Complete AutoCAD Commands

Command	Description	Options
ABOUT	Displays a dialogue box with AutoCAD version & serial numbers	can be transparent
APERTURE	Controls the size of the Object Snap "OSNAP") target box -- should set to **7**	can be transparent
ARC or A	Draws an arc. The default method of drawing arcs is selecting three points (so-called "3 point arc"), which are the two endpoints of the arc and some other point along its locus. Other methods of drawing an arc can be specified by three letters, such as SEA, which means "Start Point, End Point, and Included Angle."	A Included angle C Center point of arc D Direction angle of a line tangent to the arc E Endpoint of arc L Length of chord passing thru both endpoints of the arc R Radius S Start point of arc <RET> uses the end of the last line or arc as the start point for the arc
AREA	Computes the area of any shape by selecting points at the corners of the shape, or by selecting a circle or polyline after typing or picking the "E" option	A sets "Add" mode S sets "Subtract" mode E computes area of selected circle or polyline
ARRAY	Makes multiple copies of selected objects in a rectangular (parallel with the snap rotation) or circular pattern	R rectangular array type P "polar" array type in circular pattern, you must indicate number of items and angle to fill, and whether the objects get rotated with the angle C "circular" array type in a circular pattern, you must indicate angle between items, angle to fill, and whether objects get rotated with the angle
ATTDEF	Creates an attribute definition entity for textual information to be associated with a block definition	I Controls attribute visibility

		C Controls Constant/variable mode
		V Controls verify mode
		P Controls preset mode
ATTDISP	Controls the visibility of attribute entities on a global basis	can be transparent ON Makes all attributes visible OFF Makes all attributes invisible N Visibility of attributes set individually
ATTEDIT	Permits editing of attributes	
ATTEXT	Extracts attribute data from drawings	C CDF comma delimited format extract D DXF format extract S SDF format extract E Extracts attributes from selected entities
AUDIT	Invokes drawing integrity audit	Y Fixes errors encountered N Reports, but does not fix errors encountered
BASE	Specifies origin of current drawing for subsequent insertion into another drawing -- is normally set to point 0,0,0	can be transparent
BHATCH	Fills an automatically defined boundary with a hatch pattern through the use of dialogue boxes. Also allows previewing and replacing adjustments without starting over each time.	
BLIPMODE	Controls display of marker blips for point selection	can be transparent ON Enables temporary marker blips OFF Disables temporary marker blips
BLOCK	Forms a single compound entity from a group of entities within the current drawing	? Lists names of already-defined blocks in the current drawing

BPOLY	Creates a closed polyline of a boundary by picking an open area within a area enclosed by lines, arcs or circles.	
BREAK	Erases part of a line, arc or circle, or splits it into two lines or arcs	F allows respecification of first point
CHAMFER	Creates a chamfer (a angled line connection) at the intersection of two lines	D Sets chamfer distances P Chamfers an entire polyline

CHANGE	A multifunctioning command which can be used to alter the following:
	
	    1. location of individual endpoints of lines
	
	    2. radius of a circle
	
	    3. properties: layer, elevation, thickness, color and linetype of lines, polylines, arcs and circles
	
	    4. style, font, size, location, rotation, and wording of text entities
		P Changes properties of objects
		
		C Color
		
		E Elevation
		
		LA Layer
		
		LT Linetype
		
		T Thickness

CHPROP	Modifies properties of selected objects: layer, thickness, color and linetype -- but not elevation	C Color LA Layer LT Linetype T Thickness
CIRCLE or C	Draws a circle of any size. The default method is to pick a center point and pick a point on the radius or type the radius dimension, but other methods can be selected.	2P Specifies circle by picking 2 points on the diameter 3P Specifies circle by picking 3 points through which the circle will pass D Allows entering the diameter dimension instead of radius dimension TTR Specifies circle by picking two lines, arcs or circles for the circle to be tangent to, and entering the dimension of the radius <RET> Enters radius of circle (the default)

COLOR
 	
or COLOUR
	Sets the current color for subsequently drawn objects (objects drawn after the command is invoked). The default current color is a so-called "logical" color called "BYLAYER" which means that the color of the object drawn will be the color which the current layer has been set to.	can be transparent
		
		                                                                             <number> sets current color to color number entered
		
		    <name> sets current color to color name entered
		
		    BYBLOCK sets floating entity color
		
		    BYLAYER sets current color to whatever color is assigned to the current layer

COMPILE	Compiles shape and font files More than likely, you will never use this command, unless you are creating your own letter design or are creating "shapes" to be used as commonly used symbols in the drawing. These are not typically used in architectural drawings	
CONFIG	Displays options to allow you to reconfigure the video display, digitizer, plotter, and operating parameters. You should not have to use this command, since the configuration of your system has already been done and you will have no need to change it.	
COPY or CP	Draws a copy of selected objects using two methods -- "base point" method, or "displacement" method.	M Allows multiple copies to be made of an object
DBLIST	Lists database information for every entity in the drawing	
DDATTDEF	Displays a dialogue box that allows creation of an attribute definition for textual information to be associated with a block definition	I Controls attribute visibility C Controls Constant/variable mode V Controls verify mode P Controls preset mode

DDATTE	Allows attribute editing via a dialogue box	
DDATTEXT	Displays a dialogue box that extracts data from a drawing.	C CDF comma delimited format extract D DXF format extract S SDF format extract E Extracts attributes from selected entities
DDCHPROP	Displays a dialogue box that modifies the color, layer, linetype, and thickness of selected objects	C Color LA Layer LT Linetype T Thickness
DDEDIT	Allows text and attribute definition editing	Undo Undoes edit to return to the previous value
DDEMODES	Sets current entity properties (layer, linetype, elevation, thickness, and text style) via a dialogue box	can be transparent
DDGRIPS	Allows you to enable grips and set their colors and size via a dialogue box	can be transparent
DDIM	Controls dimensioning variables through a series of dialogue boxes	can be transparent
DDINSERT	Allows you to insert a previously made block or file into the current drawing, select X- and Y- scale factors, rotation angle, pre-explode the block, and select the insertion point, through a dialogue box	
DDLMODES	Sets layer properties and allows you to set the current layer, create a new layer, freeze and thaw layers, freeze and thaw layers in the current viewport, assign or reassign colors and linetypes to layers, through a dialogue box	can be transparent
DDOSNAP	Allows you to set running OSNAPS and set the size of the target box aperture through a dialogue box.	can be transparent

DDRMODES	Allows you to set drawing aids variables through a dialogue box	can be transparent
DDRENAME	Allows you to rename layers, text styles, linetypes, blocks, views, User Coordinate Systems, viewport configurations, and dimension styles	
DDSELECT	Allows you to set entity selection modes, size of the pickbox, and entity sort method through a dialogue box	can be transparent
DDUCS	Allows you to save the current User Coordinate System and give it a name for future retrieval, or to select a UCS that is already saved	
DDUNITS	Allows you to set the unit type (Architectural or Decimal), angle display format (degrees-minutes-seconds, or decimal degrees), and precision of the units,	can be transparent
DELAY	This command is used with SCRIPT files. It will delay the execution of the next command for the time in milliseconds specified after the DELAY command.	can be transparent
DIM	Takes you out of the drawing editor and into the semi-automatic dimensioning program built into AutoCAD. Will display a **Dim:** prompt on the command prompt line. To get back to the drawing editor and the **Command:** prompt, type the <Esc> key.	
DIM1	Same as above, but only allows one dimensioning command to be used, and then takes you automatically back to the drawing editor and the Command: prompt.	
DIM ALIGNED	**Dimensioning sub-command:** **Draws a linear dimension with the dimension line parallel to the selected dimension origin points. This lets you align a dimension with an angled line that is not either exactly horizontal or vertical.**	
DIM	**Dimensioning sub-command:**	

ANGULAR	**Draws an arc and calculates the angle between two non-parallel lines, and draws the text and arrowheads. Arrowheads are the standard AutoCAD filled triangles, regardless of your defined Arrow Block.**	
DIM BASELINE	**Dimensioning sub-command:** **Continues a linear dimension from the baseline (the so-called "first extension line") of the previous or selected dimension. This is not frequently used in architectural dimensions. The spacing between each dimension line is controlled by the AutoCAD variable DIMDLI ("Dimension Line Increment"), which you should set at 0 normally, but if you use the baseline command, you should reset to 1/8".**	
DIM CENTER	**Dimensioning sub-command:** **Draws a pair of crossed lines at the center of a circle or arc on the current layer**	
DIM CONTINUE	**Dimensioning sub-command:** **Continues a linear dimension from the second extension line of the previous or selected dimension. It is used to create a so-called "string" of dimensions. This is used quite a bit in architectural drawings.**	
DIM DIAMETER	**Dimensioning sub-command:** **Draws a dimension through the center of a circle or arc, calculating the diameter, with one of the arrow heads located at the point on the circle or arc which is picked.**	

DIM EXIT	**Dimensioning sub-command:** **Exits the dimensioning program and returns from the Dim: prompt to the normal Command: prompt**	
DIM HOMETEXT	**Dimensioning sub-command:** **Restores the text of an associative dimension to its default (home) location if you have moved it.**	
DIM HORIZONTAL	**Dimensioning sub-command:** **Draws a horizontal linear dimension line**	
DIM LEADER	**Dimensioning sub-command:** **Draws an a line or series of lines with an arrow head (commonly called a "leader") to point to an object to notate it. This command will also prompt you for the note at the end of the leader line, but it will allow only one line of text. The most useful method of using this command is to simply draw leaders between the object to be notated and text that you create with the DTEXT command.**	
DIM NEWTEXT	**Dimensioning sub-command:** **Permits changing text of an associative dimension without exploding it. However, once the text has been changed, the dimension will no longer be associative.**	
DIM OBLIQUE	**Dimensioning sub-command:** **Adjusts the obliquing angle of a linear associative dimensionsâ extension lines. Useful in dimensioning isometric drawings.**	

DIM ORDINATE	**Dimensioning sub-command:** **Draws the X and Y coordinates within a box with a leader line attached to the point, for a selected point. Automatically calculates the coordinates.**		
DIM OVERRIDE	**Dimensioning sub-command:** **Overrides the dimension variable settings on a selected dimension.**		
DIM RADIUS	**Dimensioning sub-command:** **Draws the radius dimension of an arc or circle from the center to the point on its locus where the arc or circle was selected.**		
DIM REDRAW	**Dimensioning sub-command:** **Redraws the viewport, just like the regular REDRAW command.**		
DIM RESTORE	**Dimensioning sub-command:** **Makes a dimension style current**		
DIM ROTATE	**Dimensioning sub-command:** **Draws a linear dimension with the distance measured origin points, but with the actual dimension line set at a different angle.**		
DIM SAVE	**Dimensioning sub-command:** **Saves the currently set dimension variables under a dimension style name for future retrieval through the RESTORE command.**		
DIM STATUS	**Dimensioning sub-command:** **Lists all dimension variables with their current status.**		

DIM STYLE	**Dimensioning sub-command:** **Switches to a new Text Style for all future dimensions.**	
DIM TEDIT	**Dimensioning sub-command:** **Allows moving text in associative dimensions without having to explode the dimensions.**	
DIM TROTATE	**Dimensioning sub-command:** **Allows rotation of the text of several associative dimensions at once.**	
DIM UNDO	**Dimensioning sub-command:** **Undoes any changes made by the most recent dimensioning command.**	
DIM UPDATE	**Dimensioning sub-command:** **Changes selected associative dimensions to use the current dimension variables, the current text style, and the current units setting.**	
DIM VARIABLES	**Dimensioning sub-command:** **Lists the settings of dimension variables associated with a particular dimension style.**	
DIM VERTICAL	**Dimensioning sub-command:** **Draws a vertical linear dimension line**	
DIST	Finds distance between two points	can be transparent
DIVIDE	Places points along a line, polyline, arc, or circle, dividing it into the specified number of equal parts	B Uses a specified Block to divide the object instead of a point
DONUT	Draws filled rings with specified inside	

or DOUGHNUT	and outside diameters	
DRAGMODE	Allows control of dynamic dragging feature for copy and move commands	can be transparent ON Makes dragging of object visible OFF Disables visual dragging A Auto mode -- enables dragging when possible
DTEXT	Draws text "Dynamically" so that as you enter text letters or numbers, they are visible on the screen	J Prompts for Justication options S Allows you to select text styles that are already created A Aligns text between two points, with style-specified width factor, AutoCAD adjusts height to keep letter proportion as designated in the style. C Centers text horizontally F Fits text between two points, with a specified height, AutoCAD adjusts width of letters to fit M Centers text horizontally and vertically R Right-justifies text BL Bottom-left justification BC Bottom-center justification BR Bottom-right justification ML Middle-left justification MC Middle-center justification MR Middle-right justification TL Top-left justification

		TC Top-center justification
		TR Top right justification

DVIEW	Allows perspective viewing of drawing	CA Sets CAmera angle relative to the Target
		    T Sub-option, toggles between angle in the X-Y plane, and angle from the X-Y plane
		CL Sets Front and Back CLipping planes
		
		D Sets Camera to Target distance and turns on Perspective viewing
		
		H Removes hidden lines on the selection set
		
		OFF Turns Perspective viewing off
		
		PA PAns drawing across the screen
		
		PO Allows specification of Camera and Target points
		
		TA Rotates the TArget point about the Camera
		    T Sub-option, toggles between angle in the X-Y plane, and angle from the X-Y plane
		TW Twists the view around the line of sight
		
		U Undoes a DVIEW sub-command
		
		X Exits the DVIEW command
		
		Z Sets lens length

DXBIN	Inserts specially coded binary files into the drawing	
DXFIN	Loads a drawing interchange file (DXF file) to be able to read a drawing file created by another CAD program	
DXFOUT	Writes a drawing interchange file (DXF	B Writes binary DXF file

	file) to be able to use an AutoCAD drawing in another CAD program	E Outputs selected entities only (0-16) Floating point precision
EDGESURF	Constructs a 3d Polygon mesh approximating a Coons surface patch interpolated between 4 adjoining edges	
ELEV	Sets Elevation (distance above the ground plane) and Thickness (extruded distance from the bottom of an object to the top) for subsequently drawn entities	
ELLIPSE	Draws ellipses	C Allows specification of Center point of ellipse rather than first axis endpoint R Allows specification of eccentricity rotation rather than length of second axis I Draws Isometric circle in current isoplane
END	Saves the drawing and exits AutoCAD back to DOS or WINDOWS	
ERASE or E	Erases selected entities from the drawing	
EXPLODE	Separates a block, dimension or hatch pattern into its constituent entities or makes a polyline into a series of straight lines. In the case of a block that is exploded, if it was originally drawn on the 0 layer, it returns to that layer, regardless of the layer it was inserted on, and it loses its referential connection to the original block. In the case of a dimension or hatch pattern that has been exploded, their parts go back to the 0 layer, and are assigned the logical color (BYBLOCK) regardless of the layer they were drawn on. In the case of an exploded polyline, it loses any width it may have had.	
EXTEND	Lengthens a line, arc, or polyline to meet a specified "boundary edge"	U Undoes effect of last Extend command
FILES	Allows DOS disk and file utility tasks to	can be transparent

	be done through a dialogue box	
FILL	Controls whether solids, traces, wide polylines, and donuts will be filled with ink. Note that fill is only visible when viewing these entities in plan (perpendicular to line of sight)	can be transparent ON entities visually filled with solid ink OFF entities not visually filled but are shown in outline only
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	P Fillets an entire Polyline at the vertices R Allows setting of the fillet radius. Default value is 0. Radius remains set until changed again
FILMROLL	Generates a file for rendering by AutoShade, a separate program	
GRAPHSCR	A "toggle." Flips the display on single-screen systems (most of them) from text-screen to graphics-screen, and vice-versa.	can be transparent

GRID	Displays a rectangular grid of white dots on the screen at a specified X- and Y-spacing	can be transparent
		
		ON Turns grid display on
		
		OFF Turns grid display off
		
		S Locks grid spacing to SNAP resolution
		
		A Sets grid aspect (different X- and Y- spacings
		        \<number\> Sets grid spacing for both X- and Y- coordinates
		
		        \<number X\> Sets grid spacing to multiple "X" of snap spacing

HANDLES	Assigns a unique, permanent number to each entity of the drawing	ON Enables Handles DESTROY Discards all entity Handles
HATCH	Allows drawing of cross-hatching and pattern filling	\<pattern-name\> uses hatch pattern name from library file to

		fill designated area with a pseudo-block hatch entity -- can be reduced to individual lines and points by EXPLODEing it.
		
		<*pattern-name> uses hatch pattern name from library file to fill designated area with individual lines and points, not a block
		
		U "User-defined" hatch pattern drawn of parallel lines with a specified distance between, at a specified angle, and either single-hatching or double (cross)-hatching
		
		? Lists names of available hatch patterns
		
		<pattern-name> or U can be followed by a comma and the following sub-commands:
		    I Ignores any boundaries inside the outermost boundary
		
		    N Normal style of selecting boundaries with alternating hatched and unhatched nested areas
		
		    O Hatches outermost boundary area only

HELP or ?	Displays a list of all valid commands and data entry options, or obtains help for a specific command or prompt	can be transparent if invoked while another command is in process, it will assume that you want help on that command
HIDE	Regenerates a 3d visual image of the drawing with hidden lines removed	
ID	Displays the X,Y, and Z coordinates of a point selected	can be transparent
IGESIN	Loads an IGES interchange file	
IGESOUT	Writes an IGES interchange file	
INSERT	Inserts a previously created block or	<filename> Inserts a

		filename from the default directory into the current drawing, assuming that there is no block by that name in the current drawing
drawing file into the current drawing		
		<blockname>=<filename> Creates a block in the current drawing with the same name as the file
		<*name> Inserts a block in its constituent entities rather than as a block -- also called a "Star Block" or Pre-exploded block
		C Reply to X scale prompt -- allows scale to be specified by two points (Corner specification of scale)
		XYZ Reply to X scale prompt -- allows specification of X, Y, and Z scales
		~ Displays a file dialogue box
		? Lists names of defined blocks within the current drawing
ISOPLANE	Selects the plane of an isometric grid to be the current plane for an orthogonal drawing	can be transparent L Left plane R Right plane T Top plane <RET> Toggle to next plane in series, Left, Right, and Top
LAYER or LA	Creates named drawing layers, assigns color and linetype properties to those layers, allows layers to be frozen and thawed, locked and unlocked, and allows current layer to be set	can be transparent C Assigns color to named layer F Freezes named layers (comma separates a list of layers to be frozen, * can be used as a wildcard) -- current layer cannot be frozen

		L Assigns linetype to named layer LO Locks named layer M Makes a layer -- creates a new layer and makes it the current layer N Creates new layers with named names (comma separates a list of layers to be created, * can be used as a wildcard) ON Turns named layers on if they were turned off (comma separates a list of layers to be turned on, * can be used as a wildcard) OFF Turns named layers off if they were turned on (comma separates a list of layers to be turned off, * can be used as a wildcard)
LAYER or LA (Continued)		S Sets named layer as the current layer T Thaws named layers which are frozen (comma separates a list of layers to be frozen, * can be used as a wildcard) U Unlocks named layer ? Gives information about named layers, * can be used as a wildcard
LIMITS	Changes the imaginary boundaries of the drawing, and controls whether drawing can be made outside of the boundaries. If Limit-checking is ON, drawing cannot be done outside of limits.	can be transparent <2 points> Sets lower-left and upper -right drawing limits ON Enables limit-checking OFF Disables limit-checking
LINE or	Draws straight lines	<RET> In reply to **From Point**: prompt, line begins at end of previous line or

L		arc C In reply to **To point:** prompt, closes the polygon back to first "From Point" U In reply to **To point:** prompt, undoes last line segment
LINETYPE	A linetype is a series of lines and spaces -- dots and dashes -- Sets the current linetype to be used for all subsequent entities to be drawn, allows creation of new linetype definitions, and allows loading of previously created linetypes from DOS files	can be transparent ? Lists a linetype library , or named linetypes which are loaded C Creates a linetype definition L Loads a previously creatred linetype definition S Set current linetype -- note that the current linetype should always be set to the logical linetype "BYLAYER"
LIST	Lists all information about selected object, such as what type of entity it is (Line, Arc, Circle, Block, etc.), what layer it is on, what color it is, the location of its endpoints, what its Z-axis (elevation), what its thickness is, etc.	
LOAD	Loads a file of user-defined shapes to be used with the Shape command	? Lists the names of loaded Shape files
LTSCALE	Sets scale factor to be applied to all linetypes within the drawing	can be transparent
MEASURE	Places points (or, optionally, Blocks) at intervals along a selected line, polyline, arc or circle. The interval distance is given by the user. If points are used as the marker to be placed along the entity, they are not visible unless the Point type is set to type 3 with the PDMODE command.	B specifies that a defined Block is to be used as a marker instead of a point.
MENU	Loads a Menu file into the menu areas (Screen, pull-down, tablet, and button)	
MINSERT	Inserts multiple copies of a Block in a rectangular pattern (word stands for "Multiple Insert." The entire array created then becomes a Block.	fname Loads fname and forms a rectangular array of the resulting Block fname=f Creates Block

		fname from file f and forms a rectangular array
		? Lists names of defined Blocks within the drawing
		C (as a reply to X scale prompt) specifies scale via two selected points ("Corner" specification of scale)
		XYZ (as a reply to X scale prompt) readies Minsert for X, Y, and Z scales)
		- displays a file dialogue box
MIRROR	Reflects designated entities about a user-specified axis	
MOVE or M	Moves designated entities to another location	
MSLIDE	Makes a slide from the current display. Slide shows the drawing only, and not the UCSICON, the Toolbar, the Aerial View or menus.	
MSPACE	Switches to Model Space	
MULTIPLE	Causes the next command to repeat until cancelled	
MVIEW	Creates and controls Viewports	ON Turns selected Viewports on and causes the Model to be regenerated in the selected Viewports. OFF Turns selected Viewports off and causes Model to be not displayed in the selected Viewports. Hideplot Causes hidden lines to be removed in selected Viewports during Paper Space plotting. Fit Creates a single

Viewport to fit the current Paper Space view

2 Creates 2 Viewports in specified area or fit to the current Paper Space view.

3 Creates 3 Viewports in specified area or fit to the current Paper Space view.

4 Creates 4 Viewports in specified area or fit to the current Paper Space view.

Restore Translates viewport configurations saved with the VPORTS command into individual Viewport entities in Paper Space.

<first Point> Creates a new Viewport within the area specified by 2 points.

Sub-options to the "2" Option:

Horizontal Creates a horizontal division between the 2 new Viewports

Vertical Creates a vertical division between the 2 new Viewports.

Sub-options to the "3" Option:

Above Specifies placement of larger Viewport above the other 2

Below Specifies placement of larger Viewport below the other 2

Horizontal Splits area into

		thirds horizontally
		Left Specifies placement of larger Viewport to the left of the other 2
		Right Specifies placement of the larger Viewport to the right of the other 2
		Vertical Splits area into thirds vertically
NEW	Creates a new drawing. When selected from a menu or typed in at the Command: prompt, this command brings up a dialogue box which allows setting a name for the new drawing by typing the name in the box, selection of a "prototype" drawing or typing a name of the new drawing and then an = sign and then the name of a drawing to be used as a prototype.	
OFFSET	Creates a new line, polyline arc or circle parallel to the entity and at a specified distance from it.	<number> specifies offset distance T "Through" allows specification of a point through which the offset line, polyline, arc or circle is to pass
OOPS	restores erased entities, or entities made into a Block	
OPEN	Opens an existing drawing	
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.	can be transparent
OSNAP	Enables points to be precisely located on reference points of existing objects. This is the so-called "Running Mode" of OSNAP, which sets selection method to run continuously until set to NON (none)	can be transparent CEN CENter of arc or circle

	or until overridden by selecting another "Interrupt Mode" OSNAP method from the cursor menu. Combinations of OSNAP methods can be used by selecting a series of options separated by commas. For instance, if you want ot always pick either endpoints or intersection points when locating endpoints of lines, you would issue the command as follows: **OSNAP <RET> END,INT <RET>**	END closest ENDpoint of arc or line INS INSertion point of Text or Block INT INTersection of line, arc, or circle MID MIDpoint of line, arc, rectangle side, or polygon side NEA NEArest point selected by aperture on line, polyline, arc, or circle NOD NODe (another name for a Point) NON NONe -- used when a "Running OSNAP" is on to temporarily turn off OSNAP selection PER PERpendicular point to line, arc or circle -- when used with an arc or circle it will draw a line to the surface of the arc or circle heading toward the center point QUA QUAdrant point of arc or circle (top, bottom, right or left side) QUI QUIck mode -- this is a modifier to one of the other OSNAP options -- it will find the first point that meets the requirements, not necessarily the closest point to the aperture. TAN TANgent point to arc or circle
PAN	Moves the display window without changing the magnification factor	can be transparent
PEDIT	Allows editing of polylines which are already drawn	C Closes an open polyline

			D Decurves, or returns a Spline curve to its control frame or series of connected straight lines
			E Edit vertices
			F Fits curve to a polyline -- makes a series of straight lines into a curve which will pass through the vertices
			J Joins a line or arc or another polyline to an open polyline
			L Toggles linetype generation to be either a continuous pattern of dashes passing through the vertices, or a pattern which starts and ends at each vertex
			O Opens a closed polyline
			S Uses the polyline vertices as a frame for a Spline Curve -- type of Spline Curve is set by the Variable "Splinetype."
			U Undoes one editing operation
			W Sets a uniform width for the Polyline
PEDIT (Continued)			Vertex Editing Options:
			B Sets first vertex for Break
			G "Go" -- performs a Break or Straighten operation
			I Inserts a new vertex after current one
			M Moves current vertex
			N Makes the Next vertex current
			P Makes the Previous vertex current
			R Regenerates the Polyline

			S Sets first vertex for Straighten
			T Sets tangent direction for current vertex
			W Sets new width for the following segment
			X Exits vertex editing, or cancels Break and Straighten
PEDIT Mesh	Allows editing of Meshes which are already drawn		D De-smooth -- restores original mesh
			E Edits mesh vertices
			M Opens (or closes) the mesh in the M direction
			N Opens (or closes) the mesh in the N direction
			S Fits a smooth surface as defined by the Variable SURFTYPE
			U Undoes one editing operation
			X Exits Pedit command
			Vertex Editing Options:
			D Makes previous vertex Down in M direction current
			L Makes previous vertex to Left in N direction current
			M Moves the marked vertex
			N Makes the Next vertex current
			P Makes the Previous vertex current
			R Makes next vertex to Right in N direction current
			RE Redisplays the polygon mesh

		U Makes the next vertex Up in M direction current X Exits vertex editing
PFACE	Creates a 3D mesh of arbitrary complexity and surface characteristics	
PLAN	Puts the display in "Plan" view, with Vpoint set to 0,0,1, relative to either the current UCS, a specified UCS, or the World Coordinate System	C Establishes the Plan view of the Current UCS U Establishes the Plan view of a specified UCS W Establishes the Plan view of the World Coordinate System
PLINE	Draws a 2D polyline, which are connected line or arc segments with user-determined width and taper	H sets the half-width of the polyline segments U Undoes previous segment W sets the width of polyline segments <RET> Exits the Pline command The following options are available only as long as you are drawing in Line mode: A Switches to the Arc mode to allow integration of arcs into the polyline C Closes the polyline with straight segment L Segment length (continues previous segment) The following options are available only as long as you are in the Arc mode: A Included angle CE Center Point CL Closes with arc segment D Starting direction

		L Chord length, or switches to line mode R Radius S Second point of 3-point arc
PLOT	Plots a drawing to a plotting device, a printer, or to a "Plot File."	
POINT	Draws single points. Appearance of the points is set by the Variable PDMODE	
POLYGON	Draws regular polygons with a specified number of sides. Polygons are Polyline entities.	E Specifies size and rotation of polygon by picking endpoints of one edge C Circumscribes polygon around a circle I Inscribes polygon within a circle
PSDRAG	Controls the appearance of an imported Postscript image that is being "dragged" (that is, positioned and scaled) into place by the PSIN command	0 Only the imageâs bounding box is displayed as you drag it into place 1 The complete rendered PostScript image is displayed as you drag it into place
PSFILL	Fills 2D Polyline outlines with PostScript fill patterns defined in the ACAD.PSF file	. Entering a period results in no fill pattern ? Lists the fill patterns defined in ACAD.PSF file
PSIN	Imports Encapsulated PostScript (EPS) files	
PSOUT	Exports the current view of your drawing to an Encapsulated PostScript (EPS) file	
PSPACE	Switches to Paper Space -- this will work only if the Variable TILEMODE is set to 0.	
PURGE	Removes unused Blocks, Text Styles, Layers, Linetypes, and Dimension Styles from current drawing	A Purges All unused named objects B Purges unused Blocks

		D Purges unused Dimstyles
		LA Purges unused LAyers
		LT Purges unused Linetypes
		SH Purges unused Shape files
		ST Purges unused Text Styles
QSAVE	Saves the current drawing "Quickly" without requesting a filename (as long as file has already been given a name)	
QTEXT	Controls display of text -- command stands for "QuickTEXT"	ON Lines of text displayed as rectangles OFF Lines of text displayed as text
QUIT	Exits AutoCAD -- if the current drawing has not been Saveds in its current state, a dialogue box will appear asking if you want to Save the drawing, Discard the changes, or Cancel the Exit command	
RECOVER	Attempts to recover damaged or corrupted drawings	
REDEFINE	Restores a built-in command which may have been deleted by the command UNDEFINE	
REDO	Reverses a previous command if it was U or UNDO	
REDRAW	Refreshed or cleans up the current Viewport	can be transparent
REDRAWALL	Redraws all viewports	can be transparent
REGEN	Regenerates the current Viewport by recalculating the vector locations of all endpoints	
REGENALL	Regenerates all Viewports	
REGENAUTO	Controls automatic regeneration performed by other commands	can be transparent ON Allows automatic regens OFF Prevents automatic regens

REINIT	Allows the I/O ports, the digitizer, the display, the plotter, and the ACAD.PGP file to be reinitialized. This command allows selection of what you want to reinitialize through a dialogue box. It is most useful when you make changes "on the fly" to the ACAD.PGP file and want to immediately make use of those changes without having to exit AutoCAD and get back into it.	
RENAME	Changes the names associated with Text Styles, Layers, Linetypes, Blocks, Views, User Coordinate Systems, Viewport Configurations, and Dimension Styles.	B Renames Blocks D Renames Dimension Styles LA Renames Layers LT Renames Linetypes S Renames Text Styles U Renames UCS VI Renames Views VP Renames Viewports
RESUME	Resumes an interrupted command script	can be transparent
REVSURF	Creates a 3D polygon mesh approximating a surface of revolution, by rotating a curve around a selected axis	
ROTATE	Rotates existing objects parallel to the current UCS	
RSCRIPT	Restarts a command script from the beginning	
SAVE	Requests a filename and saves the drawing	
SAVEAS	Same as SAVE, but also renames the current drawing and keeps the new name current	
SCALE	Alters the size of existing objects	R Resizes with respect to a reference size
SCRIPT	Runs a command script. A script is a an ASCII file which contains various	can be transparent

	commands that when executed operates just as if you typed in those commands, one by one, by hand	
SELECT	Groups objects into selection sets for use in subsequent commands	
SETVAR	Allows you to display or change the setting of system variables. This command is no longer necessary to change the value of system variables. You now can type in the name of the variable to change it.	can be transparent ? Lists specified (or all) system variables and their settings
SH	Short for "SHell." Allows access to internal operating system (DOS) commands. Once SH is invoked, followed by two <RET>s, the DOS Prompt will be displayed, so you can enter any DOS command. To get back to AutoCAD, simply type EXIT at the DOS prompt.	
SHADE	Shades 3D model in current Model Space Viewport	
SHAPE	Draws predefined Shapes	? Lists available Shape names
SHELL	Allows access to internal operating system (DOS) commands. Once SH is invoked, followed by two <RET>s, the DOS Prompt will be displayed, so you can enter any DOS command. To get back to AutoCAD, simply type EXIT at the DOS prompt.	
SKETCH	Permits freehand sketching simulation	C Connect: restarts SKETCH at endpoint E Erases (backs up over) temporary lines P Raises/lowers sketching pen Q Discards temporary lines, remains in SKETCH R Records temporary lines, remains in SKETCH X Records temporary lines, exits SKETCH . Draws line to current point

SNAP	Specified a round-off interval for point entry so that entities can be placed at precise locations	can be transparent
		
		       <number> sets snap alignment resolution
		
		       ON aligns designated points
		
		OFF does not align designatged points
		    A sets aspect ratio (differing X and Y spacing)
		
		    R Rotates the snap grid and crosshairs, and turns on SNAP after, if not already turned on
		
		    S Selects either Standard (rectangular) or Isometric Snap grid

SOLID	Draws filled polygons. Fill can be turned ON or OFF with the FILL command	
STATUS	Displays drawing statistics and modes on the Text Screen	can be transparent
STRETCH	Allows moving a portion of a drawing while retaining their connections to other parts of the drawing. You cannot stretch Blocks, Hatch patterns, or Text entities, however.	
STYLE	Creates named Text styles, with user-selected combinations of font, height, mirroring, obliging, and horizontal scaling.	can be transparent ? Lists specified (or all) currently defined text styles

TABLET	Aligns the digitizing tablet with coordinates of a paper drawing to accurately copy it with AutoCAD	ON Turns Tablet Mode ON (but it has to be CALibrated first)
		
		   OFF Turns Tablet Mode OFF
		
		   CAL Calibrates Tablet for use -- sets the distance to be used to copy the drawing to an accurate scale
		
		   CFG Configures tablet menus and pointing area

TABSURF	Creates a polygon mesh approximating a general tabulated surface defined by a path and a direction vector	
TEXT	Draws text characters of any size with selected styles	J Prompts for justification options
		S Lists or selects text style
		A Aligns text between two points, with style-specified width factor, AutoCAD computes approximate height proportional to length of text line
		C Centers text horizontally about a defined point
		F Fits text between two points, with specified height, AutoCAD computes approximate width factor to fill the distance between the two points
		M Centers text horizontally and vertically about a defined point
		R Right-justifies text
		BL Bottom Left justification
		BC Bottom Center justification
		BR Bottom Right justification
		ML Middle Left justification
		MC Middle Center justification
		MR Middle Right justification
		TL Top Left justification
		TC Top Center justification
		TR Top Right justification
TEXTSCR	A "toggle." Flips the display on single-screen systems (most of them) from text-screen to graphics-screen, and vice-versa.	can be transparent

TIME	Displays drawing creation and update times, and permits control of an elapsed timer.	can be transparent D Displays current times ON Starts user elapsed timer OFF Stops user elapsed timer R Resets user elapsed timer
TRACE	Draws solid filled lines of specified width. Has mostly been superseded by the PLINE command.	
TREESTAT	Displays information on the drawing's current spatial index, such as the number and depth of nodes in the drawing database.	
TRIM	Erases a portion of selected entities that cross a specified "cutting edge"	U Undoes last trim operation
U	Reverses the effect of the previous command	
UCS	Defines or modifies the current User Coordinate System	D ("Delete") Deletes one or more saved coordinate systems E ("Entity") Sets a UCS with the same extrusion direction as that of the selected entity O ("Origin") Shifts the origin of the current coordinate system P ("Previous") Restores the Previous UCS R ("Restore") Restores a previously saved UCS S ("Save") Saves the current UCS V ("View") Establishes a new UCS whose Z-axis is parallel to the current viewing direction W ("World") Sets the current UCS to the World Coordinate System

			X Rotates the current UCS around the X-axis
			Y Rotates the current UCS around the Y-axis
			Z Rotates the current UCS around the Z-axis
			ZA Defines a UCS using an origin point and a point on the positive portion of the Z-axis
			3 Defines a UCS using an origin point, a point on the positive portion of the X-axis, and a point on the positive Y-portion of the XY plane
			? Lists specified saved coordinate systems
UCSICON	Controls visibility and placement of the User Coordinate System Icon, which graphically indicates the origin and orientation of the current UCS. The options normally affect only the current viewport.		A Changes settings of all active viewports

N ("NoOrigin") Displays the UCSICON at the lower-left corner of the viewport (i.e., not at the Origin)

OR ("ORigin") Displays the UCSICON at the origin of the current UCS, if possible (if the origin point is not within the current viewport, the location of the UCSICON defaults to the lower-left corner of the viewport)

OFF Turns the UCSICON off

ON Turns the UCSICON on |
| UNDEFINE | Deletes the definition of a built-in AutoCAD command | | |
| UNDO | Reverses the effect of multiple commands, and provides control over the "Undo" facility | | <number> Undoes the <number> of most recent commands in reverse order
A ("Auto") Controls treatment of menu items as UNDO "Groups"

B ("Back") Undoes back to |

		previous UNDO "Mark"
		C ("Control") Enables/disables the UNDO feature
		E ("End") Terminates an UNDO Group (a sequence to be treated as one command)
		G ("Group") Begins an UNDO Group (a sequence to be treated as one command)
		M ("Mark") Places "marker" in UNDO file (for use with "Back")
		Control Sub-Options
		All Enables the full UNDO feature
		None Disables U and UNDO entirely, and discards any previous UNDO information saved earlier in the editing session
		One Limits U and UNDO to a single operation
UNITS	Selects coordinate and angle display formats and precision	can be transparent
VIEW	Saves the current graphic display and space as a named view, or restores a saved view and space to the display	can be transparent
		D Deletes a named view
		R Restores named view to screen
		S Saves current display as named view
		W Saves specified Window as named view
		? Lists specified named views
VIEWPORTS or	Used in Model Space only, and has been superseded by Paper Space Vports, created with the MVIEW	D Deletes a saved viewport configuration

VPORTS	command. Divides the AutoCAD graphics display into multiple viewports, each of which can contain a different view of the current drawing (maximum of 4 "tiled" viewports per viewport)	J Joins (merges) two viewports R Restores a saved viewport configuration S Saves the current viewport configuration SI ("SIngle") Displays a SIngle viewport filling the entire graphics area 2 Divides the current viewport into 2 viewports 3 Divides the current viewport into 3 viewports 4 Divides the current viewport into 4 viewports ? Lists the current and saved viewport configurations
VIEWRES	Allows you to control the precision and speed of circle and arc drawing on the monitor by specifying the number of sides in a circle. Acts like an AutoCAD variable. Recommend that it be set to 2000.	
VPLAYER	Sets viewport visibility for new and existing layers	? Lists layers frozen in a selected viewport F ("Freeze") Freezes specified layers in selected viewports T ("Thaw") Thaws specified layers in selected viewports R ("Reset") Resets specified layers to their default visibility N ("Newfrz") Creates new layers that are frozen in all viewports V ("Vpvisdflt") Sets the default viewport visibility for existing layers
VPOINT	Selects the viewpoint for a 3D visualization	R Selects viewpoint via two rotation angles

			<RET> Selects viewpoint via a graphic compass and axis tripod which appears on the screen <x,y,z> Specifies x, y, and z relative coordinates of viewpoint
VSLIDE	Displays a previously created slide file		<filename> views slide with name of "filename.SLD" *filename preloads slide which the next command VSLIDE will view
WBLOCK	Writes selected entities or previously created Block to a drawing file		<filename> writes specified entities to "filename.DWG" = filename.DWG to be created is same as the entities included in a previously defined Block by the same name * writes entire drawing to the filename.DWG and purges all unreferenced blocks, layers, linetypes, text and dimensioning styles <RET> writes selected objects
XBIND	Makes portions of an Xref a permanent part of your drawing		B ("Block") Makes a block in the Xref a permanent part of your drawing D ("Dimstyle") Makes a dimstyle in the Xref a permanent part of your drawing L ("Layer") Makes a layer in the Xref a permanent part of your drawing LT ("LType") Makes a Ltype in the Xref a permanent part of your drawing

			S ("Style") Makes a text style in the Xref a permanent part of your drawing
XREF	Allows you to work with other AutoCAD drawings without adding them permanently to your drawing and without altering their contents		A ("Attach") "Attaches" a new Xref to the drawing or updates an Xref that is already attached
			B ("Bind") Makes an Xref a permanent part of your drawing
			D ("Detach") Removes an Xref from your drawing
			P ("Path") Allows you to view and edit the filename path (directory and sub-directory) that AutoCAD searches when loading a particular Xref
			R ("Reload") Updates one or more Xrefs at any time
			? Lists Xrefs in your drawing and the drawing associated with each one
ZOOM	Enlarges or reduces the display magnification of the drawing, without changing the actual size of the entities	can be transparent	<number> multiplier from original magnification
			<number X> multiplier from current magnification
			<number XP> multiplier of magnification relative to paper space -- used for plotting to get right plot scale in each viewport
			A ("All") fills limits of drawing to screen
			C ("Center") makes picked point the center of the screen
			D ("Dynamic") makes an adjustable rectangular lens appear on the screen which is capable of being made smaller or larger and moved to different positions over the drawing and

| | | | once set by the user, the drawing will quickly zoom to the location and magnification set for the lens. This sub-command is no longer useful because all computers have very fast zooms naturally now.

E ("Extents") makes the farthest edges of the actual visible drawing fill up the graphics screen

L ("Lower-Left") makes the point picked become shoved to the lower-left corner of the graphics screen

P ("Previous") zooms back to whatever the last zoom, previous to the current zoom was -- AutoCAD stores about 10 of these, so you can walk backward in zoom magnification 10 times

V ("Virtual Screen") makes the largest area available to the graphics card fill the graphics screen -- this varies with the quantity of graphics RAM that your graphics card has

W ("Window") asks you to pick the lower left corner and the upper right corner of a zoom window and then fits that window to the graphics screen |
| 3DFACE | Draws flat planes in 3D space with either 3 corners or 4 corners (3 sided or 4 sided flat planes). These are used to hide objects behind them in space to create the illusion of a solid object. 3DFACEs can also be shaded using the SHADE command, in which case the entire plane takes on the color assigned to its edges. Normally (without using the HIDE or SHADE commands, 3DFACEs are transparent, which means that objects behind them in space are visible through them, and they appear as "wireframes." | | I ("Invisible") this is used by typing in the letter "I" just before picking a corner point for the 3DFACE, and it will make the edge of the 3DFACE drawn from that point to the next one "Invisible" (that is, it will not be shown as a line on the drawing). This is important when joining several 3DFACEs together to cover a complex, multi-faceted surface, where you do not want the "seams" of the 3DFACEs to show in the final drawing. The visibility of edges of 3DFACEs can further be turned on or off by the AutoCAD variable |

		"SPLFRAME." When SPLFRAME is set to 1 ("on"), both edges which have been made "Invisible" as well as normal, visible edges are shown on the drawing. When SPLFRAME is set to 0 ("off"), Invisible edges are not shown. The purpose of sometimes wanting to see invisible edges is that you may want to move, stretch, or copy the 3DFACE and if its edges are invisible, you cannot select them fopr modification. You can only select a 3DFACE by picking on a visible edge.
3DMESH	Draws a 3D polygon mesh by specifying its size, in terms of the number of faces in each direction (called M and N directions), and the location in 3D space of each vertex (the corner of a face) in the mesh.	
3DPOLY	Creates a 3D polyline	C Closes the polyline back to the first point U Undoes (deletes) the last segment entered <RET> Exits 3DPOLY command