

Page No. |

Date

Exp. No.

**INTRODUCTION :**

Computer Aided Drawing is the Application of conventional computer technologies, written the aid of data processing systems.

Autocad provides a set of entities for use in constructing the drawing. an entity is a drawing element such as line, circles, etc....

Autocad (a product and registered trademark of auto desk inc. USA) is a lowcost yet a very effective computer aided design and drawing software. Autocad comes with a large number of users - selectable options to support a great variety of commercially available display devices, digitalizers, mouses, printers and plotters. Auto cad supports 2d drafting and 3d modeling.

The basic drawing entities are lines, polylines, circles, arcs etc.... there are many ways of defining a drawing entities. each drawing entities has an associated line type, color, layers and thickness. The thickness is a property associated with 3d entities.

In autocad the drawings are always prepared at full scale, and the drawing size can be changed at any instant of time by using limits command.

CO-ORDINATE SYSTEM IN AUTOCAD:

1. Absolute co-ordinate system

Format : (x-co-ordinate, y-co-ordinate)

2. Relative rectangular co-ordinate system

Format : (@ x displacement, y displacement)

3. Relative polar co-ordinate system

Format : (@ distance angle with x axis)

LIMITS :

Sets and controls the limits of display in the current drawing window (after setting the limits we have to do the following) at command prompt enter.

Limits ↵

Specify lower limit: 0,0 ↵

Specify upper limit: 500,500 ↵

Z ↵

A ↵

FUNCTION KEYS :

F1	Help	Explanations of commands.
F2	Flip screen	Toggles from Text screen to Graphics screen.
F3	Osnap	Toggles Osnap On and Off.
F4	Tablet	Toggles the Tablet On and Off.
F5	Isoplane	Changes the Isoplane from Top to Right to Left
F6	Coordinate display	changes the display from ON/off/



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Kacharam, Shamshabad - 501 218, Hyderabad.

Page No. 3		Date
		Exp. No.
F7	Grid	Toggles the Grid On or Off
F8	Ortho	Toggles Ortho On or Off
F9	Snap	Toggles Increment Snap On or Off
F10	Polar	Toggles Polar Tracking On or Off
F11	Track	Toggles Object Snap Tracking On and Off.

## GRID:

The grid is Visual reference Only. The grid is a rectangular pattern of dots. To draw objects along specific alignment is or angles, you can rotate the snap angles, which rotates the crosshairs and the grid. If Ortho mode is on autocad constraints cursor movement to the new snap angle and its perpendicular angle.

## LINE RESETING:

Shortcut menu right click snap on the status bar and choose setting. The origin of the rotated snap angle is the base point. You can offset the base point by changing the X or Y base (set to 0.0000 by default) offsetting the X or Y base point is useful when we want to align the grid with complex patterns.

## LINE SETTING:

Shortcuts means right click snap on grid on the status bar and choose settings. System Variables snapping and snapplane set the snap angle and snap base.

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(AUTONOMOUS)

Kacharam, Shamshabad - 501 218, Hyderabad.

Page No. 4

Date

Exp. No.

## DRAW TOOL BAR :

SERIAL NO.	NAME OF THE TOOL	SHORTCUT	SYMBOL
1.	Line	L	
2.	Construction line		
3.	Polyline	Pl	
4.	Polygon	Pol	
5.	Rectangle	Rec	
6.	Arc	A	
7.	Circle	C	
8.	Revcloud		
9.	Spline		
10.	Ellipse	El	
11.	Ellipse arc		
12.	Insert block		
13.	Make block	B	
14.	Point	PO	
15.	Hatch	H	
16.	Region		
17.	Multiline-text	T	A

Page No. 5

Date

Exp. No.

## BASIC DRAW COMMANDS :

### LINE (L) :

The line command draws a line between two points that you can choose anywhere in the drawing area. You can designate these points with absolute co-ordinate system and relative polar co-ordinate system.

After drawing the first segment of a line we can end the command, or draw another line segment from the end of the first line. We can continue to draw as long as we like.

### STRAIGHT LINE SETTING :

Specify the endpoints of lines using two dimensional (2d) or three dimensional (3d) co-ordinates.

### DRAW TOOLBAR :

Draw menu : line

Command : line

Specify first point : Press enter to continue from the last draw line or use to specify a point.

### CIRCLE (C) :

To create a circle it offers several methods for drawing circle. By choosing centre point and radius or diameter, 3P to 1 options.



Page No. 6

Date

Exp. No.

OPTIONS :CENTRE POINT :

Draw a circle based on a center point and a diameter or radius. Specify radius of circle or (diameter); specify a point (2) enter the radius value.

3Point (3p) :

To create a circle based on specifying 3 points. The circle will be drawn to pass through these points.

2Point (2p) :

To create a circle based on specifying 2 end points of circle diameter.

Tangent to Tangent Radius (TTR) :

To create a circle based on specifying 2 tangent points and a radius.

ARC (A) :

Arc's can be created by using varieties of methods. The default method is to specify three points. A start point, a second point on a arc, and an end point. we can also specify the included angle, radius, direction, and chord length of arcs. The chord of an arc is a straight line between the end points.

To draw an arc by specifying 3 points.

1. From the draw menu, choose arc about, center, end.
2. Specify the start point (1) by entering end point and selecting the line.

Page No. 7.

Date

Exp. No.

3. specify the second point (2) by entering centrepoint and selecting the existing arc to define the centre of the arc.

4. specify the end point of the arc (3).

### RECTANGLE (REC):

It draws rectangular polyline with chamfered filleted corner or width rectangle.

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 two specified points determine the diagonal corners of a rectangle with sides parallel to the X and Y axis of the current UCS.

### CHAMFER:

Specify the chamfer distances for the rectangular. specify the first chamfer distance for rectangles.

< current >: specify a distance or press "enter" specify second chamfer distance for rectangles.

< current >: specify a distance or press "enter" the value becomes the current chamfer distance for subsequent rectangle commands.

### Elevation:

Specify the elevation of rectangle

Specify the elevation for rectangles < current >:

specify the distance or press enter

The value becomes the current elevation for subsequent rectangle commands.



Page No. 8

Date

Exp. No.

Fillet:

Specify the fillet radius of rectangle

Specify fillet radius for rectangle < current >

Specify a distance or press enter.

The value becomes the current radius for subsequent rectangle commands.

Width:

Specify the polyline width of rectangle to the drawn.

Specify the width for rectangles < current >; specify a distance or press enter

The value becomes the current polyline width for subsequent rectangle commands.

POLYGON (POL):

To draw a regular polygon of required number of sides (3 to 1024). The polygon can either be inscribed in a circle or circumscribed about the circle. The polygon may also be constructed by specifying the length of one side called edge option.

Enter no. of sides < current >; enter a value b/w 3 and 1024 or press enter specify centre of polygon or (edge); specify a point (1) enter e.

ELLIPSE (EL):

It draws an ellipse by specifying two end points of an axis and distance to other axis end point, centre point and endpoint of an axis and distance to other axis. Also we can create and



Page No. 9

Date

Exp. No.

elliptical arc by using arc option specify axis end point of ellipse or (arc / centre / iso circle); specify a point (1) or enter an option. The iso circle option is available only when you set the style option of snap to isometric.

**X-LINE (XL):**

Specify a line through (horizontal / vertical / biset / offset angle)

**M-LINE (ML):**

Draws a straight parallel lines.

**SPLINE (SPL):**

Draws a diagonal or cubic spline.

**P-LINE (PL):**

Draws a diagonal polylines it shows single entity.

**POINT (POINT):**

Draws a Point,

**BHATCH (BHATCH):**

Draws an hatching lines in the selected enclosed area.

*[Signature]*  
22/4/14

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











Kacharam, Shamshabad - 501 218, Hyderabad.

Page No. 10

Date

Exp. No.

## MODIFY TOOL BAR

S.NO	NAME OF THE TOOL	SHORTCUT	SYMBOL
1.	Erase	e	
2.	Copy object	Co	
3.	Mirror	mi	
4.	Offset	O	
5.	Array	Ar	
6.	Move	m	
7.	Rotate	ro	
8.	Scale	sc	
9.	Stretch	s	
10.	Trim	tr	
11.	Extend	ex	
12.	Break at Point		
13.	Break		
14.	Chamfer	cha	
15.	Fillet	f	
16.	Explode		



Page No. ||

Date

Exp. No.

MODIFYING COMMANDS

MOVE (M): To move required object from one point to another Point

Select the objects use an object selection method and press enter when you finish

Specify base point or displacement specify a basepoint (1), Specify second point of displacement or < use first point as displacement >; Specify a point (2) or press enter.

COPY (CO): It copies required objects from one point to another point.

The selected object can be copied 'P' objects times by using multiple option.

Selected objects select object and press enter

Specify bases points or displacement, or (multiple): specify a Point (1) for a single copy or in for multiple choices, copies.

ARRAY (AR): It creates an array of single object into same object either in a polar form or a rectangular form or a rectangular form.

Rectangular array: Creates an array defined by a no. of rows and columns of copies of the selected objects.

Enter the no. of columns (11) <1> enter a non zero integer or press enter.

Page No. 12

Date

Exp. No.

Polar array: Creates an array defined by specifying a centre point or base point about which auto-cad replicates the selected objects. Specify centre point of array: Specify a point (1)

Creates an array defined by a centre point.

Enter the no of items in the array: Enter a positive integer or Press ENTER

OFFSET (O):

To create parallel objects with required distance or through a point, it will affect only a single object.

Specify object offset or [THROUGH] <current>

Specify a distance, enter or Press ENTER

MIRROR (MI):

To create mirror image of required object by specifying two points of mirror line

Selected Objects: Use an object selection method and press ENTER to finish specifying first point of mirror line: Specify point (1).

EXTEND (EX):

It extends age of an object to meet another object. Objects that can be extended are axes, elliptical arcs, and open 2D polyline objects. Select objects: use an object selection method and press ENTER.

TRIM (TR):

It removes edges of an object beyond another cutting edge. Select objects: select an object and press ENTER.



Page No. 13

Date

Exp. No.

select objects that defines the cutting edge to trim the object  
or press ENTER.

### CHAMFER (CHA):

It levels the corners of two line segment or polyline object with required distance.

Current chamfer Distance 1 = Current, Distance 2 = Current,  
Select first line or [Polyline / Distance / Angle / Trim / Method]

### FILLET (F):

It creates a filleted curve between two objects select the first of two objects required to define a 2D fillet or the edge of a 3D solid to fillet.

select the second object and press ENTER.

Specify the radius and press ENTER. And arc with specified radius will be created.

### ZOOM (Z):

It increases or decreases an apparent size of drawing displayed in current drawing window.

Specify corner of windows, enter a scale factor (n x or n x P) or (all / centre / dynamic / extents / scale / windows) <real time>

ERASE (E): remove objects from drawing

BREAK (BR): splits an objects into two

STRETCH (S): moves or stretches the objects.

LENGTHEN (LEN): length the objects.

# VARDHAMAN COLLEGE OF ENGINEERING

(AUTONOMOUS)

Kacharam, Shamshabad - 501 218, Hyderabad.

Page No. 14

Date

Exp. No.

ROTATE (RO): rotates the objects about the base point

SCALE (SC): enlarge and reduce objects in X,Y directions.

*22/4/19*





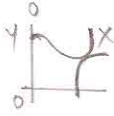





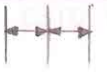









Page No. 15

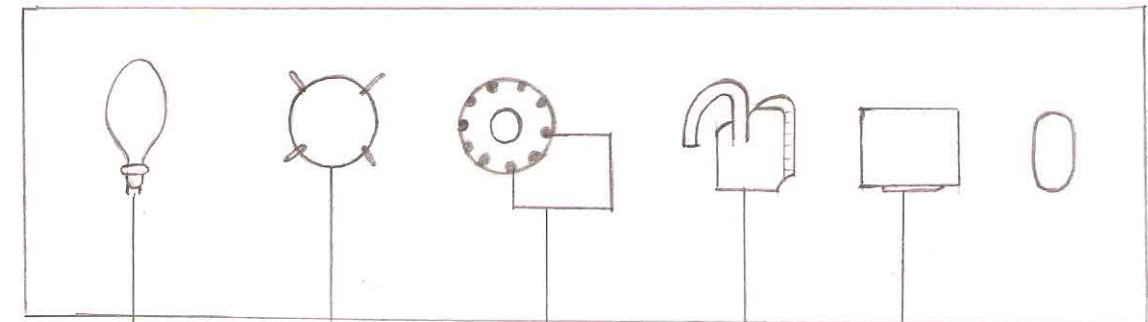
Date

Exp. No.

## DIMENSION TOOL BAR :

S.NO	NAME OF THE TOOL	SYMBOL
1.	Linear Dimension	
2.	Aligned Dimension	
3.	Ordinate Dimension	
4.	Radius Dimension	
5.	Diameter Dimension	
6.	Angular Dimension	
7.	Quick Dimension	
8.	Baseline Dimension	
9.	Continue Dimension	
10.	Quick leader	
11.	Tolerance	
12.	Center Mark	
13.	Dimension Edit	
14.	Dimension Text Edit	
15.	Dimension Update	
16.	Dimension styles	

## LAYERS TOOL BAR:



Turn a layer  
on or off

Freeze or thaw in all view point

Freeze or thaw in current view point

Lock or Unlock a layer

Color of layer.

## PROPERTIES TOOLBAR

1. ☐ BY LAYER ▼ → Color Control
2. — BY LAYER ▼ → Line type Control
3. — BY LAYER ▼ → Line weight Control.













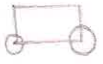



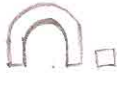


Page No. 17




















Date

Exp. No.

## OBJECT SNAP TOOL BAR

S.NO.	NAME OF THE TOOL	
1.	Temporary Tracking point	
2.	Snap from.	
3.	Snap to Endpoint	
4.	Snap to Midpoint	
5.	Snap to Intersection.	
6.	Snap to Apparent Intersect.	
7.	Snap to Extension.	
8.	Snap to Centre	
9.	Snap to Quadrant	
10.	Snap to Tangent	
11.	Snap to perpendicular	
12.	Snap to parallel	
13.	Snap to Insert	
14.	Snap to Node	
15.	Snap to Nearest	
16.	Snap to None	
17.	Object snap Settings	

## STANDARD TOOLBAR :





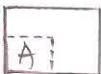
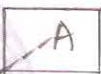
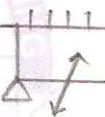
S.NO.	NAME OF THE TOOL	SYMBOL	SHORCUT
1.	New		
2.	Open		(ctrl+O)
3.	Save		(ctrl+S)
4.	Plot		(ctrl+P)
5.	Plot preview		
6.	Publish		
7.	Cut to clipboard		(ctrl+X)
8.	Copy to clipboard		(ctrl+C)
9.	Paste from Clipboard		(ctrl+V)
10.	Match properties.		
11.	Undo		
12.	Pan real time		
13.	Zoom real time		
14.	Zoom window		
15.	Zoom Previous		
16.	Properties .		(ctrl+1)
17.	Design Centre .		(ctrl+2)
18.	Tool palettes		(ctrl+3)
19.	Help		

Page No. 19.

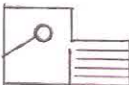
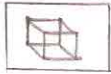

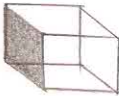

Date

Exp. No.

## TEXT TOOL BAR :

S.NO	NAME OF THE TOOL	SYMBOL
1.	Multiline Text	
2.	Single line text	AI
3.	Edit text	A 
4.	Find and Replace	
5.	Text Style	A 
6.	Scale Text	
7.	Justify style Text	
8.	Convert Distance between spaces.	

## VIEW TOOL BAR :

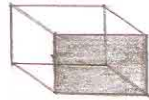













S.NO	NAME OF THE TOOL	SYMBOL
1.	Named View	
2.	Top View	
3.	Bottom View	
4.	Left View	
5.	Right View	



# VARDHAMAN COLLEGE OF ENGINEERING

(AUTONOMOUS)

Kacharam, Shamshabad - 501 218, Hyderabad.

Page No. 20.		Date
		Exp. No.
6.	Front View	
7.	Back View	
8.	SW Isometric View	
9.	SE Isometric View	
10.	NE Isometric View	
11.	NW Isometric View	
12.	Camera.	
ZOOM TOOL BAR :		
S.NO.	NAME OF THE TOOL.	SYMBOL
1.	Zoom Window	
2.	Zoom Dynamic	
3.	Zoom Scale	
4.	Zoom Centre.	
5.	Zoom In	
6.	Zoom Out	
7.	Zoom all	
8.	Zoom extents	

*[Signature]*  
22/4/24