurate dimensions.

In previous lessons, you have done some dimensioning (first was **Lesson 1-8**). To continue with this lesson, you should at least have a good concept of basic dimensioning.

There are many ways to dimension a drawing, each drafting discipline has it own set of "rules". For example, a drawing of a gear would use a different style of dimensioning than that of a subdivision. My background is in mechanical drafting, so I use that discipline as an example of how to work with setting your dimensions to work with your drawing.

This tutorial will show you some of the steps involved to create and use a custom dimension style.

Below is an example of how a drawing can be dimensioned:



As you can see, on a simple drawing like this one, only 8 dimensions are needed to convey all the information about the size and shape of the object. With a little more information you can have everything you need to build it. This is the default style you get with the AutoCAD template.

Below shows the same drawing, but with added tolerances and note about what the object is made out of.



The changes in the dimensions above were done easily using the **DDIM** command. This command opens a dialog box for changing the parameters of your dimensions. In this example, the text style was also changed.

The great thing about AutoCAD is that it is very versatile. In the case of dimensions, you can modify any component that is part of it. Below are the names of various parts of a dimension:



Also, you can modify the dimension text dramatically, here are some examples:



### The Dimension Style Manager

So now that you have seen what you can do - how do you do it? All options are available in the **DDIM** (Dimension Style Manager) dialog box.

👗 Dimension Style Manager		×
Current dimension style: Standard <u>S</u> tyles: Annotative Standard	Preview of: Standard	Set Current          New         Modify         Qvenide         Compare
List: All styles	Description Standard	
	Close	<u>H</u> elp

On the left side of the dialog box is current style for working with (highlighted in blue).

In this lesson, you will create a new Dimension Style and use it in a drawing. From there, you should try different styles and get familiar with the options. Most companies these days will have a standard style (or set of styles) to use on drawings, but this is a very important tool to know if you want to turn out professional looking drawings.

Start but invoking the DDIM command and press the **NEW** button to open the small dialog for entering the name of the style you are creating. In this example I used the name "**DIMSTYLE 1**".

🛕 Dimension Style Manager		X
Current dimension style: Standard <u>S</u> tyles:	Preview of: Standard	
Annotative Standard		Set Current
	+ 1.0159 +	<u>N</u> ew
	1.1955	2 207 <u>Override</u>
Create New Dimension Style	Y	<u>Compare</u>
New Style Name: DIMSTYLE 1	Continue	
Start With:	Cancel	
Standard  Annotative	Help	Close Help
Use for:		
All dimensions	•	

Make sure that "**Start with**" has "**Standard**" as its setting. Press the Continue Button when everything is set. This will open the dialog box and you can start making modifications to your style.

Go to the second tab "Symbols and Arrows" and change the Arrowheads as shown below.

New Dimension Style: DIMSTYLE 1	
Lines Symbols and Arrows Fext Fit Primary	/ Units Alternate Units Tolerances
Arrowneads   First:   Closed blank   Closed blank   Leader:   Closed blank   Cosed blank   Cosed blank   Arrow size:   0.1800   0.1800   None   None   None   None   Mark   Dimension Break   Break size:   0.1250	Image: symbol   Image: symbol
	OK Cancel Help

As a general rule, I recommend not to change much on this tab - especially when you're still learning. Set as defaults, the dimensions are sized proportionately, if you change the setting on one parameter, your dimension can look 'off-balance'. For example, you could end up with huge arrows and small text. For the purposes of this lesson, the only thing that will change on this tab is the arrowheads.

Go to the third tab "Text".

Start up the Text Styles dialog box by pressing the button with the ... next to the text name. Create a new textstyle using **RomanS** and a width of 0.8 and call it "**DIMTEXT**". (For more info, see <u>Lesson 1-8</u>.) Close the Text Styles dialog box. Select DIMTEXT as your text for dimensions as shown.

🔺 New Dimension Style:	DIMSTYLE 1	x l
Lines Symbols and Arrow	ws Text Fit Primary Units Alternate	Units Tolerances
Text appearance		
Text style:	Standard 🚽	1.0159
Text color:	ByBlock 🔹 4	
Fill color:	None 1.1955	2 0207
🔺 Text Style	Contraction of the second s	
Current text style: DIMTEXT Styles: Annotative DIMTEXT Standard	Font Font Name: Tromans.shx Use Big Font Size Annotative Match text orientation to layout	Font Style: Regular New Delete Height 0.0000
All styles	▼ Effects	
AaBb1	Upside down Backwards Vertical	Width Factor: 1.0000 Oblique Angle: 0
		Apply Close Help

Apply (and close) the new Text Style to the new Dimension Style.

The great thing about this dialog box is that it shows you a preview of what your changes will do the final dimension in the top right window. In the bottom right, select **ISO** as the **Text Alignment** option. Try some other changes to see their effects in the preview, then end with the settings as shown below:

🔺 New Dimension Sty	le: DIMSTYLE	1		x
Lines Symbols and A	rrows Text	Fit Primary	Units Alternate Units Tolerances	
Text appearance				
Text style:	DIMTEXT	•	1.0159	
Text <u>c</u> olor:	ByBlock		992	
Fill color:	None None	-	+++++++++++++++++++++++++++++++++++++++	
Text height:		0.1800		
Fraction <u>h</u> eight scale:		1.0000	R. Soft	
Draw <u>f</u> rame around	text			
Text placement			Text alignment	
<u>V</u> ertical:	Centered	•	Horizontal	
Hori <u>z</u> ontal:	Centered	•	Aligned with dimension line	
View <u>D</u> irection:	Left-to-Right	•		
Offset from dim line:		0.0900	ISO standard	
				-
			OK Cancel <u>H</u> elp	

So far you haven't changed much, but you've seen the options available in just 2 tabs. Click on the next tab "Fit" to continue.

As a rule, I leave this tab alone. AutoCAD does a good job of placing and fitting dimension where I want them. If I don't agree, I usually just use **grips** to edit the placement.

Click on the next tab "Primary Units" to continue.

In the Primary Units, you find some of the more common parameters that need to be changed. **Precision** is very important. First off, you usually don't need to show 4 decimal places. If you do show 4 decimals places and send your drawing off to the

machinist, you are asking him to manufacture the part to within 1/10000 of a unit - which can be a very expensive mistake. 3 Decimal places is usually enough - or less for rougher jobs. Of course this depends on your units. If you are dimensioning in millimeters, then you could get away with full units. If your units are decimal miles, or kilometers, then you might need more zeros after the decimal.

Also, I have added a couple of other changes; adding trailing zeros and a suffix denoting the units. Note how angular dimensions have a separate section - make sure you don't neglect them.

👗 New Dimension St	yle: DIMSTYLE 1	
Lines Symbols and	Arrows Text Fit Primary U	Units Altemate Units Tolerances
Linear dimensions —		
<u>U</u> nit format:	Decimal 🔻	
Precision	0.0	
Fraction format:	Horizontal 👻	
Decimal separator:	∵ (Period) 🔹	
Round off:	0.0000	20°
Prefi <u>x</u> :		v
<u>S</u> uffix:	IN (or CM)	
Measurement scale		
Scale factor:	1.0000	
Apply to layout d		Angular dimensions
Zero suppression		Units form <u>a</u> t: Decimal Degrees 💌
Leading	<u> </u>	Precision: 0.0
Su <u>b</u> -units fac 100.0000		Precisi <u>o</u> n: 0.0 💌
	v 0 inches	Zero suppression
Sub-unit suff	X:	🔲 Lea <u>d</u> ing
		Trailing
		OK Cancel <u>H</u> elp

Make the changes you see above and check the preview after each change.

The next tab "Alternate Units" is used if you want to display two different units in your dimension. For example, you can draw your drawing in imperial inches, then dimension with inches as your primary units and add the alternate units behind as millimeters.



This is easy to do as AutoCAD assumes this is what you want as the default. The most important thing here is to make sure that your "Multiplier for alt units" is correct. In this case there are 25.4mm to the inch, so that is the multiplier used. Make the checkbox at the top is checked and add a suffix to indicate what the alternate unites represent. Check your preview and press OK.

A Modify Dimension Style: DIMSTYLE 1	
Lines Symbols and Arrows Text Fit Primary	Units Alternate Units Tolerances
Display alternate units     Alternate units	Та р р р р р р р р р р р р р
Unit format: Decimal	т У м (
Precision 0.00	
Multiplier for alt units:	as the state of th
Round distances to: 0.0000	a transmission of the second s
Pre <u>f</u> ix:	_
Suffi <u>x</u> : MM	
Zero suppression	Placement
Leading <u>T</u> railing	
Sub-units factor:	<u>A</u> fter primary value
V 0 inches	© <u>B</u> elow primary value
Sub-u <u>n</u> its suffix:	
	OK Cancel <u>H</u> elp

The final tab is for tolerances. If you are not familiar with tolerances, it simply means that there can be a minimum and maximum value used to manufacture a part. For example, you might need a hole drilled that is 0.5" - but your part will still work if the hole is between 0.45" and 0.55". This is indicated by a tolerance dimension of • ±.05 (plus or minus .05"). Sometimes the tolerance will only in the plus limits as the hole has to meet a minimum requirement.

In this example, you will set the tolerances to be  $\pm .05$  units and display them at 80% of the scale of the primary units. Sound easy? It is. Look at the image below to see how this is done.

🔺 Modi	fy Dimension Sty	/le: DIMSTY	'LE 1				X
Lines	Symbols and Arro	ws Text	Fit	Primary Unit	Alternate Units	Tolerances	
Tolera	ance format			ſ	3	· _	
Metho	od:	Symmetrical		<b>-</b>		A A	
Precis	ion	0.0		-			10 NAMI B]
Upper	<u>v</u> alue:	[	0.5000				
Lo <u>w</u> er	value:		0.5000	A V	a wat the second		
Scalin	g for <u>h</u> eight:	[	0.8000		3		
Vertic	al po <u>s</u> ition:	Middle		•			
Toler	ance alignment				Alternate unit toler	ance	
	<u>A</u> lign decimal sepa Align operational s				Precisi <u>o</u> n:	10	•
		-			Zero suppression		
	suppression Leading	√ 0 <u>f</u> eet			Leading	√ 0 f <u>e</u>	et
	<u>T</u> railing	√ 0 inch	es		Traili <u>ng</u>	🗸 0 in	nches
					ОК	Cancel	<u>H</u> elp

With all of your new settings, here is what your dimensions will look like:

⊲— 36.0 IN±0.5 IN [914.40 MM±13 MM]—---

Once again - preview shows you how it will turn out. Click OK to close the dialog box. At the original Dimension Style Manger, press the name of your new style, then click the "Make Current" button. Close the dialog box to go back to your workspace.

Draw the wedge shape at the top of the less and dimension it using your new Dimension Style. It should look something like this:



Note that there is a change to the Alternate Units. In the example shown above, the Alt Units are placed below the Primary Units. Go back and edit your dimstyle to make this change.

### Using the DIMSCALE setting

If you have dimensions on your drawing that are either too large, or too small, use the DIMSCALE setting to scale them to better fit. If your DIMSCALE is currently set to "1", you can make the dimensions larger by entering larger numbers. For example, enter 12 will make your dims 12 times large than they were at "1". This does not enlarge the value of the dimension, just the size of the dimension. To make your dimensions smaller, you can also enter fractions or decimal numbers like .5.

These numbers are all based from the standard sizes set in your DDIM dialog box. For example, if you enter "12" to make them larger, entering 1/2 later will not make

them scale to "6". After changing the DIMSCALE, all new dimensions will use this scale.

Using DIMSCALE is important to make sure that people can easily read your drawings. Set your DIMSCALE before adding all of your dimensions.

After changing the scale, you will not see a change. You will need to update the dimensions. Follow these steps:

Pre-2016	2016 & Newer
Command: DIMSCALE	
Enter new value for DIMSCALE <1.0000>: 12 <enter></enter>	Command: DIMSCALE
Command: DIM	Enter new value for DIMSCALE <1.0000>: 12 <enter></enter>
Dim: UPDATE	
Select objects: ALL 2 found	Command: -DIMSTYLE Current dimension style:
Select objects: <enter></enter>	DIMSTYLE 1 Annotative: No Current dimension
Press <esc> twice to exit the Dim prompt.</esc>	overrides: DIMSCALE 12.0000 Enter a dimension style option [ANnotative/Save/Restore/
	STatus/Variables/Apply/?] <restore>: A <enter></enter></restore>
	Select objects: all 1 found
	Select objects: <enter></enter>

## **Dimensioning Tips**

Now you have the basic understanding of how dimensions work, you can experiment and work with other styles.

Here are some common rules about dimensioning:

- 1. Keep them on a separate layer.
- 2. Dimension towards the end of your project or you could be moving them as you add more objects.
- 3. Use your Osnaps and confirm what you are snapping to.
- 4. Assign them a unique colour in the drawing, and use that colour in all of your drawings to make it easier to identify dimensions (especially when zoomed in close)
- 5. After you create your first dimension, AutoCAD creates a new layer called "DEFPOINTS". This is used only for the small points you see at the end of an extension line. This layer will not print so be sure not to draw on it.
- 6. Keep even spacing between you dimensions and between your dimension and your object lines. Use DDE while placing dims.
- 7. Try not over lap lines (either object or dimension).
- 8. Use enough dimensions to make sure that all measurements are there one dimension left out can hold up a project.
- 9. Try to keep dimensions outside of the objects you are dimensioning.
- 10. For clarity, don't over-dimension.
- 11. If you need to override one dimension (perhaps a unique suffix), you can change it in the properties (select > right click > properties)

### Dimensioning in Layouts

If you are familiar with Layouts (or Paper Space) as shown in <u>Lesson 2-4</u>, you can continue with this lesson which will explain dimensioning in Layouts.

Using the same wedge shape from above, delete your dimensions and then scale the wedge up by 24X.

Go into your Layout and use a standard scale for your viewport. Start with a DIMSCALE of 1.

Start dimensioning your lines. AutoCAD now does a great job of dimensioning in Layouts. It will recognize the scale of the viewport and find the correct size of the object you are dimensioning.

I recommend doing all of your dimensioning in you Layouts. There are a few reasons for this:

It leaves your model space free of dimensions

For 3D objects, this system is MUCH easier.

You only need one size of dimensions for full view and details.

You can add your title block, notes and dimensions in one area.

I hope that this lesson has given you the ability to create professional, **accurate** dimensions. Like everything else in AutoCAD, practice make perfect.

For more practice, create a simple floor plan and create a dimension style using Architectural units and 'ticks' instead of arrows. Here's a very simple Architectural drawing using this style. Add Alternate units later and then update the dimensions using the -DIMSTYLE command. After you have dimensioned this small drawing, erase your dimensions from model space and add dimensions in a layout.



Video: Creating a Custom Dimension Style

🕒 Share / Save 🖪 🈏 🖻 ...

Take the Lesson 4-11 Quiz

What command opens the Dimension Style Manager?

STDIM

DDIM

DIMSCALE



### For Further Thought:

You might be noticing a trend in this level. I'm not really showing you how to draft with AutoCAD anymore. Instead I am showing some tools that you need to be a professional. With a well rounded knowledge, you will have an advantage in job interviews over someone who just knows the basics of drafting.

Knowing how to make a professional drawing will get your CAD career started on the right foot. Spend time looking at final drawings for large projects if you can and study what goes into them.

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# RETURN TO THE TOP OF THE LESSON | RETURN TO THE TABLE OF CONTENTS

## <u>CONTINUE TO THE NEXT LESSON: 4-</u> <u>11a</u>



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Last Update: 02/25/2019 18:27:51

### Topics covered in this Lesson:

Dimension Types in AutoCAD

#### Concept:

This tutorial will explain the different types of dimensions which are commonly used in AutoCAD and help you understand their differences. A good AutoCAD drawing will convey the needed information clearly and accurately. Dimensioning is used to let the drawing reader know what size each length or curve is, without having to open the drawing and measure it. If your dimensions are inaccurate, you will not be working in CAD for very long, no matter how accurate your drawing is. Learn when each dimension is needed and how to use it.

If you need to modify the appearance of your dimension, please refer to the previous tutorial - <u>4-11</u>.

Dim Type

lcon

**Keystroke** 

**Description** 

<u>Linear</u>	DIMLINEAR / DIMLIN	<b>├</b>	Dimensions for straight lines either horizontal or vertical to the UCS.
Aligned	DIMALIGNED / DIMALI	×,	Dimensions for straight lines that are angled to the UCS and provides the line's length.
<u>Angular</u>	DIMANGLUAR / DIMANG	$\triangle$	Dimensions between 2 points and gives the resulting angle measurement.
<u>Arc</u> Length	DIMARC	Ċ	Dimensions the length of an arc or arc segment of a polyline.
<u>Radius</u>	DIMRADIUS / DIMRAD	ĸ	Dimensions the radius of an arc, circle or arc segment of a polyline.
<u>Diameter</u>	DIMDIAMETER / DIMDIA	$\bigcirc$	Dimensions the diameter of an arc, circle or arc segment of a polyline.
<u>Center</u> <u>Mark</u>	DIMCENTER	$( \cdot )$	Places a center mark at the center point of an arc or circle
Rotated	DIMROTATED	-	A modifier to a dimension to make it parallel to a selected line.
<u>Spacing</u>	DIMSPACE	H H H	Aligns the selected dimensions to an even spacing.
<u>Continue</u>	DIMCONTINUE / DIMCONT	H	Continues dimensioning from a base dimension
<u>Baseline</u>	DIMBASELINE / DIMBASE	Ì	Continues dimension from a base dimension to give an overall dim.

## Types of Dimensions in AutoCAD

You have most likely already used some of the common dimension types like 'linear' or 'radius'. This section will describe how each one works in more detail.

#### **Linear Dimensions:**

Linear dimensions are used to create a dimension that measures a line that is either horizontal or perpendicular to the UCS. This is the most commonly used dimension and is very easy to use. Below are some typical linear dimensions:



It's good practice (thought not always allowed by time) to have your dimensions at a consistent distance from the object. Make sure that it is not to close to overlap the object and not too far to confuse which object you dimensioned. To create a linear dimension, either use the icon or key in **DIMLINEAR**. Just pick the two endpoints of a line (or any two points) and then place the dimension line where you want it. Very quick, very simple.

#### Command: DIMLINEAR

Specify first extension line origin or <select object>:SELECT
FIRST POINT
Specify second extension line origin:SELECT SECOND POINT
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: SELECT
LOCATION OF DIM LINE
Dimension text = 5.00

Note that there are some options at the command line:

OPTION	DESCRIPTION
MText /	You don't need to use these, it doesn't really affect your dimensions

Text	
Angle	This will place your text at the angle you specify
Horizontal	This will create a horizontal dim, but you select this when you place your dimension line. Rarely needed.
Vertical	This will create a vertical dim, but you select this when you place your dimension line. Rarely needed.
Rotated	This will create a 'rotated' dimension. This is explained <b>below</b> .

You'll also notice on the first prompt, you can <select object>, but press <enter> at this point and then you can just select the line instead of the two endpoints. This is a lot quicker and more accurate as you don't have to worry about picking the wrong Endpoint Osnap.

Since this is a common command, it is worth creating an alias for it (see **<u>next</u>** lesson).

#### **Aligned Dimensions:**

The Aligned dimension works just like the linear dimension. The big difference is that it works with lines or distances that are not square to the UCS. You can also use this on any 2 points that are also compatible with the linear dimension. It will create a dimension parallel to the angled line.



Command: DIMALIGNED

```
Specify first extension line origin or <select object>:SELECT
FIRST POINT
Specify second extension line origin:SELECT SECOND POINT
Specify dimension line location or
[Mtext/Text/Angle]: SELECT LOCATION OF DIM LINE
Dimension text = 2.00
```

You'll see that command line prompts are almost the same. Of course, there are no options for horizontal or vertical placement. Also Rotate is missing to avoid confusion and errors. You can also select Angle for your text to match the angle of line (just select 2 points on the line you are dimensioning). You can also set your text to be aligned with the dimension line in your Dimension Style Manager.

#### Angular Dimensions:

Not every dimension gives the reader a length. Sometimes other measurements are need. Angular dimensions will display the angle between two objects. The measurement of the angle is defined by your dimstyle, but degrees is standard. The command works similar to the linear dimensions (see a pattern?).



Command: \_dimangular
Select arc, circle, line, or <specify vertex>: <select first
line>
Select second line: <select second line>
Specify dimension arc

The example above shows 3 common uses of this tool. To get the angle on the arc, just select anywhere on it. Getting the outside measurement (225° in this example) is done by using the prompt <select vertex>.

Command: DIMANGULAR Select arc, circle, line, or <specify vertex>: Specify angle vertex: <select point A>

Specify first angle endpoint: <select point B>

Specify second angle endpoint: <select point C>
Specify dimension arc line location or
[Mtext/Text/Angle/Quadrant]:<place dim>
Dimension text = 225



#### Arc Length:

This is a fairly new command in AutoCAD and will not be available on some older versions. This command works simply, just start the command and select an arc or arc segment of a polyline. In the example below, I have shown the Angular (180°) dimension with the Arc length so that you can see the difference. Notice the symbol in front of the 3.14.



```
Command: _dimarc
Select arc or polyline arc segment: <select the arc>
Specify arc length dimension location, or
[Mtext/Text/Angle/Partial/Leader]: 1.2
Dimension text = 3.14
```

Once again, you are offered some options when dimensioning. Mtext/Text/Angle are the same options for text that you saw earlier. Partial will allow you to dimension only part of the arc. This is useful for midpoints or nodes, when the arc is segmented. Leader will draw a leader from the dim text to the arc that it refers to.



#### **Radius Dimension:**

Another common and simple command. **DIMRAD** will dimension any arc or circle and give you the radius and place a center mark at the center of the selected object. A center mark is a cross the indicates the center point of an arc or circle.



```
Command: _dimradius
Select arc or circle: <select object>
Dimension text = 0.40
Specify dimension line location or [Mtext/Text/Angle]: <place
the dim>
```

Once again, you have some options for the text at the command prompt.

#### **Diameter Dimension:**

The diameter and radius dimensions work the same, but give a different measurement. As a rule, I will use the diameter dim on things like a hole to be drilled or for the dimensions of an overall part and radius for filleted corners, etc.



As you see, the diameter and radius dimensions add a center mark (this can be turned off, or changed to a larger line (in **DIIM**). I will usually dimension the circles first so that I can select the center mark and have a small gap (it looks cleaner).

```
Command: _dimdiameter
Select arc or circle: <select the object>
Dimension text = 1.00
Specify dimension line location or [Mtext/Text/Angle]: <place
the dim>
```

In the example image above, you can see that even a very simple drawing can have enough dimensions to clutter things up. Make sure that you have enough dimensions and don't duplicate or miss any. Make sure that they can be read from one or two angles when printed (usually from the bottom right corner).

#### **Center Mark:**

By now you should be getting the hang of how this works.

The **DIMCENTER** command adds a center mark to a circle or arc. How it is displayed is set in the **DDIM** dialog box under the "Symbols and Arrows" tab. If you select "None" you will not be able to place a center mark using this command. The icon for center marks is in the sub menu on the dimension panel.



Sometimes you might use both in the drawing. Below. on the left, I used the center mark that was made with the diameter dimension, then I dimensioned from it. On the right, I used a Line center mark and extended one of the lines to the other side to indicate the holes are aligned. Then I double-clicked on the diameter text and added a suffix to indicate that both holes are the same size.



But when you do this, any of the center marks made by the diameter or radius dimensions will change to the new style that you selected. It's best to start with one style and leave it. The size of your center mark is defined by the **DIMCEN** system variable.

### **Rotated Dimensions:**

A rotated dimension is not a dimension that was rotated using the rotate command. Instead it allows you to create a linear dimension to two points that are not in a straight line. Look at the image below and see the difference between an aligned dimension and a rotated one.



First, look the dimension on the far right. See how it gives a true overall size of 10 units? On the right, the aligned dimension is not the overall size due to the 94° angle in the bottom left corner. Also in the top right, notice the rotated dimension with 2.00 units. I need to rotate it as the 2 points that I selected were not on the same axis. It's a little confusing at first, but try it a few times.

```
Command: dimlin
DIMLINEAR
Specify first extension line origin or <select object>: <first
endpoint>
Specify second extension line origin: <first endpoint>
```
```
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: R <enter>
Specify angle of dimension line <0>: <select first point on
axis>
Specify second point: <select second point on axis>
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: <place dim>
Dimension text = 2.00
```

When it comes to selecting points on the axis, pick the common parallel line (in this example, I click 2 points on the 10 unit dimension. You can also created a rotated dim by using the **DIMROTATED** command. In this case, you select the angle first.

#### **Even Spacing of Dimensions:**

Sometimes you might have a bunch of dimensions in one area and want to space them evenly. The easy way to do this is to place them quickly, then use the Adjust Space command, **DIMSPACE**. Look at the image below with the random spacing. Then mouse over it to see how this commands gives them even spacing.



Draw a similar shape and try this. Place your dims, but allow for enough room to fit them all (just don't worry about the spacing). Then start the **DIMSPACE** command.

```
Command: DIMSPACE
Select base dimension: <I selected the 2.00 dim in this
example>
Select dimensions to space:
Specify opposite corner: 3 found <select the others>
```

#### Select dimensions to space: <enter> Enter value or [Auto] <Auto>: <enter>

Fast and easy. You have the option at the end to enter a distance between each dim, but Auto should work in most cases. Just be careful to select the "Base" dimension you want first - try selecting other dimensions as your 'base' to see what results you get.

#### **Continue Dimensions:**

Another shortcut is to use is "Continued Dimensions". This is used when you have a series of linear dimensions that will all be on one side of an object. With this command, you can select a 'base' dimension and then have AutoCAD help by continuing the command so that you can just click and endpoint and continue to the next, click an endpoint, etc. Have a look at the drawing below:



When you have a series of dimensions all in one plane like this, just create the first one with DIMLIN and then use the **DIMCONT** command to create the rest of them. In this example, I created the dim on the left first.

```
Command: _dimcontinue
Specify second extension line origin or [Select/Undo]
<Select>: <SELECT A>
Dimension text = 2.0000
Specify second extension line origin or [Select/Undo]
<Select>: <SELECT A>
```

```
Dimension text = 1.0000
Specify second extension line origin or [Select/Undo]
<Select>: <ENTER>
```

ng this command or the Baseline dimensions, select your points for the base dim in the direction that the others bove, since I selected the base dim on the left, I had to first pick the left point and then the right point. This sets t to follow.

After doing this, the next time you start the **DIMCONT** command, it will return to this point and continue on. Try this out in your own drawing with a simple shape like the one above.

#### **Baseline Dimensions**

This command is very similar in function to the continue command, but instead provides an overall dimension based upon the first point you select in your first (base) linear dimension.



```
Command: _dimbaseline
Specify second extension line origin or [Select/Undo]
<Select>: <SELECT A>
Dimension text = 3.0000
Specify second extension line origin or [Select/Undo]
<Select>: <SELECT B>
Dimension text = 4.0000
```

# Specify second extension line origin or [Select/Undo] <Select>: <ENTER>

Once again, I first needed to create a base dimension with the **DIMLIN** command. I again used the dim on the left (first point was on the left). Then I started the **DIMBASE** command and selected points A and B. Once again, try it out and see how this works. See what happens when you create your base dim from right to left.

#### Summary:

You have a lot of tools at your disposal when it comes to dimensions. The trick is to know when to use each one. It's also extremely important that you are accurate. Clicking on another dimension instead of a line's endpoint can be a huge mistake - especially in mechanical drafting where tolerances are small. Review your dimensions before sending a drawing off to someone else, or printing it- then check it again. Check for overlaps or other confusing areas. The commands themselves are easy to use, but accuracy is critical.

## Video: Types of Dimensions in AutoCAD

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### Take the Lesson 4-11a Quiz

How do you know if the dimension shown is for arc length?

You can't dimension an arc

There is an arc symbol in the prefix

It has the word Arc after it



## For Further Thought:

Hey look! It's another tutorial about communication! I can't stress how important it is that your AutoCAD drawings are clear and easy to read. Before you complete a drawing and pass it on to someone else, make sure you take time to review your dimensions for clarity, accuracy and completeness.

Spending a few minutes on this can save hours down the road if people have to come back to you to get more information that should have been there in the first place. Be a pro and do it right the first time.

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# <u>CONTINUE TO THE NEXT LESSON: 4-</u> <u>12</u>



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Last Update: 02/25/2019 18:28:10

# Topics covered in this Lesson:

#### Keyboard Shortcuts in AutoCAD | Customizing the Interface | Adding a Toolbar

#### Concept:

One of the great things about AutoCAD is that it can be easily customized to suit the individual user. By now, you have seen how you can change the osnaps for example, but you can change a lot more than that. This lesson will introduce you to some of the customization options you have.

## **Keyboard Shortcuts**

So far you have been using AutoCAD's default shortcuts. This section will show you how you can create your own to help your productivity. All shortcuts are stored in the acad.pgp file. This file is loaded into AutoCAD every time you start the program. It is now easy to edit thanks to an express tool names **ALIASEDIT**(you will need to have the express tools installed). If you don't have Express Tools installed, this command will not work - but you can still edit you acad.pgp file in a text editor.

Type in **ALIASEDIT** and you'll see a dialog box pop up. Press the **Add** button.

acad.pgp - AutoCAD Alias Editor					
File	Edit Help				
Co	mmand Aliases	Shell Commands			
	AutoCAD comma	ands entered from shortcut:			
	Alias	AutoCAD Command	Add		
	ЗA	3DARRAY	Remove		
New Comr	mand Alias		Edit		
the box Alias:	e alias and its Au es below.	utoCAD command			
MATCH MATCH MATCH	CELL PROP	Cancel	Apply		
	ITORCLOSE ITOROPEN	-			

In this example, I have created a shortcut for **MATCHPROP** which is usually **MA** (I find that two keys on opposite sides of the keyboard slow me down). Since **n** is not used for anything, I find that one letter is more than twice as fast as two. Type in what you see and press OK.

Now press **Apply** so that the changes take effect. You'll see this dialog box warning you that you are about to overwrite your acad.pgp file. Press yes, ONLY if you are sure you did the correct changes.

Overwrite	C:\Users\Me\appdata\roaming\autodesk\autocad 2016\r20.1\enu\su
C:\Users\Me\appdata\roaming\autodesk\autocad 2016\r20.1\enu\support\acad.pgp This configuration file already exists. Do you want to overwrite it?	
	Yes No Cancel

Press Yes and you will see a message pop up that you have saved your changes and that your current AutoCAD session has been updated. That means that you can now use the shortcut you just added.

If the shortcut command you select is already used by an existing command, then you can still overwrite that alias - but be aware that once changed, the old command will still work, but the shortcut will not.



In the example above, I entered **DL** to use as a shortcut for **DIMLINEAR**, but it is already in use for **DATALINK**. In this example, I would overwrite is as I use **DIMLINEAR** much more often than the **DATALINK** command. I can still type in DATALINK to use this command.

Think about what other commands you use often that aren't convenient to type in.

### CUSTOMIZING THE INTERFACE

#### **Quick Access Toolbar**

The quick access toolbar is the row of icons at the top of the screen. You'll find Save, Print and other common commands there. One that you won't find is Spell Check. This is one that *should* get used a lot, so I added it. It's easy and you can do it to. Use this method for adding any commands to this toolbar. Navigate the Ribbon to **Manage > Customization > User Interface**. You'll see this dialog box come up.

Customizations in All Files All Customization Files			
Quick Access Toolbar 1 Quick Access Toolbar 1 Open Save Save Plot			
New Open Save Save Plot	• •		
	-		
Save SaveAs Plot	-		
SaveAs	-		
Redo	=		
Billion Ribbon			
Toolbars			
Menus     Quick Properties			
Bhotcut Menus to position			
E Keyboard Shortcuts	Ŧ		
Command List:	\$		
spell	8		
All Commands Only			
Command Source			
Spell Check ACAD			
OK Cancel Apply Help	۲		

In the top left section, look for the Quick Access Toolbar 1 folder. Then type spell in the Command List box and you'll see the Spell Check command appear below. Drag the command up to it's desired position on the toolbar. Press Apply. Your toolbar should now look like this.



If you have read all the tutorials, you'll know that I don't recommend using icons. Still there are sometimes you may want to. Perhaps the command isn't used much, but you want easy access to it. Perhaps you can't create a shortcut for the command. There could be a few reasons, but here is how to create a new toolbar with the icons you want.

# Adding a New Toolbar

With the CUI dialog box open, right click on Toolbars and select New Toolbar.

🔺 Customize User Inter	face					
Customizations in All Files						
All Customization Files						
3D Basics						
Quick Access Toolbars     Right-Click Menu						
🗄 🕂 🗖 Sm:	New Toolbar					
i ⊕… E Sme	Rename					
⊕ Drav	Delete					
	<b>D</b>					
i ⊡ Inqu	Paste					
ter Lay	Find					
Here Moc	Replace					
⊕ Mod, ⊕ E Properties						
Command List:		\$				
Search command list		Q				
All Commands Only		- <sup>5</sup> à 😤				
Command		Source				
3 3 Point		ACAD				
3D Adjust Clip Planes		ACAD				
🚊 3D Adjust Distance		ACAD 👻				
	<u>O</u> K <u>C</u> ancel	Apply Help ()				

Once you have your toolbar added to the list, drag a command up to it. I've decided that I want to have a toolbar with the common commands that I don't type in. Once you have some commands on your toolbar, press **Apply** to see your new toolbar.



Now you will add icons to your new toolbar. Right-click a toolbar and select Customize again. You'll see all the commands listed. You can now drag and drop an icon from that list to your custom toolbar. For this exercise, look at the list for commands you don't recognize and add them to your toolbar so you can try them out.

# **Right-Click Customization**

You can also control how your mouse works. By default AutoCAD displays a menu when you right click outside of a command. To do this, type **OP** for options and go to the user preferences tab. Select the "Right-Click Customization" button. You'll see this dialog box.

Kight-Click Customization
Um on time-sensitive right-click Quick click for ENTER Longer click to display Shortcut Menu
Longer click <u>d</u> uration: 250 milliseconds
Default Mode
If no objects are selected, right-click means
Repeat Last Command
Shortcut Menu
Edit Mode If one or more objects are selected, right-click means © Repeat <u>L</u> ast Command © Shortcut <u>M</u> enu
Command Mode If a command is in progress, right-click means ENTER
Shortcut Menu: always enabled
Shortcut Menu: enabled when command options are present
Apply & Close Cancel <u>H</u> elp

Look at the top item. I suggest turning this on as it can speed things up. Unless you frequently use the menus, you should switch to this. Remember, you can always change it back if you don't like this setting.

#### Summary

These are just a few ways that AutoCAD can be customized. You can create custom hatch patterns and linetypes - even fonts. Other options include programming to automate tedious tasks in AutoLISP, VisualLisp, VBA or C++ (ARX). As you get more familiar with AutoCAD, look into these options. There is almost no end to how you can change the ways that AutoCAD looks and acts.

If you want to take the next step and look into programming for AutoCAD, check out this great site for learning AutoLISP - <u>Afralisp.net</u>.

## Video: Customizing AutoCAD

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Take the Lesson 4-12 Quiz

What is the name of the file which contains your aliases?

acad.pgp aliasedit.pgp

shortcuts.pgp



## For Further Thought:

If you have been paying attention through these AutoCAD tutorials (I hope so!) you have noticed that I try to remind you to be quick and accurate in your drawing. Learning how to customize AutoCAD to your specific way of drafting is a huge step to helping your work.

Only you will know which tasks you do repeatedly throughout the day. Learn how to create shortcuts and automate the repetition. This leads to greater accuracy, faster

drafting and less boredom.

Consider the difference between drawing something 20 times instead of inserting 20 blocks. These are the kinds of shortcuts you should continually be looking for. Good luck and thank you for using myCADsite - this is the last lesson (for now).

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