

EXPERIMENT NO:2

DRAWING TOOLS

A. LINE COMMAND: It creates one or series of straight line segments here each line segment is separate object.

COMMAND: LINE or L (PRESS ENTER)

From point: 10, 10

To point: (Move the cursor horizontally to right with ortho ON) type 20, press enter.

To point: (Move the cursor horizontal, vertically up) type 20, press enter.

To point: (Move the cursor horizontally to left) type 20, press enter.

To point: c.

B. CIRCLE COMMAND: It creates a circle.

CENTER AND RADIUS: Draw a circle based on a center point and radius command circle.

CENTER AND DIAMETER: Draw a circle based on a center point and diameter. **THREE POINT (3P) CIRCLE:** Draw a circle based on 3point on circumference. **TWO POINT (2P) CIRCLE:** Draw a circle based on two points of a diameters.

TANGENT TANGENT RADIUS: Draw a circle tangent to existing object with specified radius.

COMMAND: CIRCLE or C.

3p/2p/TTR/ <enter point>; pick center or circle enter co-ordinates from center.

Diameter/<radius>; D

Diameter; specify diameter.

COMMAND: CIRCLE or C.

3p/2p/TTR/ <enter point>; 3p

First point; pick first

point. Second point; pick second

point. Third point; pick third

point.

COMMAND: CIRCLE or C.

3p/2p/TTR/<enter point>; 2p

First point of diameter; pick first point.

Second point of diameter; pick second

point.

COMMAND: CIRCLE or C.

3p/ 2p/ TTR/ <enter point>; TTR

Enter tangent specification; pick first tangential object.

Enter second tangent specification; pick second tangential object.

Radius; enter radius of circle.

C.ARC COMMAND: It creates an arc and is used to add curved segments to the drawing.

THREE POINT ARC: Draw's an arc using three specified points on the circumference.

COMMAND: ARC or A. centre / <start point>; specify a point enter C.

COMMAND: ARC or A.

Center/<start point>; 100, 100.

Center/ end/<second point>; 50, 150.

End point; 100, 100.

or pick first point, 2nd point and 3rd point using mouse.

COMMAND: ARC or A.

Center/<start point>; 100, 100.

Center; 20, 40.

Angle /l length off the cord / < end point >; 20, 60.

or pick start point, center of arc and end point of arc using mouse.

D.PLINE COMMAND (POLY LINE): Poly line is a series of connected line and are segments created as one object width of the object can be controlled.

COMMAND: PLINE or PL.

From point; specify a point.

Specify a point (2) or enter an option.

End point of line; Draw a line segment,

Arc; changes P LINE to ARC mode.

Close; close a poly line

Half width; specify width from center of a wide poly line segment to one of its edges.

Length; draw a line segment of a specified length.

Undo; removes the most recent line segment.

Width; specify width of the next line segment.

E.MLINE COMMAND: It is used to create a set of parallel lines with different properties in a single go.

COMMAND: MLINE or ml.

F.POLYGON COMMAND: It creates a regular polygon with the given number of sides and side length.

COMMAND: POLYGON or pol.

RECTANGLE COMMAND: It creates a rectangle. This rectangle acts as one object.

COMMAND: RECTANGLE or rec.

Chamfer/ Elevation/ Fillet/ Thickness/ Width/ <first corner>; pick first corner.

Other corner; pick opposite corner.

G.SPLINE COMMAND: It is used to draw smooth curves that possess through the fit points.

COMMAND: SPLINE or spl.

H.ELLIPSE COMMAND: It is used to an ellipse by selecting three points.

COMMAND: ELLIPSE or el.

1.12 MODIFY TOOLS

It consists of a set of commands that can be used to alter the existing objects.

LERASE COMMAND: It deletes the selected objects from drawing.

COMMAND: ERASE or e.

J.COPY COMMAND: It creates one or more number of copies of selected objects within the drawing.

COMMAND: COPY or co.

K.MIRROR COMMAND: It creates mirror image of the selected object in selected direction. It helps, to complete drawing faster if the object is symmetrical about any axis.

COMMAND: MIRROR or MI

Select object; pick objects to mirror. First point of mirror line; pick a point. Second point; pick second point.

Delete old objects? <N>; enter Y for yes or N for no. If y is entered deletes the original objects.

L.OFFSET COMMAND: It creates a new object that which is similar to a selected object at a distance from the original object.

COMMAND: OFFSET or O.

Offset distance or through <current>; specify a distance or enter 'T' press enter.

Select object to offset; select an object, side of object, specify a side to which object is to be created. If 'T' is entered.

Select object to offset; select one object.

Through point; pick a point where offset object is required.

M. ARRAY COMMAND: It creates multiple copies of objects in given number of rows and columns or around an imaginary circle.

COMMAND: ARRAY or ar.

N.MOVE COMMAND: Objects can be shifted from one place to another place within the drawing area.

COMMAND: MOVE or M

Select object: select objects to move and press enter.

Base point or displacement: specify a base point 1.

Second point or displacement: specify a second point 2.

O.ROTATE COMMAND: Rotates selected object around given axis to the given angle or about a base point.

COMMAND: ROTATE or RO

Select object: select object to rotate.

Base point: select a point on which object rotates.

<Rotate angle>/reference: specify a scale factor or type 'R'

Reference length <L>; specify a distance or press enter.

New length; specify a distance or pick point.

P. SCALE COMMAND: It enlarges or reduces the size of the objects equal in x, y, z. Directions according to the scale factor given. If scale factor is greater than 1, objects are enlarged and if it is less than 1 object is reduced.

COMMAND: SCALE or SC.

Select objects; select an object to be scaled, press enter.

Base point; specify a point on object.

<Scale factor>/reference; specify a scale factor or type 'R'.

Reference length; specify the distance or press enter.

New length; specify a distance or pick a point.

Q.STRETCH COMMAND: To change the snap and size of the object by pulling or pushing from one side and also to move objects from one place to another place.

COMMAND: STRETCH.

Select objects; select objects by making crossing windows only, press enter. Base point or displacement; specify a point.

Second point of displacement; specify a point.

Line to be stretched.

R.TRIM COMMAND: To cutoff or erase an object precisely at an edge defined by other objects. It can also be used in.

COMMAND: TRIM or TR.

Select object: select the boundary edges for trimming.

<Select object to trim >/project/edges/ undo; select object to be trim or enter option.

Project; used in 3D space.

Edge; determine whether the object is trimmed to another object.
Extend; extends the cutting edge along its natural path and cuts the objects.
No extend; cuts the object only when the cutting edge physically intersects with the object to be trimmed.

Undo; reverse the most recent change made by trim.

S. EXTEND COMMAND: It extends an object to meet another object

COMMAND: EXTEND or EX.

Select object; select object until another object is to be extended, press enter.

< Select object to extend>/project/ edge/ undo; select object to be extended or enter option.

Project; used in 3D space.

Edge; determine whether the object is extended to another object. Extend/ No extend < current >; enter option.

Extend; extends the ends even though the boundary is not externally intersecting with it after extending. No extend; object will not extend.

Undo; reverse the most recent modes changes by extend.

T.BREAK COMMAND: This method is used to cut an object into two parts at selected point (or) to remove part of the object in between two selected points.

COMMAND: BREAK or BR.

Select object; select an object or specify the first break point of an object.

Enter second point; specify the second break point.

U.CHAMFER COMMAND: Connects two non-parallel lines by extending them to intersect or to join with a beveled line at specified distance from intersection.

COMMAND: Chamfer or CHA.

Enter first chamfer distance <current>; specify a distance or press enter.

Enter second chamfer distance current>; specify a distance or press enter.

COMMAND: FILLET.

Polyline/ radius/ trim/ <select first object >; select the first object or press enter.

Select second object; select second object, if 'R' is entered.

Enter fillet radius < current >; enter radius required for round arc.

Polyline; insert fillet arc at each corner of 2D objects where two line segment meet.

Trim: controls whether auto cad trims the selected edges to fillet.

No trim; does not trim the selected edge.

V.FILLET COMMAND: Filletting connects two objects with a round arc of a specified radius.