AUTOCAD AT A GLANCE

S.N	Command	Description
1.	Starting AutoCAD	From the start menu choose programs. Choose AutoCAD from the menu. Then startup dialog box is displayed. The dialog box provides you with four ways to start drawing. Choose one of the following: **Open a drawing** : Opens a drawing you select from the list of four most recently opened drawings. Also, displays the browse button that you choose to look for another file. **Start from scratch** : Opens a new drawing based on the measurement system you choose- English (inches) of metric(millimeters). **Use a template** : Opens a new drawing based on a template you select from a list. The list displays template files (.dwt extension) that exist in the drawing template file location as specified in the Options dialog box. Template file store all the settings for a drawing and can also include predefined layers, dimension styles, and views. **Use a wizard** : Opens a new drawing that you setup using either the Quick Setup wizard or the Advanced Setup wizard.
2.	AutoCAD Window	The AutoCAD window is your design workspace. It contains elements that you see to create your design and to receive information about them. The main parts of AutoCAD window are : **Menu bar** : Contains the default AutoCAD menus. Menus are defined by menu files that you can modify or design on your own. **Standard toolbar** : Contains frequently used buttons such as Redraw, Undo and Zoom, as well as Microsoft Office standard buttons such as Open, Save, Print and Spell. Buttons with small black triangles in lower right corner have flyouts containing tools that invoke commands related to first tool shown. Click and hold down the first button to display the flyout. **Object properties toolbar** : Sets object properties such as color, linetype and lineweight and manages layers. **Draw and modify toolbars** : Provide access to common draw and

		modify commands. Draw Modify toolbars are displayed when you start AutoCAD. These toolbars are docked on the left side of the window. You can easily move toolbars and turn them on and off. **Drawing area** : Displays drawings. The drawing area size varies depending on the size of the AutoCAD window and on the number of other elements (such as toolbars and dialog boxes) that are displayed. **Crosshairs** : Identifies pick and drawing points within the drawing area. Use the crosshairs, which are controlled by your pointing device, to locate points and select and draw objects. **User Co-ordinate system icon** : Shows the orientation of the drawing. AutoCAD drawings are superimposed on an invisible grid, or coordinate system. Coordinate systems are based on X,Y, and (for 3D) Z coordinates. AutoCAD has a fixed world coordinate system (WCS) and a moveable user coordinate system (UCS). To help you visualize the UCS location and orientation. A UCS icon is displayed in the lower left corner of the drawing area. **Model tab/ Layout tab** : Switch your drawing between model (drawing) space and paper (layout) space. You generally create your designs in model space, and then create layouts to plot and print your drawing in paper space. **Command window** : Displays prompts and messages. Also to invoke a command by entering the command on the command line. **Status bar** : Displays the cursor coordinates in the lower left corner. The status bar also contains buttons that you can use to turn on common drawing aids. These include Snap (Snap mode), Grid (drawing grid), Ortho (Ortho mode), Polar (polar tracking), Osnap (object snaps), Otrack (object snap tracking), Lwt (lineweight display) and Model (model and paper space toggle).
3.	Saving drawing files SAVE, SAVE AS and QSAVE	All these three commands allow you to save your drawing by writing it to a permanent storage device such as hard drive. SAVE command saves the current named drawing. The SAVE AS command sets the name of the current drawing to the new file name you specify. QSAVE command saves the current named drawing without asking you to enter a filename, thus allowing you to do a quick save. To invoke save command : Toolbar : Standard, Save Pulldown : File, Save or Save As Command : Save, Save As , Qsave

4.	Invoking commands in AutoCAD	You can use any of the following methods to start commands. Pull down menus Screen menus Toolbars Command line To repeat the last command : press ENTER or SPACEBAR or right click in the drawing area and choose repeat To repeat one of the last six commands : right click in the command window or text window and from the shortcut menu choose recent commands . To repeat the same command multiple times : at the command prompt enter multiple and at the next prompt enter the command you want to repeat . To cancel a command : press ESC.
5.	Coordinate system	**Absolute coordinate system** : In absolute coordinate system the points are located in respect to the origin (0,0). The absolute coordinates are specified by entering X and Y coordinates. For example point 2,3 represents point with X=2 and Y=3 from origin. **Relative Cartesian coordinates** : Relative coordinates specify the X and Y distance from a previous point. They are called relative coordinates because they only have meaning relative to a point previously specified. 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 polar coordinates, which can be either absolute or relative. Most commonly, you use relative polar coordinates. Polar coordinates take the form distance‹angle. (To type the angle symbol, use the less than symbol on your keyboard.) Relative polar coordinates must have the @ sign before the coordinate. **Direct distance entry** : One shortcut for entering coordinates is direct distance entry. After you specify the start point of a line, at the Specify next point or [Undo]: prompt, simply move the mouse cursor in the direction you want the line to go and type in the line's length. It works best in orthogonal mode or with polar tracking, discussed next, which makes it easy to specify exact angles. You can use direct distance entry for any command that requires you to specify a distance and a direction, including both drawing and editing commands.

Creating Elementary Objects

6.	LINE	Creates straight line segments Specify first point : press ENTER to continue from the last drawn line or arc or specify a point.
7.	XLINE	Creates an infinite line Specify a point or [Hor/Ver/Ang/Bisect/Offset]: Specify a point or enter an option.
8.	MLINE	Creates multiple parallel lines. Specify start point or [Justification/Scale/Style] : Specify a point or enter an option.
9.	PLINE	Create two-dimensional polylines. Specify start point :Specify point (1) Current line-width is <current> Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: Specify a point (2) or enter an option.
10.	POLYGON	Creates an equilateral closed polyline. Enter number of sides <current> : Enter a value between 3 and 1024 or press ENTER. Specify center of polygon or[Edge]: Specify point of center From the Draw menu, choose Polygon. Enter c (Circumscribed about Circle). Specify the radius length.
11.	RECTANG	Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: Enter an option or specify a point (1). First corner Specify other corner point : Specify a point (2) The different options of RECTANG command are: Chamfer　　: Creates a rectangle with chamfer by specifying chamfer distances. Fillet　　　: Creates a filleted rectangle by specifying fillet radius. Width　　　: Controls the line width of rectangle Thickness : Controls thickness of rectangle Elevation　: To draw a rectangle at a specified height from the XY plane.
12.	ARC	Creates an arc. In AutoCAD an arc can be drawn in 11 distinct ways. The default method of drawing an arc is 3point method. Other options are Start, Center, End　　: Specify start point, center point and end

		point of arc
		Start, Center, Angle : Specify start point, center point and included angle of arc
		Start, Center, Length : Specify start point, center point and cord length
		Start, End, Angle : Specify start point, end point and included angle
		Start, End, Direction : Specify start point, end point and starting direction of arc
		Start, End, Angle : Specify start point, end point and radius of arc
		Center ,Start, End : Specify center point, start point and end point of arc
		Center ,Start, Angle : Specify center point, start point and included angle of arc
		Center ,Start, Length : Specify center point, start point and cord length
		Continue : Continue drawing an arc from a previously drawn arc
13.	CIRCLE	Creates a circle Specify center point for circle or [3P/2P/Ttr(tan tan radius)] : Specify a point or enter an option . Center, Radius : By defining the center point and radius of the circle Two Point : By specifying the endpoints of circle diameter Three Point : By specifying 3 points on the circle Tangent, Tangent Radius : Draw a circle tangent to two object (line, circle or arc) and with a specified radius.
14.	DONUT	Draws filled circles and rings Specify inside diameter of donut <current>: Specify a distance or press ENTER Specify outside diameter of donut <current>: Specify a distance or press ENTER Specify center diameter of donut <exit>: Specify a point or press ENTER to end the command.
15.	SPLINE	Creates a quadratic of cubic spline (NURBS) curve Specify first point or [Object]: Specify a point or enter 'O'
16.	ELLIPSE	Creates an ellipse or an elliptical arc Specify axis endpoint of ellipse [Acr/Center/Isocircle]: specify a center or enter an option. The Isocircle option is available only when you set the Style option of SNAP to Isocircle.

17.	BLOCK	Creates a block definition from objects you select. AutoCAD displays the Block Definition dialog box. If you enter-block at the command box, BLOCK displays prompts on the command line.
18.	POINT	Creates a point object (Try PDMODE & PDSIZE)
19.	HATCH	Fills a specified boundary with a pattern Enter a pattern name or [?/Solid/User Defined] <current>: Enter a predefined or custom pattern name, enter an option or press ENTER
20.	BHATCH	To create associative hatches. This command invokes a boundary hatch dialog box and allows you to hatch a closed area by selecting a point inside the boundary or by selecting the boundary objects. The boundary hatch option has several options Type Pattern Swatch
21.	REGION	Creates a region of objects from a selection set of existing objects Select objects : Use an object selection method and press ENTER when you finish.

Modifying Commands

S.N	Command	Description
1.	ERASE	• Removes objects from a drawing • Select Objects : Use an object selection method and press ENTER
2.	COPY	• Duplicates objects. • Select objects : Select objects and press ENTER • Specify base point or displacement, or [Multiple] : Specify a point for a single copy or enter m for multiple copies.
3.	MIRROR	• Creates a mirror image copy of objects • Select objects: Use an object selection method and press ENTER to finish • Specify first point of mirror line: Specify a point (1) • Specify second point of mirror line : Specify a point (2)
4.	OFFSET	• Specify offset distance of [Through] ‹current› Specify a distance, enter t, or press ENTER • Offset Distance : Creates an object at a specified distance from an existing object • Through : Creates an object passing through a specified point
5.	ARRAY	• Creates multiple copies of objects in a pattern • Select objects : • Enter the type of array [Rectangular/Polar]‹current›
6.	MOVE	• Displaces objects at a specified distance in a specified direction • Select objects : Use an object selection method and press ENTER when you finish • Specify base point or displacement. Specify a base point(1) • Specify second point of displacement or ‹use first point as displacement› Specify a point (2) or press ENTER
7.	ROTATE	• Moves objects about a base point • Specify rotation angle or [Reference]: Specify an angle, specify a point, or enter r

		• Rotation Angle : Determines how far an object rotates around the base point.
		• Reference: Specifies the absolute current rotation angle and desired new rotation angle. You can use the Reference option to straighten an object or align it with other features in a drawing.
8.	SCALE	• Enlarges or reduces selected objects equally in the X, Y and Z directions.
		• Select objects: Use an object selection method and press ENTER when you finish
		• Specify base point: Specify a point (1)
		• Specify scale factor or [Reference]: Specify a scale or enter
		• Scale Factor: Multiples the dimensions of the selected objects by the specified scale. A scale factor greater than 1 enlarges the objects. A scale factor between 0 and 1 shrinks the objects.
9.	TRIM	• Trims objects at a cutting edge defined by other objects.
10.	EXTEND	• Extend an object to meet another object
		• Select boundary edges.
		• Select objects : Use an object selection method and press ENTER when you finish
		• Select object to extend or {Project/Edge/Undo]: Select an object or enter an option
11.	CHAMFER	• Command line: chamfer
		• (TRIM mode) current chamfer Dist1 = current, Dist2 = current
		• Select first line or [Polyline/Distance/Angle/Trim/Method]
12.	FILLET	• Rounds and fillets the edges of objects.
		• Current Setting: Mode = current, Radius = current
		• Select first object or [Polytechnic/Radius/Trim]: use an object selection method or enter an option
13.	BREAK	• Erases parts of objects or splits an object in two
		• Select object: Use an object selection method, or specify the first break point (1) on an object
		• Specify second break point or [First point]: Specify the second break point (2) or enter f
		• Second Break Point: Specifies the second point for

		AutoCAD to use the object.
		• First Point: Overrides the original first point with the new point that you specify.
		• Specify first break point.
		• Specify second break point.
14.	EXPLODE	• Breaks a compound object into its component objects
		• Select objects : Use an object selection method and press ENTER when you finish selecting objects.

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Drawing Aids

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15.	Setting units & drawing limits	• Pull down menu : Format, units
		• Command line : Units
16.	Drawing Limits	• You can set the limits of the drawing area in Metric or English before you start a drawing. The grid provides a visual representation of the limits. Setting the limits serves as a reference tool that marks the area in your drawing in which you are currently working
		• A drawing tool that optionally prevents drawing outside the grid limits
		• A plot option that defines an area to be printed
17.	Layer, Color, Lineweight & linetype	• In AutoCAD two tools are available to edit object properties.
		• Object properties tool bar
		• Properties
18.	Snap & Grid	• Snap and Grid settings help you create and align objects. The grid is a visual guide displaying points at user specified intervals, like in a grid paper. Snap spacing restricts cursor movements to specified intervals.
		• Pull down menu : Tools Drafting setting, Snap & Grid
		• Command Line : DSETTINGS
		• Isometric snap/grid mode helps you to create 2D drawings that represents 3D objects.
19.	Object snaps	• AutoCAD provides tracking and object snap tools, to make drawings quickly and accurately.
		• Pull down menu : Tools, Drafting, setting, Object snap
		• Command line : DSETTINGS, OSNAP

20.	Function and Control keys	- To change the status of coordinate display, ortho etc. you can use Function and Control keys. The following is the list of function and control keys. - F1 Help - F2 Graphics Screen/Text screen - F3 Running osnap ON/OFF (CTRL+F) - F4 Table mode ON/OFF (CTRL+T) - F5 Iso top/Iso right/Iso left (CTRL+E) - F6 Coordinate display On/Off (CTRL+D) - F7 Grid ON/OFF (CTRL+G) - F8 Ortho ON/OFF (CTRL+L) - F9 Snap ON/OFF (CTRL+B) - F10 Status Bar ON/OFF (CTRL+F)
	Controlling Drawing Display	
21.	Redraw	- This command redraws the screen, thereby removing the small cross marks that appear when a point is specified on the screen. - Standard Tool bar : Redraw - Pull down menu : View, redraw - Command : REDRAW
22.	Regen	- The REGEN command makes AutoCAD regenerate the entire drawing to update it. The regen occurs when you change certain aspects of the drawing. One of the advantages of this command is that the drawing is refined by smoothing out the circles and arcs. - Pull down menu : View, Regen - Command : REGEN
23.	Zoom	- Zoom command is useful to work on minute details of drawing. - Tool bar : Zoom - Pull down menu: View, Zoom - Command : Zoom - This command has different options.
24.	Pan	- Using PAN command, you can view or draw on a particular area outside the current viewport - Tool bar : Pan - Pull down menu : View, pan - Command : PAN
25.	Aerial view	Aerial view is a navigation tool that displays a view of the drawing in a separate window so that you can

		quickly move to that area. • Pull down menu : View, Aerial View • Command line : DSVIEWER

<h1 align="center">Inquiry Commands</h1>

26.	Area	• Calculates the area and perimeter of objects or of defined areas. • Inquiry toolbar : Area • Pull down menu : Inquiry, Area • Command line : Area
27.	Dist	• Measures the distance and angle between two points • Specify first point : Specify a point (1) • Specify second point : Specify a point (2)
28.	List	• Displays database information for selected objects • Select objects : Use an object selection method • AutoCAD lists the object type, object layer, and X,Y,Z position relative to the current user coordinate system (UCS) • ID Displays the coordinates values of a location • Inquiry Toolbar : Locate point • Pull down menu : Inquiry, ID Point • Command line : ID

<h1 align="center">Text & Dimensions</h1>

29.	Text	• Text conveys important information in drawings. • Pull down menu : Draw, Single line text • Command line : TEXT • To modify line text object properties. • Pull down menu : Modify properties • Command line : Properties • To add special characters while using single line text • %%P Plus/minus symbol • %%C Diameter symbol • %%D Degree symbol • %% U Underline
30.	Multiline text	• Multiline text consists of any number of text lines or paragraphs that fit within a width you specify • Tool bar : Multiline text • Pull down menu : Draw Multiline text