

Computer Aided Design (CAD)

Computer Aided Design is the use of computer technology for the design of objects, real or virtual. CAD systems are used in many fields where high precision is work is required. Hitherto, many geomatic applications had involved the use of paper maps, using drawing boards and other drawing tools to show the final representation of raw or processed spatial data(x, y and z coordinates etc.). In the production of maps (cartography) has taken a huge jump since the introduction of CAD systems, which has greatly enhanced the speed and accuracy of draughting or plotting. The use of CAD systems helps us to design or draw up features such as buildings, terrain, facilities and other assets. It is also simple to make certain calculations related to the area of interest, like area and volume calculations. Drawing using CAD software provides the ease and flexibility that cannot be found in traditional draughting. One of the advantages of using CAD systems is capability of 3-dimensional data representation. CAD may be used to design curves and figures in two-dimensional (“2D”) space; or curves, surfaces, and solids in three-dimensional (“3D”) objects. The convenience of CAD systems lie in the user’s ability to:

- ✚ Easily pan the map
- ✚ Zoom to any scale with ease
- ✚ Make multiple copies of drawn objects by the click of a button
- ✚ Draw and then erase objects in order to try possible designs
- ✚ View multiple views of a design within the object space

The basics of using AutoCAD, 2D/3D CAD software by AutoDesk are outlined in the following.

Drawing Objects



The Draw commands can be used to create new objects such as lines and circles. Most AutoCAD drawings are composed purely and simply from these basic components. A good understanding

Line	
Ray	
Construction Line	
Multiline	
<hr/>	
Polyline	
3D Polyline	
Polygon	
Rectangle	
<hr/>	
Arc	▶
Circle	▶
Donut	
Spline	
Ellipse	▶
<hr/>	
Block	▶
Point	▶
<hr/>	
Hatch...	
Boundary...	
Region	
Wipeout	
Revision Cloud	
<hr/>	
Text	▶
<hr/>	
Surfaces	▶
Solids	▶

of the Draw commands is fundamental to the efficient use of AutoCAD.

In common with most AutoCAD commands, the Draw commands can be started in a number of ways. Command names or short-cuts can be entered at the keyboard, commands can be started from the Draw pull-down menu, shown on the right or from the Draw toolbar. The method you use is dependent upon the type of work you are doing and how experienced a user you are. Use whatever method feels easiest or most convenient at the time. Your drawing technique will improve over time and with experience so don't expect to be working very quickly at first.

If you are working with the pull-down menus, it is worth considering the visual syntax that is common to all pull-downs used in the Windows operating system. For example, a small arrow like so " ▶ " next to a menu item means that the item leads to a sub-menu that may contain other commands or command options. An ellipsis, "... " after a menu item means that the item displays a dialogue box. These little visual clues will help you to work more effectively with menus because they tell you what to expect and help to avoid surprises for the newcomer.

AutoCAD Commands and their uses

FILLET	Constructs an arc of specified radius between two lines, arcs, circles, or will create arcs of the specified radius at the vertices of a polyline. Radius of the arc to be constructed may be set to 0, which will make a sharp corner
LINE or L	Draws straight lines
OFFSET	Creates a new line, polyline arc or circle parallel to the entity and at a specified distance from it.
ORTHO	Constrains drawing so that only lines aligned with the grid can be drawn -- usually means only horizontal or vertical lines, however, if the crosshairs are rotated through the "Snap" "Rotate" command sequence, the lines drawn are constrained to being parallel with the crosshair rotation. Constraint can be overridden by snapping to a point or by entering exact coordinates for endpoints.
PEDIT	Allows editing of polylines which are already drawn
POINT	Draws single points. Appearance of the points is set by the Variable PDMODE
REGEN	Regenerates the current Viewport by recalculating the vector locations of all endpoints
SCALE	Alters the size of existing objects
ROTATE	Rotates existing objects parallel to the current UCS
UCS	Defines or modifies the current User Coordinate System
ZOOM	Enlarges or reduces the display magnification of the drawing, without changing the actual size of the entities
EXTEND	Lengthens a line, arc, or polyline to meet a specified "boundary edge"

Lines

Lines are probably the most simple of AutoCAD objects. Using the Line command, a line can be drawn between any two points picked within the drawing area. Lines are usually the first objects you will want to draw when starting a new drawing because they can be used as "construction lines" upon which the rest of your drawing will be based. Never forget that creating drawings with AutoCAD is not so dissimilar from creating drawings on a drawing board. Many of the basic drawing methods are the same.

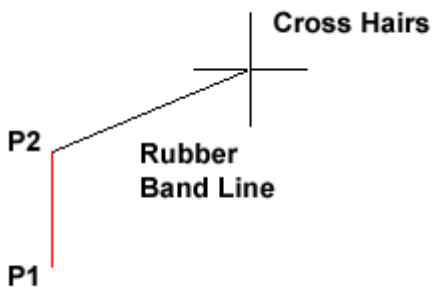
Anyone familiar with mathematics will know that lines drawn between points are often called *vectors*. This terminology is used to describe the type of drawings that AutoCAD creates. AutoCAD drawings are generically referred to as "vector drawings". Vector drawings are extremely useful where precision is the most important criterion because they retain their accuracy irrespective of scale.

The Line Command

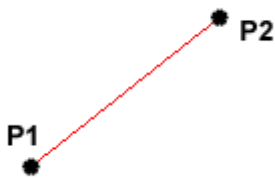
Toolbar Draw 

Pull-down Draw ▶ Line

Keyboard LINE short-cut L



With the Line command you can draw a simple line from one point to another. When you *pick* the first point and move the *cross-hairs* to the location of the second point you will see a *rubber band line* which shows you where the line will be drawn when the second point is picked. Line *objects* have two ends (the first point and the last point). You can continue picking points and AutoCAD will draw a straight line between each picked point and the previous point. Each line segment drawn is a separate *object* and can be moved or erased as required. To end this command, just hit the \leftarrow key on the keyboard.



Command Sequence

Command: LINE

Specify first point: (pick P1)

Specify next point or [Undo]: (pick P2)

Specify next point or [Undo]: \leftarrow (to end)

You can also draw lines by entering the co-ordinates of their end points at the command prompt rather than picking their position from the screen. This enables you to draw lines that are off

screen, should you want to. (See [Using Co-ordinates](#) for more details). You can also draw lines using something called direct distance entry. See the [Direct Distance Entry](#) tutorial for details.

The Construction Line Command

Toolbar Draw 

Pull-down Draw ▶ Construction Line

Keyboard XLINE short-cut XL

The Construction Line command creates a line of infinite length which passes through two picked points. Construction lines are very useful for creating construction frameworks or grids within which to design.

Construction lines are not normally used as objects in finished drawings, it is usual, therefore, to draw all your construction lines on a separate *layer* which will be turned off or frozen prior to printing. See the [Object Properties](#) tutorial to find out how to create new layers. Because of their nature, the Zoom Extents command option ignores construction lines.

Command Sequence

Command: XLINE

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: (pick a point)

Specify through point: (pick a second point)

Specify through point: ← (to end or pick another point)

You may notice that there are a number of options with this command. For example, the "Hor" and "Ver" options can be used to draw construction lines that are truly horizontal or vertical. In both these cases, only a single pick point is required because the direction of the line is predetermined. To use a command option, simply enter the capitalised part of the option name at the command prompt. Follow the command sequence below to see how you would draw a construction line using the Horizontal option.

Command Sequence

Command: XLINE

Hor/Ver/Ang/Bisect/Offset/<From point>: H ←

Through point: (pick a point to position the line)

Through point: ← (to end or pick a point for another horizontal line)

The Polyline Command

Toolbar Draw 

Pull-down Draw ► Polyline

Keyboard PLINE short-cut PL

The Polyline or Pline command is similar to the line command except that the resulting object may be composed of a number of segments which form a single *object*. In addition to the two ends a polyline is said to have *vertices* (singular *vertex*) where intermediate line segments join. In practice the Polyline command works in the same way as the Line command allowing you to pick as many points as you like. Again, just hit \leftarrow to end. As with the Line command, you also have the option to automatically *close* a polyline end to end. To do this, type C to use the close option instead of hitting \leftarrow . Follow the command sequence below to see how this works.

Command Sequence

Command: PLINE

Specify start point: (pick P1)

Current line-width is 0.0000

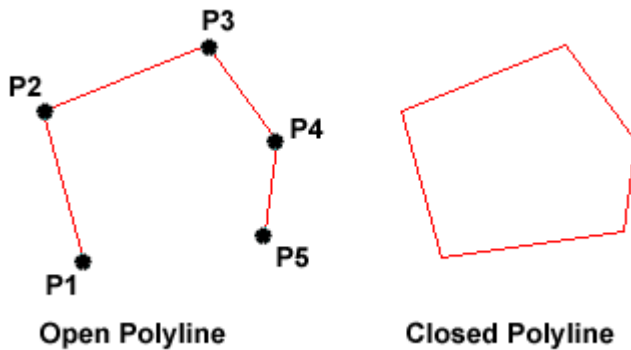
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: (pick P2)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: (pick P3)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: (pick P4)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: (pick P5)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: \leftarrow (or C to close)



In the illustration on the right, the figure on the left was created by hitting the \leftarrow key after the fifth point was picked. The figure on the right demonstrates the effect of using the Close option.

It is worth while taking some time to familiarise yourself with the Polyline command as it is an extremely useful command to know. Try experimenting with options such as Arc and Width and see if you can create polylines like the ones in the illustration above. The Undo option is particularly useful. This allows you to unpick polyline vertices, one at a time so that you can easily correct mistakes.

Polylines can be edited after they are created to, for example, change their width. You can do this using the PEDIT command, Modify ► Object ► Polyline from the pull-down menu.

The Rectangle Command

Toolbar Draw



Pull-down Draw ► Rectangle

Keyboard RECTANGLE short-cuts REC, RECTANG



The Rectangle command is used to draw a rectangle whose sides are vertical and horizontal. The position and size of the rectangle are defined by picking two diagonal corners. The rectangle isn't really an AutoCAD object at all. It is, in fact, just a closed polyline which is automatically drawn for you.

Command Sequence

Command: RECTANG

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: (pick P1)

Specify other corner point or [Dimensions]: (pick P2)

The Rectangle command also has a number of options. Width works in the same way as for the Polyline command. The Chamfer and Fillet options have the same effect as the [Chamfer](#) and [Fillet](#) commands, see the [Modifying Objects](#) tutorial for details. Elevation and Thickness are 3D options.

Notice that, instead of picking a second point to draw the rectangle, you have the option of entering dimensions. Say you wanted to draw a rectangle 20 drawing units long and 10 drawing units wide. The command sequence would look like this:

Command Sequence

Command: RECTANG

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: (pick a point)

Specify other corner point or [Dimensions]: D

Specify length for rectangles <0.0000>: 20

Specify width for rectangles <0.0000>: 10

Specify other corner point or [Dimensions]: (pick a point to fix the orientation)

This method provides a good alternative to using relative cartesian co-ordinates for determining length and width. See the [Using Co-ordinates](#) tutorial for more details.

The Polygon Command

Toolbar Draw 

Pull-down Draw ▸ Polygon

Keyboard POLYGON short-cut POL

The Polygon command can be used to draw any regular polygon from 3 sides up to 1024 sides. This command requires four inputs from the user, the number of sides, a pick point for the centre of the polygon, whether you want the polygon *inscribed* or *circumscribed* and then a pick point which determines both the radius of this imaginary circle and the orientation of the polygon. The polygon command creates a closed polyline in the shape of the required polygon.

This command also allows you to define the polygon by entering the length of a side using the Edge option. You can also control the size of the polygon by entering an exact radius for the circle. Follow the command sequence below to see how this command works.

Command Sequence

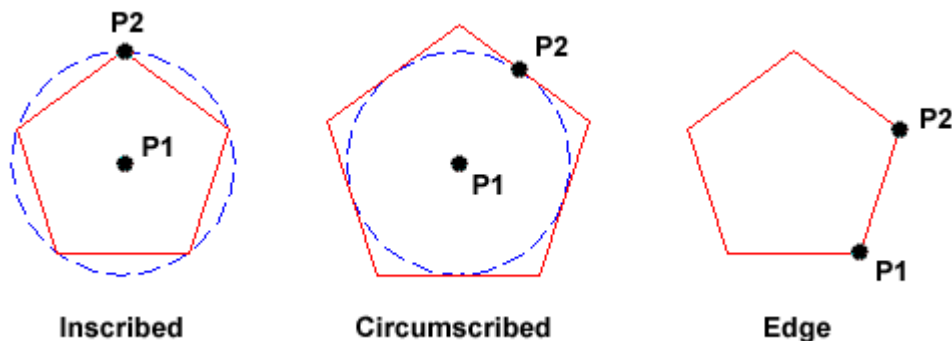
Command: POLYGON

Enter number of sides <4>: 5 ↵

Specify center of polygon or [Edge]: (pick P1 or type E to define by edge length)

Enter an option [Inscribed in circle/Circumscribed about circle] <I>: ↵ (to accept the inscribed default or type C for circumscribed)

Specify radius of circle: (pick P2 or enter exact radius)

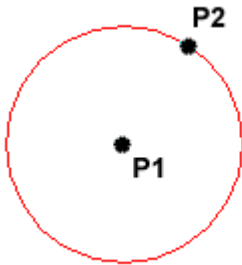



In the illustration above, the polygon on the left is inscribed (inside the circle with the polygon vertexes touching it), the one in the middle is circumscribed (outside the circle with the polyline edges tangential to it) and the one on the right is defined by the length of an edge.

Circles, Arcs etc.

Along with Line and Polyline, the Circle command is probably one of the most frequently used. Fortunately it is also one of the simplest. However, in common with the other commands in this section there are a number of options that can help you construct just the circle you need. Most of these options are self explanatory but in some cases it can be quite confusing. The Circle command, for example, offers 6 ways to create a circle, while the Arc command offers 10 different methods for drawing an arc. The sections below concentrate mainly on the *default* options but feel free to experiment.

The Circle Command



Toolbar Draw 
Pull-down Draw ▶ Circle ▶ Center, Radius
Keyboard CIRCLE short-cut C

The Circle command is used to draw circles. There are a number of ways you can define the circle. The default method is to pick the centre point and then to either pick a second point on the circumference of the circle or enter the circle radius at the keyboard.

Command Sequence

Command: CIRCLE


Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (pick P1)

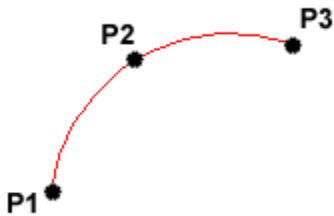
Specify radius of circle or [Diameter] <50.0195>: (pick P2 or enter the exact radius)

Center, Radius
Center, Diameter
2 Points
3 Points
Tan, Tan, Radius
Tan, Tan, Tan

As you can see from the command prompt above the default options are always indicated in triangular brackets like so <Default> and command options appear within square brackets like so [Option]. Each *option* is separated by a forward slash like this /. You can choose to use the alternative options by typing them at the prompt. For example, the circle command gives you three extra options to define a circle. 3P which uses any three points on the circumference, 2P which uses two points on the circumference to form a diameter and Ttr which stands for Tangent Tangent Radius. Obviously to use this last option you need to have drawn two lines which you can use as tangents to the circle. Try these options out to see how they work. Note that to invoke a command option, you need only type the upper-case part of the option name. For example, if you want to use the Ttr option, you need only enter "T". There are two more circle options on the pull-down menu that enable you to draw a circle by defining the center and diameter or by using 3 tangents.

The Arc Command

Toolbar Draw 
 Pull-down Draw ► Arc ► 3 Points
 Keyboard ARC short-cut A



The Arc command allows you to draw an arc of a circle. There are numerous ways to define an arc, the default method uses three pick points, a start point, a second point and an end point. Using this method, the drawn arc will start at the first pick point, pass through the second point and end at the third point. Once you have mastered the default method try some of the others. You may, for example need to draw an arc with a specific radius. All of the Arc command options are available from the pull-down menu.

Command Sequence

Command: ARC
 Specify start point of arc or [Center]: (pick P1)
 Specify second point of arc or [Center/End]: (pick P2)
 Specify end point of arc: (pick P3)

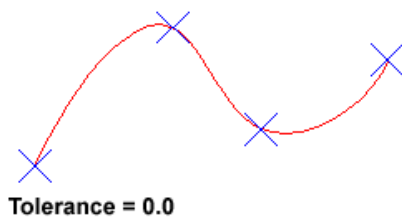
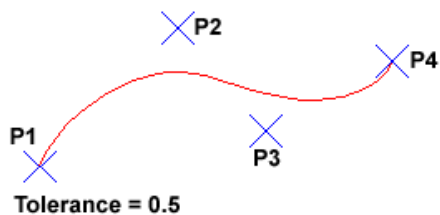
It is also possible to create an arc by trimming a circle object. In practice, many arcs are actually created this way. See the [Trim](#) command on the [Modifying Objects](#) tutorial for details.

The Spline Command

Toolbar Draw 

Pull-down Draw ▶ Spline

Keyboard SPLINE short-cut SPL



The Spline command creates a type of spline known as a nonuniform rational B-spline, NURBS for short. A spline is a smooth curve that is fitted along a number of control points. The Fit Tolerance option can be used to control how closely the spline conforms to the control points. A low tolerance value causes the spline to form close to the control points. A tolerance of 0 (zero) forces the spline to pass through the control points. The illustration on the right shows the effect of different tolerance values on a spline that is defined using the same four control points, P1, P2, P3 and P4.

Splines can be edited after they have been created using the SPLINEDIT command, Modify ▶ Object ▶ Spline from the pull-down menu. Using this command, you can change the tolerance, add more control points move control points and close splines, amongst other things. However, if you just want to move spline control points, it is best to use grips. See the [Stretching with Grips](#) section of the [Modifying Objects](#) tutorial for details.


Command Sequence

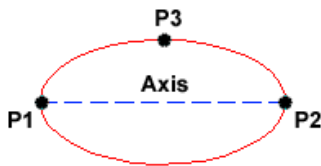
Command: SPLINE

Specify first point or [Object]: (Pick P1)
Specify next point: (Pick P2)
Specify next point or [Close/Fit tolerance] <start tangent>: (Pick P3)
Specify next point or [Close/Fit tolerance] <start tangent>: (Pick P4)
Specify next point or [Close/Fit tolerance] <start tangent>: ↵
Specify start tangent: (pick a point)
Specify end tangent: (pick a point)

You can create linear approximations to splines by smoothing polylines with the PEDIT command, Modify ► Polyline from the pull-down menu. However, you can also turn polylines into true splines using the Object option of the Spline command.

The Ellipse Command

Toolbar Draw 
Pull-down Draw ► Ellipse ► Axis, End
Keyboard ELLIPSE short-cut EL



The Ellipse command gives you a number of different creation options. The default option is to pick the two end points of an axis and then a third point to define the eccentricity of the ellipse. After you have mastered the default option, try out the others.

Command Sequence

Command: ELLIPSE
Specify axis endpoint of ellipse or [Arc/Center]: (pick P1)
Specify other endpoint of axis: (pick P2)
Specify distance to other axis or [Rotation]: (pick P3)

The ellipse command can also be used to draw isometric circles. See the [worked example](#) in the [Drawing Aids](#) tutorial to find out how to do this and how to draw in isometric projection with AutoCAD.

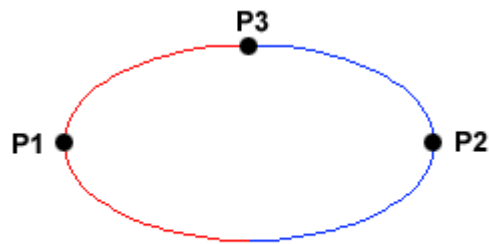
The Ellipse Arc Command

Toolbar Draw



Pull-down Draw ▸ Ellipse ▸ Arc

Keyboard ELLIPSE ←A short-cut EL ←A



Start = 270 degrees
End = 90 degrees



Result = half an ellipse

The Ellipse Arc command is very similar to the Ellipse command, described above. The only difference is that, in addition to specifying the two axis end points and the "distance to other axis" point, you are prompted for a start and end angle for the arc. You may specify angles by picking points or by entering values at the command prompt. Remember that angles are measured in an anti-clockwise direction, starting at the 3 o'clock position.

In truth, the Ellipse Arc command is not a new or separate command; it is just an option of the Ellipse command and it therefore has no unique command line name. It is curious why Autodesk considered this option important enough to give it its own button on the Draw toolbar. Still, there it is.

Command Sequence

Command: ELLIPSE

Specify axis endpoint of ellipse or [Arc/Center]: A

Specify axis endpoint of elliptical arc or [Center]: (pick P1)

Specify other endpoint of axis: (pick P2)

Specify distance to other axis or [Rotation]: (pick P3)

Specify start angle or [Parameter]: 270

Specify end angle or [Parameter/Included angle]: 90

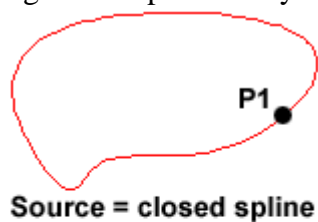
The Region Command

Toolbar Draw 

Pull-down Draw ▸ Region

Keyboard REGION short-cut REG

A region is a surface created from objects that form a closed shape, known as a loop. The Region command is used to transform objects into regions rather than actually drawing them (i.e. you will need to draw the closed shape or *loop* first). Once a region is created, there may be little visual difference to the drawing. However, if you set the shade mode to "Flat Shaded", View ▸ Shade ▸ Flat Shaded, you will see that the region is, in fact, a surface and not simply an outline. Regions are particularly useful in 3D modeling because they can be



extruded. **Result = region**

Before starting the Region command, draw a closed shape such as a rectangle, circle or any closed polyline or spline.

Command Sequence

Command: REGION
Select objects: (Pick P1)
Select objects: ←
1 loop extracted.
1 Region created.

You can use the boolean commands, Union, Subtract and Intersect to create complex regions.

Points and Point Styles

Points are very simple objects and the process of creating them is also very simple. Points are rarely used as drawing components although there is no reason why they could not be. They are

normally used just as drawing aids in a similar way that Construction Lines and Rays are used. For example, points are automatically created when you use the Measure and Divide commands to set out distances along a line.

When adding points to a drawing it is usually desirable to set the point style first because the default style can be difficult to see.

The Point Command

Toolbar Draw



Pull-down Draw ▶ Point ▶ Single Point

Keyboard POINT short-cut PO

The point command will insert a point marker in your drawing at a position which you pick in the drawing window or at any co-ordinate location which you enter at the keyboard. The default point style is a simple dot, which is often difficult to see but you can change the point style to something more easily visible or elaborate using the point style dialogue box. Points can be used for "setting out" a drawing in addition to construction lines. You can *Snap* to points using the Node object snap.

The Object Snaps (Osnaps for short) are drawing aids which are used in conjunction with other commands to help you draw accurately. Osnaps allow you to *snap* onto a specific object location when you are picking a point. For example, using Osnaps you can accurately pick the end point of a line or the center of a circle. Osnaps in AutoCAD are so important that you cannot draw accurately without them. For this reason, you must develop a good understanding of what the Osnaps are and how they work.

Command Sequence

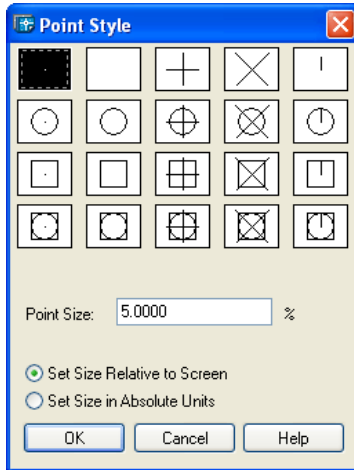
Command: POINT

Current point modes: PDMODE=0 PDSIZE=0.0000

Specify a point: (pick any point)

Strangely, in Multiple Point mode (the default for the Point button on the Draw toolbar) you will need to use the escape key (Esc) on your keyboard to end the command. The usual right-click or enter doesn't work.

The Point Style Command



Toolbar none

Pull-down Format ► Point Style...

Keyboard DDPTYPE

You can start the point style command from the keyboard by typing DDPTYPE or you can start it from the pull-down menu at Format ► Point Style... The command starts by displaying a *dialogue box* offering a number of options.

To change the point style, just pick the picture of the style you want and then click the "OK" button. You will need to use the Regen command, REGEN at the keyboard or View ► Regen from the pull-down to force any existing points in your drawing to display in the new style. Any new points created after the style has been set will automatically display in the new style.


One interesting aspect of points is that their size can be set to an absolute value or relative to the screen size, expressed as a percentage. The default is for points to display relative to the screen size, which is very useful because it means that points will remain the same size, irrespective of zoom factor. This is particularly convenient when drawings become complex and the drawing process requires a lot of zooming in and out.

Multilines

Multilines are complex lines that consist of between 1 and 16 parallel lines, known as *elements*. The default multiline style has just two elements but you can create additional styles of an almost endless variety. The Multiline Style command enables you to create new multiline styles by

adding line elements, changing the [colour](#) and [linetype](#) of elements, adding *end caps* and the option of displaying as a solid colour.

The Multiline Command

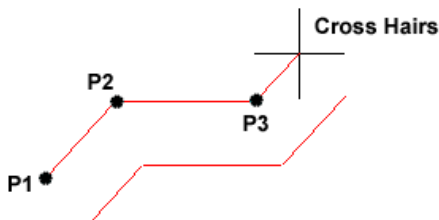
Toolbar custom 

Pull-down Draw ▾ Multiline

Keyboard MLINE short-cut ML

The Multiline command is used to draw multilines. This process of drawing is pretty much the same as drawing polylines, additional line segments are added to the multiline as points are picked. As with polylines, points can be unpicked with the Undo option and multilines can be closed.

When you start the Multiline command you also have the option to specify the Justification, Scale and Style of the multiline. The Justification option allows you to set the justification to "Top", the default, "Zero" or "Bottom". When justification is set to top, the top of the multiline is drawn through the pick points, as in the illustration below. Zero justification draws the centreline of the multiline through the pick points and Bottom draws the bottom line through the pick points. Justification allows you to control how the multiline is drawn relative to your setting out information. For example, if you are drawing a new road with reference to its centre line, then Zero justification would be appropriate.



The Scale option allows you to set a scale factor, which effectively changes the width of the multiline. The default scale factor is set to 1.0 so to half the width of the multiline, a value of 0.5 would be entered. A value of 2.0 would double the width.

The Style option enables you to set the current multiline style. The default style is called "Standard". This is the only style available unless you have previously created a new style with the Multiline Style command. Follow the command sequence below to see how the Multiline command works and then try changing the Justification and Scale options.

Command Sequence

Command: MLINE

Current settings: Justification = Top, Scale = 20.00, Style = STANDARD

Specify start point or [Justification/Scale/STyle]: (Pick P1)

Specify next point: (Pick P2)

Specify next point or [Undo]: (Pick P3)

Specify next point or [Close/Undo]: ← (to end or continue picking or C to close)

The trim, fillet and

The TRIM command allows you to shorten an entity (line, circle etc) to an intersection or remove a section of an entity between two intersections.

To select the **Trim** command you can:

type **trim** and pressing **Enter** at a **Command:** prompt,

OR

select **Trim** under the **Modify** menu,

OR

click on the **Trim**  **button.**

--Prompt: **Select cutting edges(s)...**

Select objects:

Click on entity or entities that the element should be trimmed **back to** or **between**. See **FIGURE** below.

--Prompt: **Select objects:**

Prompt repeats to allow you to continue selecting boundary edges. You can trim more than one element at a time.

--Press the **Enter** key to tell AutoCAD you are through selecting boundary entities.

--Prompt: **<Select object to trim>/Project**

/Edge/Undo: Click on the part of the objects you wish to remove.

--Prompt: **<Select object to trim>/Project**

/Edge/Undo: Either select additional objects that you wish to trim or press **Enter** to

exit the command.



The EXTEND command allows you to extend an entity to touch another entity in a drawing.

To access the **Extend** command you can:

click the **Extend**  **button,**

OR

select **Extend** under the **Modify Menu,**

OR

type **extend** at a **Command:** prompt and press **Enter.**

--Prompt: **Select boundary edge(s)...**

-Select objects: Click on the item(s) that you wish to use as a boundary. The boundary is the object you want to extend an element TO (you can select more than one boundary edge so that multiple extensions can be made at one time).

Press **Enter** or click the **right mouse button** to end the boundary selection process.

--Prompt: **<Select object to extend>/Project/Edge/Undo:** Click **on the end of the line you want AutoCAD to extend (on the side that should be extended).** AutoCAD is directional in this command. If you have more than one boundary edge selected, or the boundary edge could intersect the object at more than one end (as with an arc), it is possible to extend the element in the wrong direction.

--Prompt: **<SELECT object to extend> /Project/Edge/Undo:** If you have other lines to extend, you can continue selecting them or, if an element was extended incorrectly, you can type **u** and press **Enter** to undo the mistake. After all of the extensions are complete, press **Enter** to exit the command.

The OFFSET command allows you to copy selected entities (lines, arcs, circles, plines or others) and place the copy at a specific distance from the original or through an existing position on another element (such as the endpoint of a line,

center of a circle, etc.). An OFFSET item will be parallel or concentric to its original.

To access the **Offset** command you can:

type **offset** at a **Command:** prompt

OR

select the **Offset**  button on the **Drawing toolbar**.

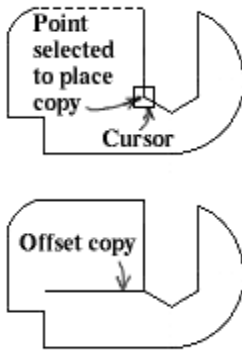
--Prompt: **OFFSET distance or through <through>**: Press **Enter** to select the default of **through** (for through a position on another entity) or **type a value** for the distance you want the offset to be from the original.

--Prompt: **Select object to offset**: Place the cursor on the element you wish to copy and click the LEFT mouse button. The selected object will change to a dashed line. Only one feature can be selected.



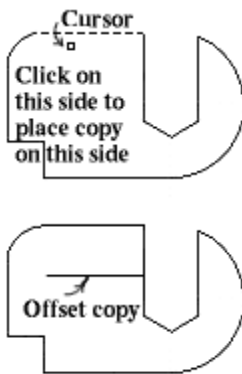
If Through was selected:

-Prompt: **Through point**: Use an [Osnap](#), an [Absolute Coordinate](#), a [Relative Rectangular Coordinate](#), or [Relative Polar Coordinate](#) to select the point to place the offset object through.



If a distance was typed at the prompt:

--Prompt: **Side to offset:** With the LEFT mouse button, click on the side of the original where you want the parallel copy located. **See the FIGURE below.**



NOTE: The last prompt will continue until you press **Enter** TWICE to get back to a **Command:** prompt and exit the **Offset** command.

Offset copies of lines are always parallel to the original.
 Offset copies of an arc or circle are always **CONCENTRIC** with the original.

Using AREA Command

2/17/2010 Erhan Toker in [Command](#) labels. [3 yorum](#)

As it can also be understood from its name AREA command is used to measure areas. Its usage is very simple. However, there are certain rules that you should pay attention to. Before explaining about them, let's start with the

fundamentals.

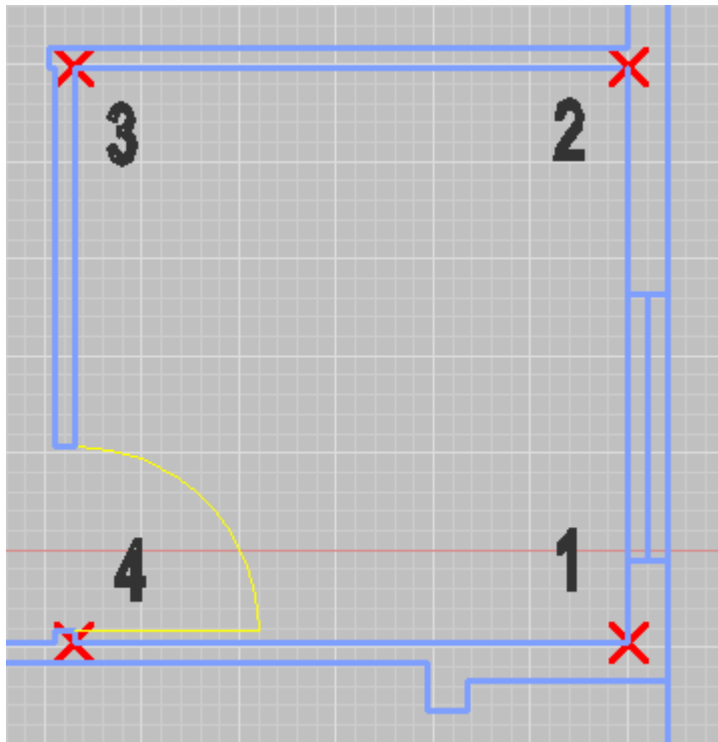


Fig.1

Let's try to measure the area that is shown in Fig. 1. It is very easy to access the command. All you have to do is to write 'AREA' into the command line.

```
Command : _AREA
Specify first corner point or [Object/Add/Subtract]: ( 1 )
Specify next corner point or press ENTER for total: ( 2 )
Specify next corner point or press ENTER for total: ( 3 )
Specify next corner point or press ENTER for total: ( 4 )
Specify next corner point or press ENTER for total: ( ENTER )
Area = 84360.0000, Perimeter = 1162.0000
```

Areas are defined by using the corner points that make them up. The result is displayed by the same unit that you assumed while creating the drawing. In the example drawing, this unit is accepted as cm. As a result, the measured area will be given as cm^2 . The final point that you select does not have to be the first one that was selected.

AutoCAD assumes that the first and the last point that you select are joined. As a result, you obtain both area and

perimeter.

Now, let's examine the other options of the command. As you may have noticed, when you first enter the command, you will get **[Object/Add/Subtract]** options. By using Object option, AutoCAD prompts you to select the entities that may create areas. You can select a polyline that has at least three corners (vertex) or any other closed entities. If the selection is not appropriate then AutoCAD will prompt you the error message 'Selected object does not have an area'. **Add/Subtract** options make the command in a continuous way and it permits you to select as many areas as you want until you finish the execution of command. The area that you select is either added to the total area or subtracted from it.

\

3D Modeling in AutoCAD- Solids

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).

The EXTRUDE command allows you to create a solid object from circles, polygons, and objects, created from or converted to [Polylines](#), that have "closed" paths.

To access the **Extrude** command you can:

type **extrude** or **ext** at a **Command:** prompt

OR

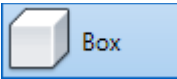





select **Solids**, and then **Extrude**, under the **Draw Menu**.

—Prompt: **Current wire frame density: ISOLINES=4**

Select objects: Click on the **object** you wish to extrude and press **Enter**.

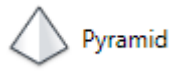
—Prompt: **Specify height of extrusion or [Path]:** Type in **value** and press **Enter**

(Click on the **ICON** below to go to the command)

SHAPE	COMMAND	ICON	DESCRIPTION
BOX	BOX	 Box	Creates a solid box after you provide 2 opposite corners and a height.
SPHERE	SPHERE	 Sphere	Creates a solid sphere from a center point and radius.
CYLINDER	CYLINDER	 Cylinder	Creates a straight cylinder from a center point, radius and height.
CONE	CONE	 Cone	Creates a tapered cone from a center point, radius and height.
WEDGE	WEDGE	 Wedge	Creates a triangular wedge from 2 opposite points.
TORUS	TORUS	 Torus	Creates a torus (donut shape) based on center point, radius and tube radius.

PYRAMID

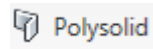
**PYRAMID /
PYR**



Draws a solid object with a polygon (3-32 sides) base that rises to a central point.

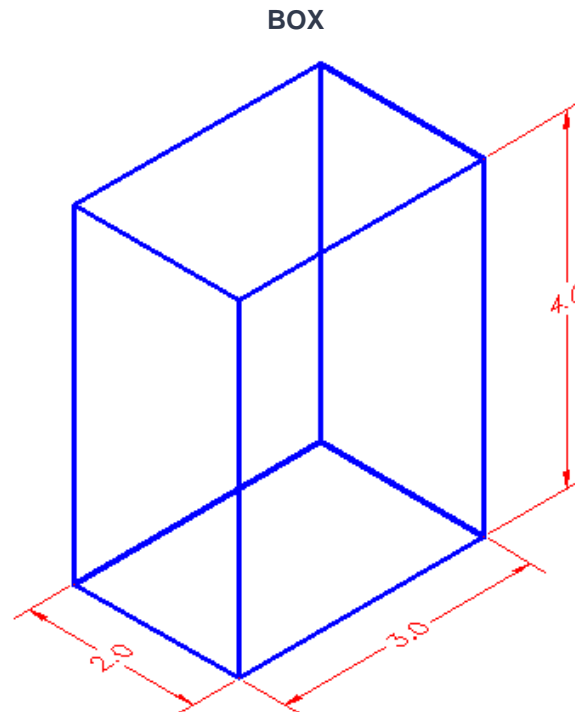
POLYSOLID

PSOLID



Draws a solid object with width and height as you would draw a polyline.

You can use primitives to either begin building a model, or it can even 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.



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 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>:

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.

AutoCAD 3D and AutoCAD Map are AutoDesk's software that has capabilities for analyzing 3 dimensional data like LIDAR point cloud data (x, y, z data). AutoCAD® Map 3D GIS mapping software offers easier access to design, GIS, imagery, point cloud, and business information from a broad range of sources, including Esri, Bentley, Oracle, GE, and other software providers. By using comprehensive data models for the gas, water, wastewater, and electric industries, you can organize disparate asset information and apply industry standards and business requirements to your data.

CAD STANDARDS

The essence of standards is to promote improvement of the quality of products, services and sound management practices in the geospatial industry. CAD drawings need to conform with industry standards so as to improve on sharing and appreciable understanding of the meaning of a drawing. The following is an extract from the City of Everett Survey and CAD standards. The standards describe vertical applications, layer names and color standards, linetypes and standard symbols that can be used in survey drawings. The CAD for a city or country is usually an elaborate document detailing every feature and its representation in terms of the geography and map symbology for both text and other graphics.

Abbreviation	Description
CULV	Culvert
CYL	Cylinder
D	
D	Depth, Dipole
DB	Direct Burial Cable
D/L	Daylight
D/W	Driveway
DBL	Double
DCL	Ditch CL
DE	DE-Energize
DEG	Degree
DET	Detail or Detector
DI	Ductile Iron
DIA	Diameter

E	
E	East, Electrical
EA	Each
ECb	Buried Electrical Cable
ECC	Eccentric
EF	Each Face
EJB	Electric Junction Box
EL	Easement Line, Elevation
ELB	Elbow
ELEC	Electrical
ELEV	Elevation
EMB	Embankment
EMH	Electrical Manhole
ENCL	Enclose
ENG	Engine
ENGR	Engineer
EO	Edge of
EOA	Edge of Asphalt
EOC	Edge of Concrete
EOD	Edge of Dirt
EOG	Edge of Gravel
EP	Edge of Pavement
EQ	Equal
EQUIP	Equipment
ET	Everett Transit
EV	Electrical Vault
EVC	End Vertical Curve
	Emergency Vehicle Pre-
EVP	Emption
EVT	Everett, City of
EW	Each Way
EX	Existing, Example
EXC	Excavation
EXIST	Existing
EXP	Expansion
EXP	JT Expansion Joint
EXT	Exterior, Extension, Extruded
F	
F	Flanged

ELECTRICAL

CABINETS

ELEC-SCAB-36X28	
36"X28" SERVICE CAB W/PHOTO CELL	
LT-SERVE	
UN-METERED 5T LIGHT SERVICE CAB W/BASE	
SG-SERVE	
SIG SERVICE CAB	
SG-SERVEB	
SIG SERVICE CAB W/BASE	
SG-SERVP	
POLE/PEDESTAL MOUNTED SIG SERVICE CAB	

JUNCTION BOXES

LT-JBHWL	
WALL MOUNTED JB	
EL-JB1 DOT / SG-JB1	
DOT TYPE 1 JB	
EL-JB2 DOT / SG-JB2	
DOT TYPE 2 JB	
EL-JB3 DOT / SG-JB3	
DOT TYPE 3 JB	
EL-JB-RD / SG-JB-RD	
ROUND ELEC JB	
SG-JB-SP	
42"X60" ELEC JB	
UT-TEL-PED	

GRAVITY SYSTEMS

CATCH BASINS

UT-CB-2G	
TYPE 2 CATCH BASIN WITH GRATE	
UT-CB-2R	
TYPE 2 CATCH BASIN WITH ROUND SOLID LID	
UT-CB-2S	
TYPE A CATCH BASIN WITH SOLID SQUARE LID	
UT-CB-AG	
TYPE A CATCH BASIN WITH GRATE	
UT-CB-AS	
TYPE A CATCH BASIN WITH SOLID SQUARE LID	
UT-CB-BG	
TYPE B CATCH BASIN WITH GRATE	
UT-CB-BS	
TYPE B CATCH BASIN WITH SOLID SQUARE LID	

INLETS

UT-ILOLD	
OLD RECTANGULAR INLET	
UT-INLET	
INLET	

LANDSCAPING

LD-BUSH	
LARGE BUSH	
LD-BUSH-2	
DECIDUOUS BUSH	
LD-BUSH-3	
CONIFER BUSH	
LD-HEDGE	
HEDGE	
LD-SHRUB	
SHRUB	
LD-TR-D10FT	
DECIDUOUS TREE 10' DIA DRIP LINE	NAME
LD-TR-C10FT	
CONIFER TREE 10' DIA DRIP LINE	NAME
LD-TR-C(S,M,L)	
CONIFER TREE	NAME
LD-TR-D(S,M,L)	
DECIDUOUS TREE	NAME

LIGHTING

LT-Y-BX	
SQUARE OFFSET YARD	

PRESSURE SYSTEMS

BENDS

WTR-BND11	
11 1/4" BEND	
WTR-BND22	
22 1/2" BEND	
WTR-BND45	
45" BEND	
WTR-BND90	
90" BEND	

CONNECTIONS

WTR-COUPLING	
COUPLING	
WTR-CROSS	
CROSS	
WTR-TEE	
TEE	
WTR-WYE	
WYE	
WTR-UNION	
UNION	
WTR-FL	
FLANGE	
WTR-FLXFL	
FLANGE X FLANGE	
WTR-MJ	
MECHANICAL JOINT	
SF-S	
CONTINUATION SYMBOL	
UT-T-BLK	
THRUST BLOCKING	

RAMPS

PV-WC-310A	
TYPE A 5' MIN SIDEWALK	
PV-WC-310B S(15,20,25,30)R	
TYPE B SINGLE XWALK 6' MIN SIDEWALK	
PV-WC-310B D(15,20,25)R	
TYPE B TWO XWALKS 6' MIN SIDEWALK	
PV-WC-310C-(5,6)FT	
TYPE C 5' OR 6' SIDEWALK	
PV-WC-(15,20,25,30)	
OLD TYPE B 5.5' MIN SIDEWALK 6' WING	
PV-WCX(15,20,25,30)	
OLD TYPE B 3' WING	
PV-WCXST	
OLD TYPE A 3' WING	

SIGNING

SN-O-PST	
SIGN ONLY	
SN-T-PST	
SIGN W/POST	

STRIPING

SP-2WLT	
TWO-WAY LEFT TURN	
SP-BICYC	
BICYCLE LANE	
SP-GUARD	
6" DIA CONC GUARD POST	
SP-HANDY	
HANDICAP PARKING	
SP-HOV	
HOV LANE	
SP-LT	
LEFT TURN	
SP-LT-RT	
LEFT & RIGHT TURN	
SP-LT-TH	
LEFT TURN & THRU	
SP-ONLY	
ONLY	
SP-RPM-1	
TYPE 1 ROUND RPM	
SP-RPM-2	
TYPE 2 SQUARE RPM	
SP-RR	
RAILROAD CROSSING	
SP-RT	
RIGHT TURN	
SP-RTL	
RIGHT TURN & THRU	
ALL WAY	
SP-RT-TH	
RIGHT TURN & THRU	

UTILITY LINES GENERIC (UT)

-EV-UT-WATER	=====
-EV-UT-SD	-----
-EV-UT-SS	-----
-EV-UT-EL	=====
-EV-UT-G	-----

WATER MAIN LINES (WTR)

-EV-WTR-W	=====	W	=====	W	=====
-EV-WTR-2	=====	2"W	=====	2"W	=====
-EV-WTR-2PL	=====	2"W PL	=====	2"W PL	=====
-EV-WTR-2.5	=====	2-1/2"W	=====	2-1/2"W	=====
-EV-WTR-3	=====	3"W	=====	3"W	=====
-EV-WTR-3CI	=====	3"W CI	=====	3"W CI	=====
-EV-WTR-4	=====	4"W	=====	4"W	=====
-EV-WTR-4AC	=====	4"W AC	=====	4"W AC	=====
-EV-WTR-4DI	=====	4"W DI	=====	4"W DI	=====
-EV-WTR-6	=====	6"W	=====	6"W	=====
-EV-WTR-6AC	=====	6"W AC	=====	6"W AC	=====

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