

EXPERIMENT NO: Module:1

TITLE: DRAWING BASICS

LEARNING OBJECTIVES:

- To concentrate on basics of AutoCAD, which is the base for drafting.
- To use the functions and commands of AutoCAD software to create, save, and print drawings that make use of multiple lines, geometric shapes, and curves.
- To locate and apply the many features of AutoCAD that automates the drafting process and facilitate the creation of more accurate drawings in less time than traditional drafting methods.
- To locate and apply the features of AutoCAD that provide for the accurate addition of dimensions, tolerances, and drawing notes and labels using symbols and placements recognized by multiple standards organizations.
- To use the functions and commands of AutoCAD software to create isometric and three dimensional drawings and models.

AIM:

- To study the basics of AutoCAD.

MATERIAL / EQUIPMENT REQUIRED:

- Computer with AutoCAD

THEORY / HYPOTHESIS:

- **AutoCAD (Automated Computer Aided Drafting)** is the most widely used computer aided drawing / drafting package. AutoCAD is a 2 D and 3 D computer aided drafting software application used in architecture, construction and manufacturing to assist in the preparation of blueprints and other engineering plans. AutoCAD provides a common look and feel GUI facilities to the learner to make great use of windows environment. AutoCAD is simple to learn, use and draw which provides multiple document opening facility so the user can exchange data between different files and to compare files more easily.
- Start an AutoCAD session by double clicking on an AutoCAD icon.
- AutoCAD opens, loads the menu and AutoCAD editor screen which contains the drawing area, command line, standard tool bar, status bar, etc.
- Some of the basic commands used in AutoCAD are as follows.

DRAWING TOOLS

LINE

- Line command is used to draw straight line segment.
- Command: LINE or L
- Tool Bar: Draw line

- **CO ORDINATE SYSTEMS**

- **Absolute co ordinate system**

- In this system, location of a point is described with reference of a previous point and hence is called relative co ordinate system.

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Polar co ordinate system

- In this system, the location of a point is described in terms of the length of a line and direction with reference to a previous point.

POLY LINE

- Command: PO
- Tool bar: Draw Poly line

CIRCLE

- Circle command is used to draw a circle.
- Command: Circle
- Tool bar: Draw circle

Types can be drawn by using following methods.

- Center point:
 - Specify the radius of circle or diameter, specify or point (P), enter a radius or enter a diameter by d
 - Radius:
 - Specify the centre of the circle and give the value of radius.
 - Specify the value of diameter after d
 - Diameter:
 - Specify the center of circle and to give diameter press d . Specify the value of diameter of given circle.
- 3P (3 points):
 - This method allows user to pick any three points randomly, then AutoCAD draws a circle touching all the three points.
- Specify the first point of the circle P1
- Specify the second point of the circle P2
- Specify the third point of the circle P3
- TTR (Tangent Tangent_Radius) allows the user to pick two tangent points and a radius.

- Specify the first tangent of given point(T1)
- Specify the second tangent of given object(T2)
- Specify the radius of circle
- TTT (tangent tangent tangent)
- Specify the first tangent (T1) second tangent(T2) and third tangent (T3) to draw the circle.

ARC

- Arc command is used to draw an arc. An arc is a portion of a circle. Therefore it is necessary to specify the centre, radius, start and end point of the arc.
- Command: NARC or a
- Tool bar: draw an arc.

Types of arc

- Three point:
 - Arc passes through given three points. Pick start, second and end point.
- Start and direction:
 - Pick start point, pick end point and pick direction through straight point.
- Start center end:
 - Start end radius:
 - Pick start point; pick end and type radius of arc or show radius on arc.
 - Start center angle:
 - Pick start point, pick center and end point.
- Center start end:
 - Pick center, pick start point and end point.
- Start center length of chord
 - Pick start, pick center and enter length of chord
- Center start angle
 - Pick center, start point, center angle (counter clockwise from start point)
- Start end angle
 - Pick start point, pick end point, specify angle.

DONUT

- The DONUT command can be used to create a solid filled Circle as well as the standard

object that the DONUT command is normally used for. The DONUT command normally is used to draw a circle with a thick outer edge and open in the middle, hence the command name DONUT. By changing the inside diameter to 0 (zero) you can make it create a solid filled circle.

- Command: DONUT or do
- Tool bar: Draw DONUT.
- Enter inside diameter of DONUT.
- Enter outside diameter of DONUT.
- Specify the centre of the DONUT.

ELLIPSE

- Command: ELLIPSE or El
- Tool bar: Draw ellipse
- Pick the centre point option of ellipse.
- Pick axis and point.
- Specify the distance to the other axis or rotation, pick a point P or specify a distance using key board and press .

POLYGON

- Polygons are multi sided closed figure with equal side lengths.
- Command: POLYGON or POLY
- Tool bar: Draw polygon.
- Inscribed or circumscribed polygons are available in AutoCAD.
- Specify the number of sides.
- Specify the centre of polygon or edges.
- Enter the option pick I or C from command bar.
- Pick the value of the radius and specify the radius.

MODIFYING TOOLS

ERASE

- Removes objects from a drawing.
- Command: E or Erase
- Tool bar: Erase and select the object to be erased.

MOVE

- Move helps to displace the objects to the specified distance in specified direction.

- Command: Move or M
- Tool bar: Move
- Select the required object and then and by selecting the base point, displace the selected object by scrolling the mouse and place it.

COPY

- Copy helps to duplicate or create a copy of a particular object.
- Command: CO or copy
- Tool bar: Copy
- Specified object is selected and then press and select the base point and press . If multiple copies are required and copied objects are placed in the destiny.

MIRROR

- Many drawings have identical elements, say for an example, twin house drawings or any mechanical drawings; you create one have or one quarter of a drawing and complete it simply mirroring what you have drawn already.
- Command: MI or mirror
- Tool bar: Mirror
- Select the object and press , then specify the first point of the mirror line and the second point and press Y if the copy is to be displayed and N for mirrored image.

OFFSET

- The offset command is used to create any entity parallel to one existing object.
- Command: O or offset
- Tool bar: Offset
- Select the object to offset, specify the distance or T to press the distance required for the user to place the object.

ARRAY

- It creates multiple copies of an object in a pattern.
- Command: Array
- Tool bar: Array
- Array menu bar will appear and select the type of array required.

POLAR ARRAY

- It creates an array defined by specifying a centre point or base point of the object
-

which it replicates the selected object.

RECTANGULAR ARRAY

- It creates an array defined by a number of rows or columns of copies of the selected object.

ROTATE

- It is used to rotate the selected group of objects about specified base point through a given angle.
- Command: Rotate or RO
- Tool bar: Rotate
- Select object to rotate and press . Specify the base point, pick a base point (point about which object rotates). Specify rotation angle (reference) or simply specify an angle.

REFERENCE

- The option is used to align object to another by specifying angle.

SCALE

- Enlarges or reduces the selected objects equally in x, y and z directions.
- Command: SC or scale
- Tool bar: Scale
- Select the object which are to be scaled. Specify the base point on object, specify the scale factor and then enter the value for scaling.

SCALE FACTOR

- Multiplies the dimensions of the selected objects by the specified scale. A scale factor is greater than 1, enlarges the object and scale between 0 and 1, and shrinks the object.

STRETCH

- Moves or stretches the object.
- Command: Stretch or S
- Tool bar: Stretch
- Selecting the object by making crossing window and press specify the base point or pick a point to enlarge the shape and line of the object by pulling or pushing from

one side and also to move object from one place to another place.

TRIM

- Trim command is used to trim objects. To trim an object, you must specify the cutting edge, which defines the point at which the AutoCAD cuts the object you want to trim.
- Command: Trim or TR
- Tool bar: Trim
- By giving the trim command and press and select the object which should be discovered from the drawing.

EXTEND

- Extend is used for extension of a line or an arc to the destiny point.
- Command: Ex
- Tool bar: Extender

BREAKING OBJECTS

- The break command is used to break the object at the point or points you specify. To break line, choose break from modify tool bar and follow the prompts.
- Command: Break
- Select object: pick object to break.

CHAMFER

- The CHAMFER command creates corners from two non parallel lines. It allows the user to chamfer poly line, rays etc.
- Command: Chamfer

FILLET

The fillet command creates rounded corners replacing two lines with arc. It allows the user to chamfer poly lines, rays etc.

- Command: Fillet

DIVIDE

- This divides selected objects into specified number of equal distance by intersecting marker point on the object.
- Command: divide or div
- Explode

- Entities like rectangular

MEASURE

- Starts at one end of the object measure out segments of specified length between each point or block.
- Command: Measure
- Tool bar: Draw point measure
- Select object to measure: Pick an object
- Specify length of the segment: enter length of segment

EXPLODE

- Entities like rectangle, polygon and poly lines are considered as one entity even though they are made up of segments. To connect them into individual segment, explode command is used.
- Command: X
- Tool bar: Explode
- Select object to explode and press

CREATING TEXT OBJECT

- The text information is essential to understand any drawing. Text is used for title box, labeling of parts, to give specification or to make notations in the drawing. Broadly there are two types of text items used in AutoCAD.
- Single line text
- Double line text
- Command: Text
- Tool bar: Single line text
- A text style dialogue box will appear as follows.
- Select font name, font style and height of text.
- Also enter width factor and oblique angle. Click on apply and then on exit.

1. WRITING TEXT

- Command: Dtext or dt
- Tool bar: draw text single line text
- Dtext / justify / style <start point> pick a point rotation angle <0> Enter angle for

text line. Type the text required.

2. JUSTIFY

- Specify the start point of text or (justify/style) enter an option
- Align/fit/centre/middle/right/TL/TC/TR/ML/MC/MR/BL/BC/BR; Enter any one of option for text.

3. ALIGN

- Specify both text height and text orientation by designating the end point of base line.

4. FIT

- Specified text will fit in the specified area and at an orientation defined with two points and a height.
- Specify first end point of the base line: specify a point (1)
- Specify second end point of the base line: specify a point (2).

5. CENTRE

- Aligns the text from horizontal centre of base line, which you specify with points.
- Centre points: pick centre point for text line
- Rotation angle: specify the angle for text base line.
- Text: type the text and press

6. MIDDLE

- The selected midpoint is the exact midpoint of the text.
- Midpoint: pick a point.
- Rotation angle: specify an angle for line, enter text
- Right: The picked point will be extreme right end of text extend
- End point: Pick right end for text
- Rotation angle: specify an angle
- Text: type the text required

7. OTHER OPTIONS

- Other points are TL,TC,TR the texts are aligned with the top of text box and ML,MC,MR are the text which are aligned with a middle of text box and BL,BC,BR are the text which are aligned with bottom of text box.
- Multi line text:
- Command: mtext
- Tool bar: Multiline text
- It is a text in which the single style of text colour, its height can be adjusted according to the requirement. The arrangement will be displayed.
- Style: Select the style name.
- Font: Select the font style
- Height: set the height of letters
- We can even select the style of letter < bold/italic/underline>. We can select the

colour required also.

8. HATCHING:

- Hatches are shaded pattern which generally represents a cross section of component or object.
- Command: H
- Hatch tab: define the appearance of hatch pattern to be applied.
- Type: sets the pattern type. User defined patterns are based on current line type in user drawing.

9. PATTERN:

- List the available pre defined patterns. The fix most recently used pre defined patterns appears at the top of list.

10. SWATCH:

- Displays a previous selected pattern, you can click the swatch display the hatch pattern box.

11. ANGLE:

- Specify an angle for the hatch pattern, relative to the x axis of current use.

12. SCALE:

- You can add, delete and rename the layers change their properties and set layer property for layout view ports or add layers descriptions and apply changes.

13. SPACING:

- It specifies spacing of line in a used defined pattern.

14. LAYER:

- Layers are used to group information I a drawing, foundation and reinforcement line type, colour and other standards. Layers are equivalent to overlays used in paper based drafting. Layer is the primary organizational tool used in drawing. You can turn off the layers that you don't need and plot a drawing containing only the required information. You can add, delete and rename the layers change their properties and set layer property for layout view ports or add layers descriptions and apply changes.
- Layer filter controls the layers displayed and can also used to make changes to more than one layer at a time.

Function key Function defined in AutoCAD

F1 -Online help

F2 -Toggles between command window on and off

F3 -Toggles between OSNAP on and off

F4 -Toggles between Tablet on and off

F5 -Switches among Isoplanes Top, Right and Left

F6 -Toggles between co-ordinates on and off

F7 -Toggles between Grid on and off

F8 -Toggles between Ortho on and off

F9 -Toggles between Snap on and off

F10- Toggles between Polar Tracking on and off

F11 -Toggles between Object Snap Tracking on and off

F12 -Dynamic Input on and off

LEARNING OUTCOMES :

- Knowledge about basic commands and its usage of AutoCAD software

APPLICATION AREAS:

- Drafting
- Modelling