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

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

Object Snaps

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Entering Points in AutoCAD

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

ABSOLUTE CO-ORDINATES - Using this method, you enter the points as they relate to the origin of the WCS. To enter a point just enter in the exact point as X,Y.

RELATIVE CO-ORDINATES - This allows you to enter points in relation to the first point you have entered. After you've entered one point, the next would be entered as **@X,Y**. This means that AutoCAD will draw a line from the first point to another point X units over and Y units up relative to the previous point.

POLAR CO-ORDINATES - You would use this system if you know that you want to draw a line a certain distance at a particular angle. You would enter this as **@D<A**. In this case, **D** is the distance and **A** is the angle. Example: @10<90 will draw a line 10 units straight up from the first point.

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

The AutoCAD Screen

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

- **Application Button** - This button displays commands for printing, saving, drawing utilities and other non-drawing tool.
- **Quick Access Toolbar** - This is for quick access to common commands like New, Open, Save, Plot
- **Filename** - The name of the current file you are working on.
- **Search Bar** - Search for text in your drawing or search the help files.
- **Ribbon** - The Ribbon has most of the commands/tools that you will use while you

are working.

- **Tabs** - A series of Tabs make up the Ribbon (Home, Insert, Manage, etc) and organize the Tools into common groups.
- **Panels** - Contain a group of tools
- **Tools** - These are the icon that start the commands you use to draw, modify, etc.
- **Tool Tip** - If you hover your mouse over a tool, a tool tip will appear to give your more information. Hold it longer for more info.
- **Drawing Space** - These is where you draw your designs.
- **Command line** - When you type a command, you will see it here. AutoCAD uses this space to 'prompt' you for information. It will give you a lot of information and tell you where you are in the command. **Watch this line while learning.**
- **Status bar** - This allows to see and change different modes of drawing such as Ortho, Osnaps, Grid, Otrack, etc. You can right click this area to toggle between icons and text for this area.
- **Basic AutoCAD Terminology**
- Here are some basic terms that you will want to review before using AutoCAD. Some terms have links to give you more information (but it is not necessary to memorize them all now).

Absolute coordinates	A way of inputting points based on AutoCAD's origin.
Acad.dwt	This is the default template that automatically loads whenever you start a drawing session. It can be customized to suit your needs.
Associated Dimensioning	Dimensions that are associated with specific points will update as that point is moved.
Backup file	AutoCAD can be set to automatically backup your drawing and save it. This is a safeguard in case your file gets corrupted. It is saved with a .BAK extension
Block	A pre-drawn image you can insert in your drawing to save time and make your file size smaller.
Clean Screen	A display setting that gives you maximum drawing space.
Crosshairs	This is your cursor when it is in the drawing space.
Cursor	Your cursor will change depending on what function it is performing in the program.
Database	An AutoCAD drawing file is actually one large database containing all the information needed to reproduce the objects when the file is opened. Info for layers and linetypes, etc are stored in this manner.
Dialog box	AutoCAD uses a large number of dialog boxes to get information from you. You must know how input the information that it asks for.
Drawing template file	This is a file that contains preset values for frequently used settings. AKA a prototype drawing. The file extension is DWT.

Extents	The outer boundaries of the objects you have drawn.
Grid	This is pattern of dots displayed on the screen to guide you. It can be toggled on and off by pressing the F7 key.
Grips	Small 'handles' on objects that allow for quick editing.
Layer	All objects are drawn on a layer. You can group objects (such as electrical) on a single layer and organize your drawing.
Layout Tabs	A space used for plotting your drawings (formerly called Paper Space).
Limits (Grid)	A setting to impose an 'artificial' boundary on your drawing that sets the area of the grid, and when turned on, limits you to drawing in the grid area.
Linetype	All objects are drawn with a particular linetype. Examples would be solid, center, dashed, etc.
Model space	The drawing space where you 'model' the objects.
Modify	A generic term used for changing your objects
Object	Any item that is in the AutoCAD database. Also known as an entity.
Origin	The (0,0) point of your current coordinate system.
Ortho mode	This is a drawing mode that allows you to draw only perpendicular lines. It is toggled on and off by pressing the F8 key.
Orthographic Projection	A standard drawing method that shows 2 or more views of the same part.
Osnap - Object Snap	This is a method of 'snapping' to certain, precise points on an object.
Pan	To move around drawing by dragging the drawing area around your screen.
Panel	A grouping of commands on the ribbon
Path	The specific folder where AutoCAD looks for, or saves files.
Pick	To select an object by 'left-clicking' on it.
Plot	Also known as print. To make a hard copy of your drawing.
Polar coordinates	A way of inputting points based on distance and angle.
Property	Any specific characteristic of an object such as layer, scale, linetype, start point, etc.
Ribbon	The Ribbon runs across the top of the drawing space and contains panel - each panel has a group of associated tool. Switch to different panels by clicking on the tabs at the top of the ribbon.

Relative coordinates	A way of inputting points based on a starting point.
Section View	A drawing that represents a cross section of a part or assembly.
Selection set	The current group of objects selected for modifying.
Snap	This is a drawing mode that allows you to snap your cursor to precise points laid out in a grid pattern. Toggle with the F9 key.
Styles	Formatting that defines the look of text, dimensions, etc.
Units	The basic drawing unit set for you drawing. For example, you can use inches or millimeters depending on your needs. You can also set the precision you want displayed, such nearest 1/4", 1/2" 1/64", etc.
User coordinate system (UCS)	Modifications made to the World Coordinate System (WCS) results in a User Coordinate System (UCS)
View	A particular area of your drawing.
Viewport	A separate 'window' on your drawing. You may have more than one viewport visible to see different areas of your drawing at the same time.
Wizard	An easy step-by-step instruction set to help you set-up certain aspects of your drawing.
World Coordinate System (WCS)	This is the common X-Y coordinate system that is the default. If it is modified, it becomes a User coordinate System (UCS)
Zoom	To view either a smaller section of your drawing (zoom in) or a larger section (zoom out)

THE COORDINATES SYSTEM

Specifying coordinates on the screen is one of the most fundamental tasks in AutoCAD. Unless you know how to specify a point you can't draw anything real.

Every point on your screen is defined in terms of **X** and **Y** coordinates.

Absolute cartesian coordinates

When you type aline and enter the actual coordinates such as line from point 3,2 to 5,8 you are using absolute cartesian coordinates. Absolute coordinates are measured from 0,0.

Relative cartesian coordinates

Relative coordinates specify the X and the Y distance from the previous point. They are called relative coordinates because they have meaning relative to a point previously defined.

You tell AutoCAD that the coordinates are relative by using the @symbol

Polar Coordinates:

Another common situation is to know the distance and angle of a point from either 0,0 or a previous point. In this case you can use the polar coordinates, which can be either absolute or relative.

Polar Coordinate take the form of *distance<angle.*

Polylines.

. Polylines are single objects that can combine line segments and arcs they are unique in that they can allow you to make thick lines and arcs.

PLINE

This command Creates two-dimensional polylines

At the Command prompt, enter pline

From point: Specify a point.

Current line-width is <current>

Arc / Close / Halfwidth / Length / Undo / Width / <Endpoint of line>: Specify

a second point or enter an option

SPLINE

This command Creates a quadratic or cubic spline (NURBS) curve

SPLINE fits a smooth curve to a sequence of points within a specified tolerance.

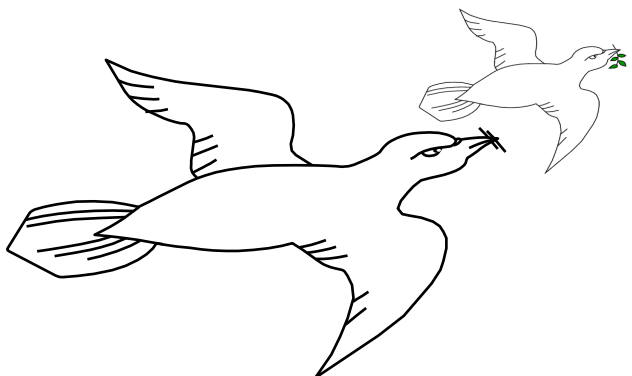
HATCH

HATCH fills the specified hatch boundary with a nonassociative hatch. A nonassociative hatch is not updated when its boundaries are modified. A hatch boundary consists of an object or objects that completely enclose an area. If the boundary is made up of multiple objects, their endpoints must coincide for the hatch to be created properly. You can also define a polyline hatch boundary with the direct hatch option. Unless otherwise specified, HATCH combines the lines that make up the hatch into a block.

SKETCH

This command lets you Create a series of freehand line segments and drawings

Drawing with the SKETCH command controls a screen-based pen with a pointing device. SKETCH is useful for entering map outlines, signatures, or other freehand drawings. Sketched lines are not added to the drawing until they are recorded. The standard button menu is disabled while SKETCH is in progress.



BLOCKS:

A common drawing task is placing the same group of objects several times in a drawing. An architect needs to place windows and doors many times in the layout plan of a house. An electrical engineer places electrical symbols in a drawing over and over. A mechanical model may include many nuts ,bolts ,surface finish symbols etc. blocks are group of objects that you create ,save and name so that you can insert them in your drawing whenever you need them. A block is one object regardless of the number of individual objects used to create it.once an object is made a block its grouped automatically. Thus it can be ungrouped by exploding it.

BLOCK

This command Creates a block definition from a set of objects

At the Command prompt, enter block

Block name (or ?): Enter a name or ?

EDITING DRAWINGS

Copying commands.

Four commands allows you to copy objects in very specific ways. These are mainly the Mirror,Array,Copy and Offset commands.

Mirror Command

Many drawings have symmetrical elements. Often in mechanical drawings you can create one-half or one-quarter of an object and complete it with simply by mirroring what you have drawn. To mirror select an object and

then choose mirror from the modify toolbar (or typr mirror at the prompt) Choose two points that will be used as mirror line.

Array command

The array command creates a rectangular or circular pattern by copying the object(s) you select as many times as you specify.

To create an array select the object or objects and choose array from the modify toolbar.

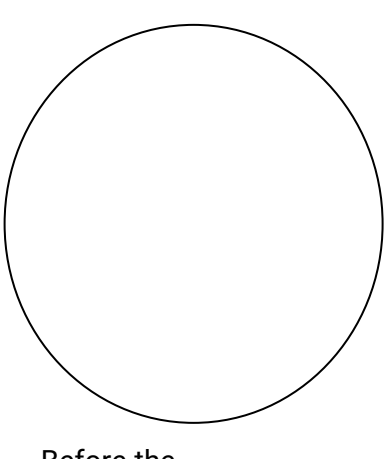
Offset command

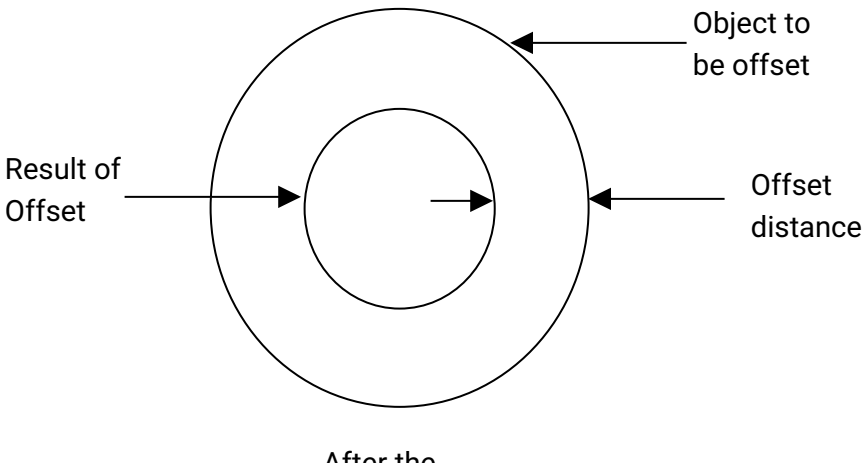
OFFSET creates a new object at a specified distance from an existing object or through a specified point.

Type the command **offset** command

When you specify a distance, offset Creates an object at a specified distance from an existing object.

When you choose through, offset Creates an object through the point you have specified.





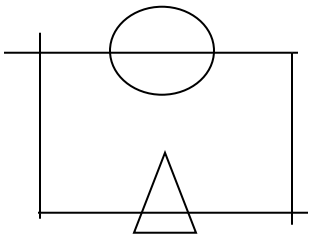
Copy command

The copy command duplicates a drawing.

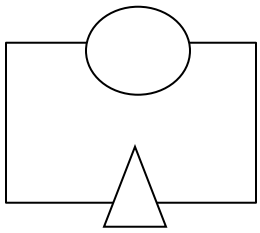
Type the command **copy** then select an object you want to copy.

Trimming objects

As you edit your drawing you may realise that the lines that used to meet perfectly now hang over. In this case you need to trim such objects. To trim an object you must first specify the cutting edge. When you select an object to trim you must select the object that that you want trimmed.



Objects before the trim effect



Objects after the trim effect

Erasing an object

The Erase command Removes objects from a drawing

From the Modify menu choose Erase, while At the Command prompt, enter erase

Select objects: Use an object selection method to select the objects you want to erase.

AutoCAD removes the objects from the drawing.

Moving an object

Now and then you will be required to move objects from one position to another in your drawing. Basically the command **move** Displaces objects a specified distance in a specified direction in your drawing.

To move an object
At the Command prompt, enter **move**
Select objects: Use an object selection method to select the objects you want to move
Base point or displacement: Specify a base point.
Second point of displacement: Specify a second point or press

The two points you specify define a displacement vector that indicates how far the selected objects are to be moved and in what direction. If you press enter at the second point, the first point is interpreted as relative X,Y,Z displacement.

Rotating an object

You may require to rotate your objects to suit your needs.the command **rotate** Moves objects about a base point. To rotate an object about a point

At the Command prompt, enter rotate

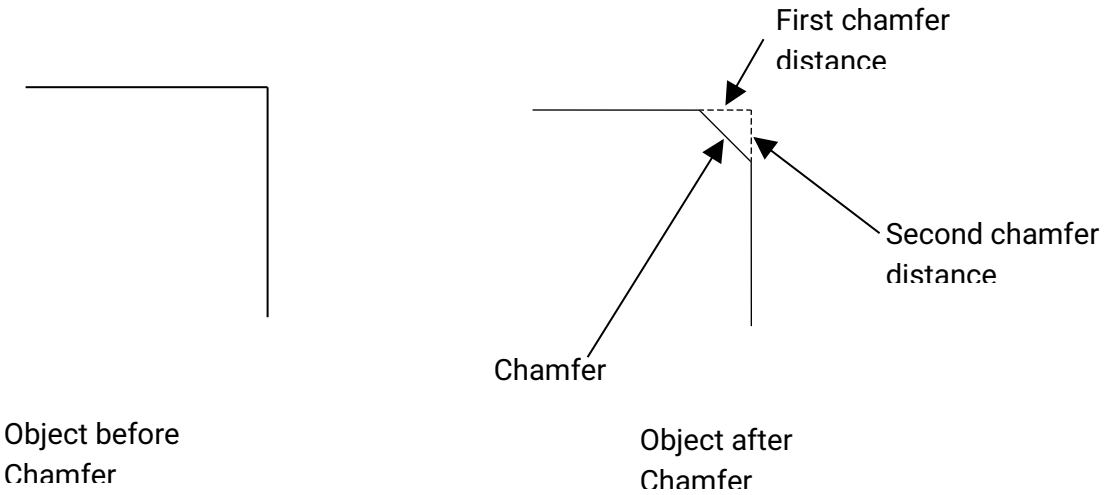
Select objects: Use an object selection method to select athe object you want rotated.

Base point: Specify a point

<Rotation angle> / Reference: Specify an angle for rotation.

Chamfer

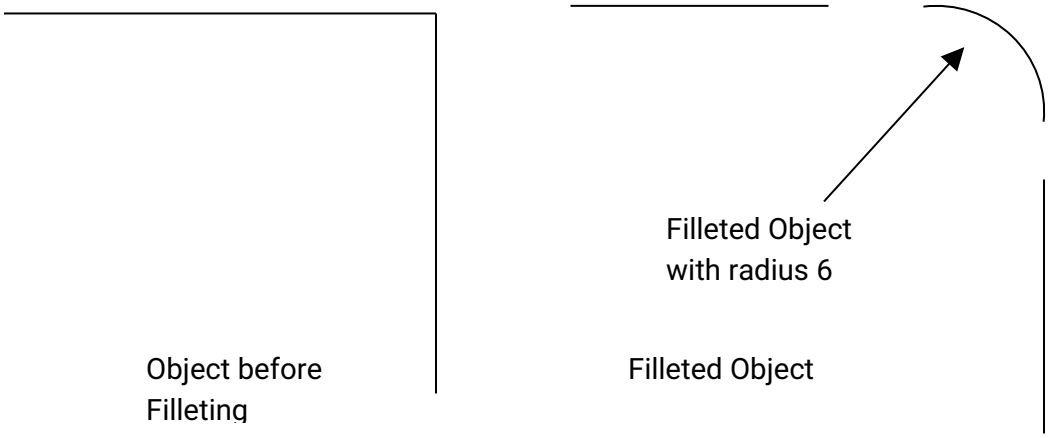
The **chamfer** command creates corners from two nonparrallel lines. Chamfering is a two step process. First you define how you want to chamfer the corner after which the command ends. To actually chamfer you need to start the chamfer command again . AutoCAD chamfers them using the information you previously specified.



Fillet

The FILLET command rounds or fillets the edges of two arcs, circles, elliptical arcs, lines, polylines, rays, splines, or xlines with an arc of a specified radius. If the selected lines do not intersect, AutoCAD extends or trims them so that they do. FILLET also rounds or fillets the edges of solids. If both objects to be filleted are on the same layer, AutoCAD creates the fillet line on that layer. Otherwise, AutoCAD creates the fillet line on the current layer. This is also true for the fillet color and linetype.

You cannot fillet line segments that intersect outside the drawing limits when limits checking is on.

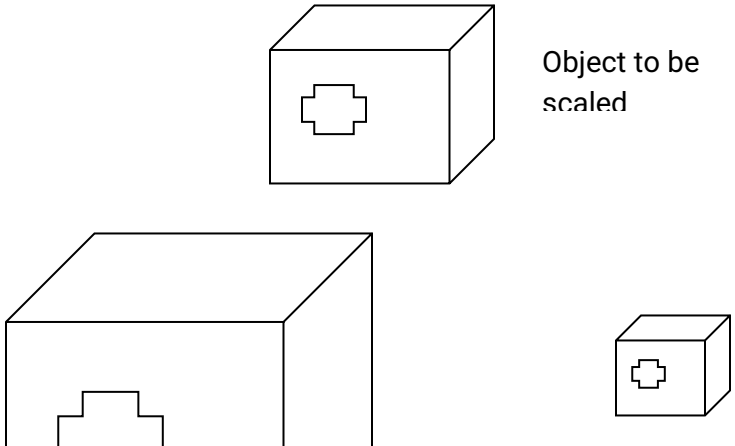


Dividing objects

The **Divide** command Places evenly spaced point objects or blocks along the length or perimeter of an object

Scaling objects

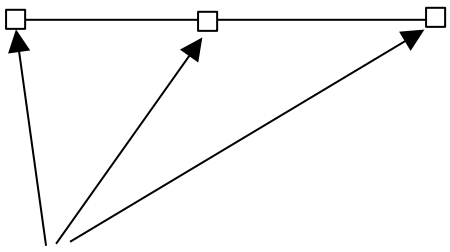
The **Scale** command Enlarges or reduces selected objects equally in the X, Y, and Z directions



Using Grips:

Grips offer a whole new way to edit objects without choosing commands. By using grips you can quickly stretch, move,rotate,scale and mirror objects.

When you select an object without first choosing a command the object appears highlighted with grips.



Line with Grips

Viewing drawings

Regen

REGEN regenerates the entire drawing and recomputes the screen coordinates for all objects. It also reindexes the drawing database for optimum display and object selection performance.

At the Command prompt, enter regen AutoCAD regenerates the current view and it also removes blip marks

Redraw

This command Refreshes the display of the current viewport removes marker blips and display artifacts (stray pixels) left by editing commands.

Pan

This command Moves the drawing display in the current viewport

PAN works in two ways. You can specify a single point, indicating the relative displacement of the drawing with respect to the screen, or (more commonly) you can specify two points, in which case AutoCAD computes the displacement from the first point to the second point. You can use PAN transparently but you cant use it transparently while in paper space

Zoom

The **Zoom** command Increases or decreases the apparent size of objects in the current viewport

You can use ZOOM transparently. You can't use ZOOM transparently while in paper space or while another ZOOM, PAN, or VIEW command is in progress.