

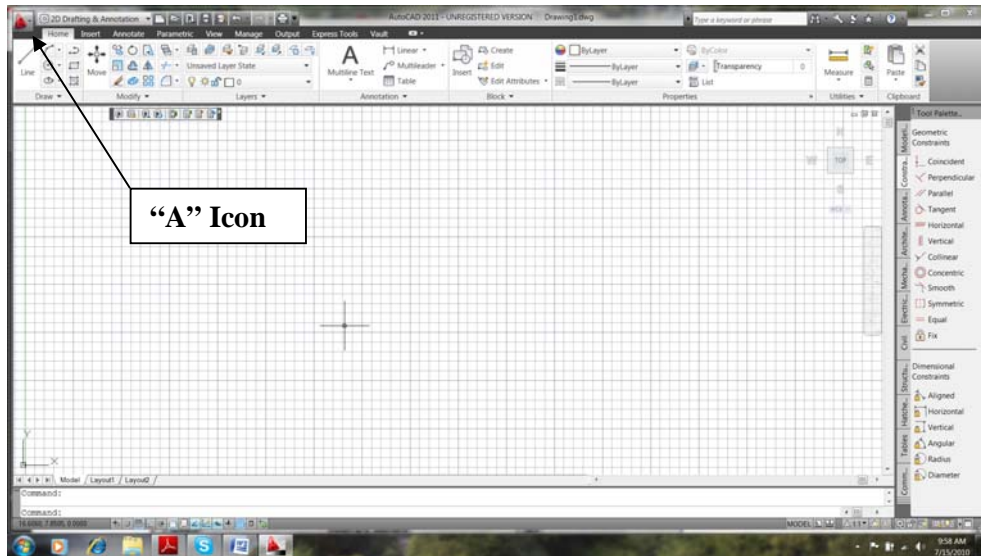
1 – Introduction to AutoCAD

The term CAD (Computer Aided Design) applies to a wide range of programs that allow the user to create drawings, plans, and designs electronically. AutoCAD is one such program and its main claim to fame is that it is relatively easy to use, it is very comprehensive in its ability to create 2D and some 3D drawings, and it is very popular. Seventy percent of the CAD users in the world use AutoCAD.

I Starting AutoCAD

You can start AutoCAD by either double clicking on the program icon on the desktop or by clicking on the program name in the Start menu.

The program will start and after a minute or so should display a screen similar to the one shown below. The dialog box in the middle will aid you in getting started at either creating a new drawing or continuing your work on a drawing that is not finished.



If you are continuing work on a drawing, click on the “A” icon in the extreme upper left corner of the window and Open->Drawing. A “Select File” dialog box will open allowing you to select the drawing file you want to open.

II The Initial Screen

AutoCAD has a very versatile user interface that allows you to control the program in several different ways. At the top of the window is a row of menus. Clicking on the Home, Insert, or Annotate causes another selection of menus to appear. This new selection of commands is frequently called a Ribbon or a Dashboard. You can operate the program by clicking on the icons in these menus.

Another method of using the program is typing in the command names. This is frequently faster than using drop down menus for frequently used commands because you do not have to search for the correct menu or icon. You just type in the command name. For the most part, we will use this approach in this series of

tutorials. The commands that you type will appear at the bottom of the of the AutoCAD window.

III The LINE Command

Now that you have started AutoCAD and configured tool bars you want, you are ready to start learning to use the program. We will start with relatively simple commands and eventually, in later lessons, look at some of the more complex things that AutoCAD can do. The first command we will look at drawing straight lines. At the keyboard, type:

line

and press the ENTER key. You can use either upper or lower case when you type in AutoCAD commands.

The program will respond with:

Specify First Point:

Each line has a beginning and ending point and the program wants you to specify the beginning point of the line. You enter the beginning point by either typing the point coordinates at the keyboard or by clicking the mouse on a location of the screen where you want the line to begin. It is certainly much simpler to click with the mouse than it is to type in coordinates but engineering drawings are drawn precisely to scale and for the most part we will have to enter coordinates from the keyboard.

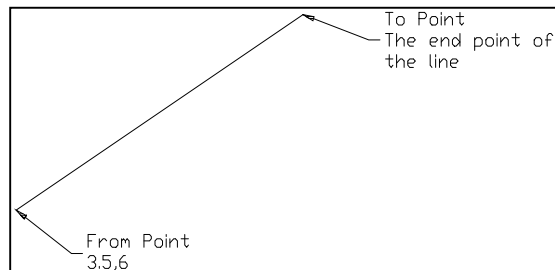
When you type a coordinate, enter the X or horizontal coordinate first followed by a comma and the Y or vertical coordinate. You cannot enter a space between the two coordinates. AutoCAD interprets a space as the ENTER key and assumes that you have finished entering the coordinates.

For Example, you could type:

Specify First Point: 3.5,6

The 3.5 coordinate is the X or horizontal coordinate and the 6 is the vertical coordinate.

After you enter the coordinates, press the enter key. The enter key tells the program that you have entered the first coordinate and are ready to enter the coordinates for the next which will be the end of the line. The program responds by displaying:

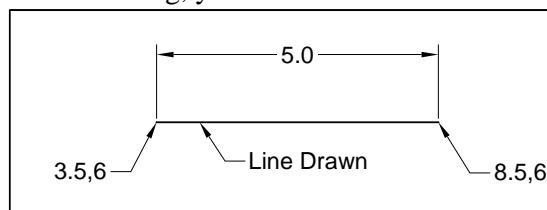


To Point:

If you want a horizontal line that is 5 units long, you enter the coordinates @5,0 which is shown below.

To Point: @5,0

The @ sign tells the program this coordinate is measured from the last coordinate entered. In other



words, it says place the end of the line 5 units horizontally from the beginning point and 0 units vertically. The line drawn is shown above.

Using the @ sign to specify relative coordinates is easier than specifying absolute coordinates without the @ sign. The first point we drew had an absolute coordinate of 3.5,6 and the second point had an absolute coordinate of 8.5,6 since it is displaced 5 units horizontally from the first point.

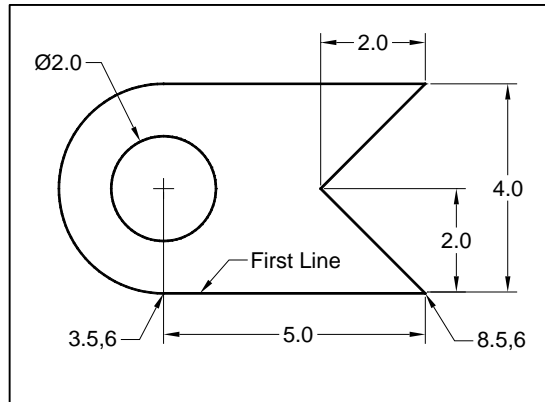
We will continue with this to create the object shown on the right. It has lines, an arc, and a circle. We have drawn the first and we will continue drawing the rest of the lines.

As a shortcut, you can start the **LINE** command by typing **L** instead of the entire word **LINE**. Many AutoCAD commands can be abbreviated to just the first letter of the command.

IV Continue Drawing the Object

We can continue drawing the object shown on the right by adding more lines. If the line command is still operating, press ENTER to end it. We will start it again to draw the remaining lines.

You can draw the remaining lines by typing:



line

Specify First Point: 8.5,6

{these are the coordinates of the end of the first line we drew}

To Point: @-2,2

To Point @2,2

To Point @-5,0

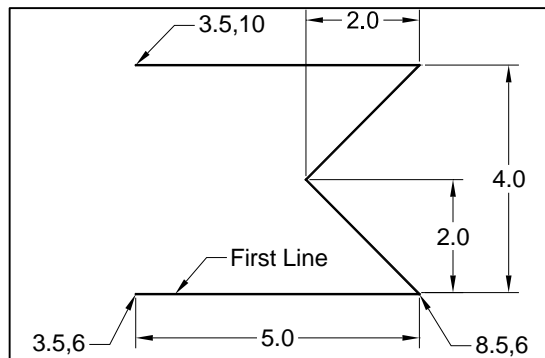
To point

{Press ENTER without entering coordinates. This will end the line command}

When you have finished entering all of the coordinates, you should have the object shown on the right.

V Erasing Objects

AutoCAD calls lines, circles, arcs and other things that you draw objects. You can erase any of these objects by typing the command:



ERASE

The program will respond with:

Select Objects:

You select the objects (lines, arcs, circles, etc.) in several different ways. The easiest way is to click on the object you want to erase. When you do, the object is redrawn as a dashed line. This shows the object has been selected for deletion. Click on all of the objects that you want to erase then press the ENTER key to terminate the command and erase the objects.

AutoCAD commands frequently have command modifiers that change the way the command works. For the ERASE command, you can type:

ERASE ALL

and AutoCAD selects all of the objects in the drawing for erasure. The word **ALL** modifies the way command works.

Another option is:

ERASE W

The **W** stands for window which allows you to select the objects by drawing a box around them. First click above and to the right of the objects that you want to erase. When you do, the mouse pointer changes to an elastic box with one corner fixed at the place where you clicked. Move the mouse until the box completely covers the information you want selected and click the mouse button again. All of the objects inside the box will be selected for erasure. Press the ENTER key to erase the objects.

You can type **E** to start the ERASE command.

VI Oops

If you make a mistake and erase something that you did not want to erase, type:

OOPS

to undo the last erasure. OOPS always undoes the last erasure even though you have continued with other commands since the objects were erased.

VII Canceling a Command

If you start a command and do not want to complete it, you can press the **Esc** key to cancel the command. For some commands, you may have to press the key more than once. Keep pressing the **Esc** key until you see the **Command:** prompt at the bottom of the screen.

VIII Drawing Arcs

The **ARC** command is used to draw arcs. We can use this command to draw the semicircle on the left side of the object. Enter:

Arc

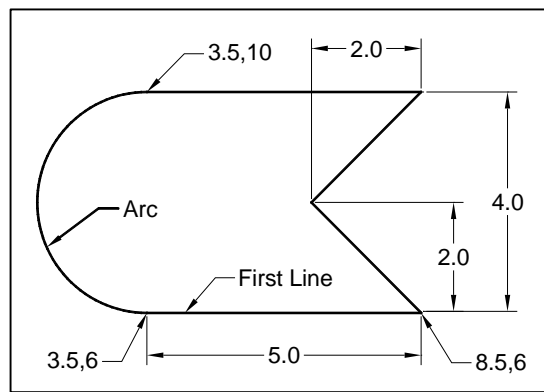
Specify start point of arc or [Center]: **3.5,10** *{The end point of the last line we drew}*

Specify second point of arc or [Center/End]: **c** *{Enter C to tell the program we want to enter the center point instead of the end point of the arc}*

Specify center point of arc: **@0,-2** *{The center of the arc is 2 units below the start point}*

Specify end point of arc: **@0,-2** *{The end of the arc is 2 units below the center}*

The completed arc is shown in the drawing on the right. Unless otherwise specified, AutoCAD will draw arcs in a counterclockwise (anticlockwise) direction.



IX Drawing Circles

Circles are created with the CIRCLE command. Type:

CIRCLE

at the command prompt and AutoCAD will respond with:

3P/2P/TTR/<Center point>:

There are several different ways you can define a circle. In the computer response above, the words **Center point** are surrounded by angle brackets and this shows you the program is expecting you to enter the coordinates of the center of the circle. You can either type the coordinates or click with the mouse. The quantity in angle brackets is always the default selection for a command. The letters 3P/2P/TTR/are options you can use to modify the input required to create a circle. These options are:

- 3P** Define the circle with 3 non-collinear points.
- 2P** Define the circle with points on either end of the circle diameter.
- TTR** Define the circle by specifying two other objects that are tangent to the circle and the radius of the circle.

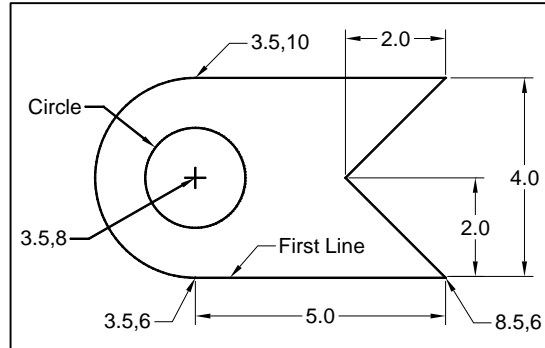
We can complete the drawing by drawing a circle. The center of the circle is two units vertically above the beginning point where we started the drawing. The coordinates for the center of the circle are 3.5,8. The circle has a diameter of 2.0.

```

circle
Specify center point for circle or [3P/2P/Ttr]: 3.5,8
Specify radius of circle or [Diameter]: d
Specify diameter of circle: 2

```

The completed object is shown in the figure on the right.



X Program Help

If you need more information on the various options for drawing an arc, park the mouse over the icon and after a few seconds, a help message will pop up. The help message will stay on the screen for as long as the mouse is parked over the icon. If you want more help, you can press the F1 key and AutoCAD will open a web page where you can look up the command and read a more in depth description of how it works.

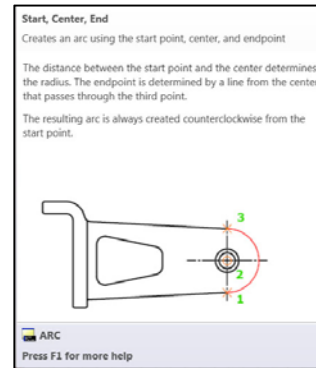


Figure 1 Help message produced when the mouse is parked over the arc command.

XI Undoing Mistakes

If you make a mistake with a command you can undo anything it has done by typing **U** at the command prompt. The entire effects of the last thing you typed will be undone and AutoCAD will return to the state it was in prior to the typing.

If you are inside a **LINE** or other command, you can enter **U** to undo the last coordinates entered or the last option selected.

You can undo the last command by holding down the **Ctrl** key and pressing **Z**. Each time you press **Z**, AutoCAD will remove the last command. If your press **Z** five times, it will backup through the last five commands.

XII Zooming in on Detail

You enlarge or reduce the size of an object on the screen with the ZOOM command. At the command prompt type:

ZOOM

and AutoCAD responds with:

```
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>:
```

There are many options to the ZOOM command. Some of the more useful are:

All	<i>Change the scale so that all of the drawing is shown in the window.</i>
Previous	<i>ZOOM to the previous view of the drawing.</i>
Scale	Enter a scale factor in the form nX where n is a scaling factor and X is just the letter X. A number larger than 1 makes the drawing appear larger and a number smaller than 1 makes the drawing smaller. For example 2 makes the drawing twice as large as it currently is and .5 makes the drawing half its current size.
Window	Allows you to draw a window around the area of the drawing you want to see enlarged. The window is drawn by selecting opposite diagonal corners with the mouse.

You can start the ZOOM command by just typing **ZOOM** or **Z** at the command prompt.

A very easy to zoom in or out can be done with the mouse. Move the mouse till it is near a location you want to remain on the screen. Roll the wheel between the two mouse buttons forward to make the object larger and roll it backwards (towards you) to make the object on the screen smaller.

If you zoom in to make the object larger, the object you want to see can zoom off of the screen if you did not place your mouse close to it before you started zooming. If it does scroll off of the screen, move your mouse over until it is as close as you can get to the object, hold down on the wheel, and drag the screen over until you can see the object you want to see. This operation is called PANNING.

XIII Size of Drawing Area

The drawing area is as large as you need it to be. The usable drawing area does not just consist of the area that you can see. You can pan around the drawing area to reveal areas of your drawing that are out of view. You can also ZOOM in and out to reveal more or less of the drawing area. Because the drawing area is so large, it is a good idea to indicate the region that you wish to use. This is your drawing size or limits. This is usually the area that will be printed. You can change your drawing size using the LIMITS command.

Setting your drawing size type:

limits

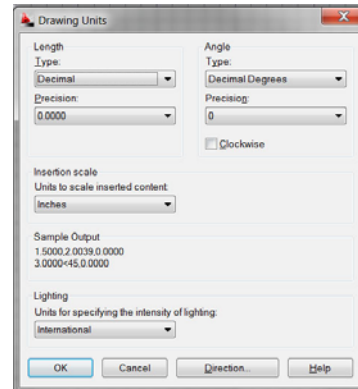
```
Specify lower left corner or [ON/OFF] <0.0000,0.0000>: {Enter the lower left corner of your limits.}
```

Specify upper right corner <420.0000,297.0000>:

*Usually this should always remain 0,0.
{Enter the coordinates for the upper right corner of the drawing area you want to use. You can use the mouse to select the upper right corner of the limits.}*

XIV Specifying the Units for the Drawing

The units (i.e. inches, millimeters, feet) used to draw objects in the drawing area can be selected using the UNITS command. Type UNITS then press enter. The following dialog box will appear allowing you to select the units and the number of decimals displayed in the commands.



XV Shortcuts

AutoCAD has many short cuts that make the work easier. One very useful one is pressing the ENTER key to repeat the last command.

XVI Saving Your Work

Periodically, you should save your work. You can do this by clicking on the “A” icon then selecting “Save” with your mouse. You can also save your work by typing Ctrl S (holding down the Ctrl key and pressing the S key.)

The first time you do this, AutoCAD produces a dialog box that allows you to select the file folder where you want the files saved and the name you want to use for the drawing. The next time you use the command, AutoCAD saves the file at the same location using the same name.

XVII Plotting

You can plot the drawings you have made by clicking on the “A” icon then selecting print. This creates the dialog box shown on the right.

Select the printer or plotter you are going to use then define the plotting area. If you set limits for the area you want to plot select Limits in the “What to plot:” pull down menu. If you did not define the limits, you can use Extents. The Extents will plot selection will plot everything you have drawn.

We are not plotting scaled drawings at this time so mark the “Fit to paper” box. This will stretch or shrink the drawing so that it fits on the paper.

Before you click on OK to plot the drawing, it is always a good idea to “Preview” to drawing. Click on this button to see what the drawing will look like on your sheet of paper. If it looks ok, you can click on the printer button in the preview screen. If it does not look ok, click on the circle with an X in it. This will return to the plot Model dialog box so you can make changes that will hopefully produce the plot you want. After you have made changes, be sure to preview it again to make sure it is working correctly.



XVIII Ending the Program

You terminate the program by clicking on “A” then “Exit AutoCAD” in the lower right corner of the pull down menu. The program asks you if you want to save the changes you have made to the drawing. You should click on the “Yes” button to save the changes.

Problems

