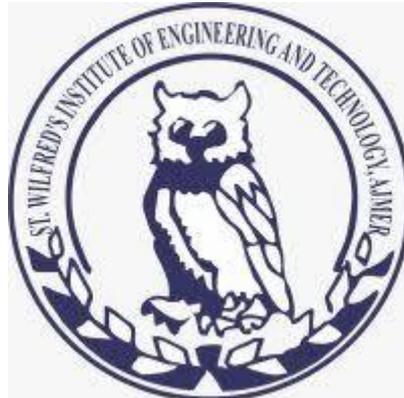


# LAB MANUAL

**Lab Name** : CAD Lab  
**Lab Code** : 8ME6A  
**Branch** : Mechanical Engineering  
**Year** : 4<sup>th</sup> Year



Department of Mechanical Engineering  
**St. Wilfred's Institute of Engineering & Technology, Ajmer**  
(RTU, Kota)

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**Experiment No. 1**

**OBJECTIVE:** Introduction to the different features of CAD softwares.

**DESCRIPTION:**

Computer Aided Drafting is a process of preparing a drawing of an object on the screen of a computer. There are various types of drawings in different fields of engineering and sciences. In the fields of mechanical or aeronautical engineering, the drawings of machine components and the layouts of them are prepared. In the field of civil engineering, plans and layouts of the buildings are prepared. In the field of electrical engineering, the layouts of power distribution system are prepared. In all fields of engineering use of computer is made for drawing and drafting.

The use of CAD process provides enhanced graphics capabilities which allows any designer to

Conceptualize his ideas  
Modify the design very easily  
Perform animation  
Make design calculations  
Use colours, fonts and other aesthetic features.

**REASONS FOR IMPLEMENTING A CAD SYSTEM:**

1. Increases the productivity of the designer: CAD improves the productivity of the designer to visualize the product and its component, parts and reduces the time required in synthesizing, analyzing and documenting the design
2. Improves the quality of the design: CAD system improves the quality of the design. A CAD system permits a more detailed engineering analysis and a larger number of design alternatives can be investigated. The design errors are also reduced because of the greater accuracy provided by the system

3. Improves communication: It improves the communication in design. The use of a CAD system provides better engineering drawings, more standardization in the drawing, and better documentation of the design, few drawing errors and legibility.
4. Create data base for manufacturing: In the process of creating the documentation for these products, much of the required data base to manufacture the products is also created.
5. Improves the efficiency of the design: It improves the efficiency of the design process and the wastage at the design stage can be reduced.

#### **APPLICATION OF CAD:**

There are various processes which can be performed by use of computer in the drafting process.

1. Automated drafting: This involves the creation of hard copy engineering drawings directly from CAD data base. Drafting also includes features like automatic dimensioning, generation of cross – hatched areas, scaling of the drawing and the capability to develop sectional views and enlarged views in detail. It has ability to perform transformations of images and prepare 3D drawings like isometric views, perspective views etc.,
2. Geometric modeling: concerned with the computer compatible mathematical description of the geometry of an object. The mathematical description allows the image of an object to be displayed and manipulated on a graphics terminal through signals from the CPU of the CAD system. The software that provides geometric modeling capabilities must be designed for efficient use both by computer and the human designer.

#### **BENEFITS OF CAD:**

The implementation of the CAD system provides variety of benefits to the industries in design and production as given below:

1. Improved productivity in drafting
2. Shorter preparation time for drawing
3. Reduced man power requirement

4. Customer modifications in drawing are easier
5. More efficient operation in drafting
6. Low wastage in drafting
7. Minimized transcription errors in drawing
8. Improved accuracy of drawing
9. Assistance in preparation of documentation
10. Better designs can be evolved
11. Revisions are possible
12. Colors can be used to customize the product
13. Production of orthographic projections with dimensions and tolerances
14. Hatching of all sections with different filling patterns
15. Preparation of assembly or sub assembly drawings
- . Preparation of part list
17. Machining and tolerance symbols at the required surfaces
18. Hydraulic and pneumatic circuit diagrams with symbols
19. Printing can be done to any scale

**LIMITATIONS OF CAD:**

1. 32 – bit word computer is necessary because of large amount of computer memory and time
2. The size of the software package is large
3. Skill and judgment are required to prepare the drawing
4. Huge investment

**CAD SOFTWARES**

The software is an interpreter or translator which allows the user to perform specific type of application or job related to CAD. The following softwares are available for drafting

- |                |                    |               |
|----------------|--------------------|---------------|
| 1. AUTOCAD     | 2. Pro – E         | 3. CATIA      |
| 4. MS OFFICE   | 5. PAINT           | 6. ANSYS      |
| 7. MSc.NASTRAN | 8. IDEAS           | 9. SOLIDWORKS |
| 10. HYPERMESH  | 11. FLUENT –GAMBIT |               |

**EXPECTED OUTCOME:**

**COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1- What is meant by AutoCAD?

Q2-What are the applications of CAD?

Q3-Give some example of latest CAD software.

Q4- What are the benefits of CAD?

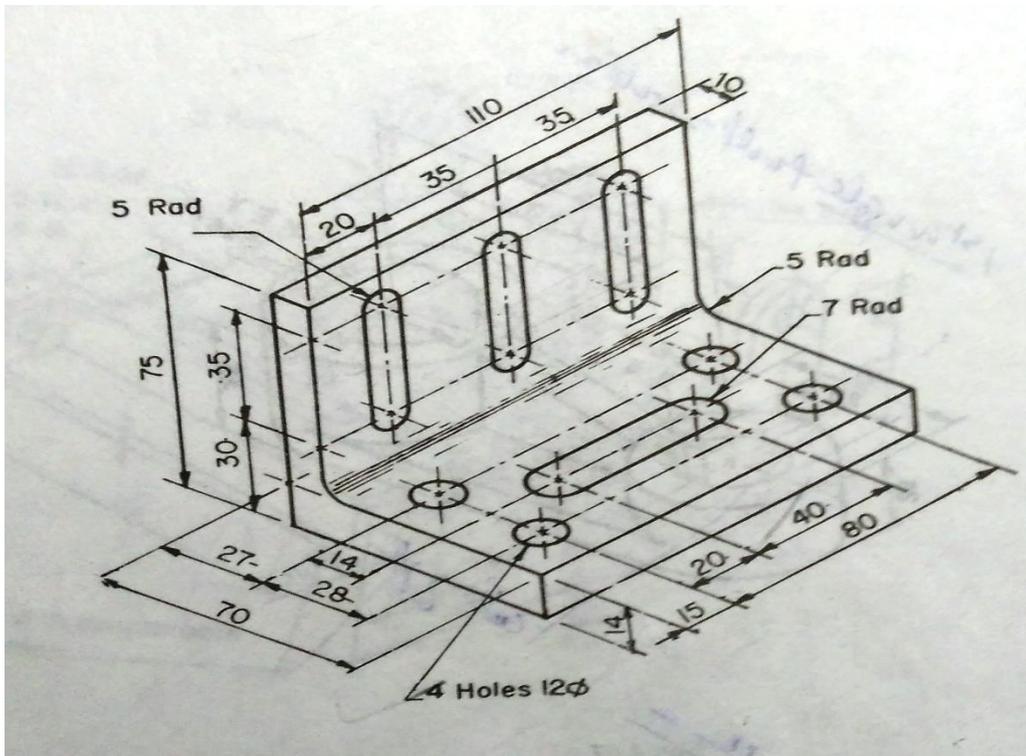
Q5-Give any one limitation of CAD.

**Experiment No. 2**

**OBJECTIVE:** To make a given 2-D drawing using CAD software.

**DESCRIPTION:**

Make orthographic projections by the drawing given below.



**COMMANDS USED:**

**LINE:**

A line is specified by giving its two end points or first point and the distance of line along with its angle of inclination. A line can be drawn by using two commands.

Command: **line**

Specify first point: Specify a point (1)

Specify next point or [Undo]: Specify a point (2)

The second point can be indicated by @d<a

Where **d** is the distance of line and **a** is the angle of inclination in degrees.

**PLINE:**

This is a poly line which allows continuous segment of the line and it is drawn similar to the line command. The polyline allows changing the thickness of the line according to the requirement.

From the Draw tool bar choose the Polyline flyout.

Draw pull down menu: **Polyline**

At the command prompt, enter **pline**

**Syntax**

Specify start point: Specify a 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

**LINETYPE**

Creates, loads, and sets linetypes. The LINETYPE command defines line characteristics consisting of dashes, dots, and spaces.

Format menu: Linetype or Command line: linetype

**CIRCLE:** This command draws the circle by using four methods:

Center point and radius

Two point circle

Three point circle

Tangent circle

At the command prompt, enter **circle**

**Specify centre point for circle or [3P (Three Points)/2P (Two Points)/Ttr]: Specify a point or enter an option**

**TRIM:** Trims objects at a cutting object defined by other objects. Objects that can be trimmed include arcs, circles, elliptical arcs, lines, open 2D and 3D polylines, rays and splines

At the command prompt, enter **trim**

Select cutting edges:

Select objects: use object selection method

<Select object to trim>/project/edge/undo: select an object, enter an option, or press enter

**BREAK:** This erases an object or splits the object in to two parts

From the modify toolbar select break flyout

At the command prompt, enter **break**

**ERASING OF OBJECT:**

The object can be removed or erased by use of erase command

**ERASE**

This removes object from drawing

At the command prompt, enter **erase**

**Select objects: use an object selection method.**

**DIMENSIONING IN DRAWINGS:**

The dimensions are inserted in the drawing by use of DIM command. There are various types of dimensions used in AutoCAD.

**1. Linear dimensions:**

Horizontal- this allows horizontal dimensions

Vertical- this allows vertical dimensions

Aligned- this allows inclined dimensions

Rotated- this allows inclined dimensions

**2. Angular dimensions:**

This allows angular dimensioning of objects

**3. Radial dimensions:**

This allows radial dimensioning of arc or circle

**4. Diametric dimensions:**

**FILLET:**

Rounds and fillets the edges of the object

At the command prompt enter **fillet**

Polyline / Radius / Trim / <Select first object>: use an object selection method or enter an option

Select first object

Select second object: use an object selection method

Enter radius <current>: specify a distance or press

Chain / Radius <Select edge>: Select edges or enter **c** or **r** their intersection

**EXPECTED OUTCOME:**

**COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1- What is orthographic projection?

Q2-What is the use of LINE command?

Q3- What is the use of FILLET command?

Q4- Which command is use for dimensioning?

Q5-What is the use of PLINE?

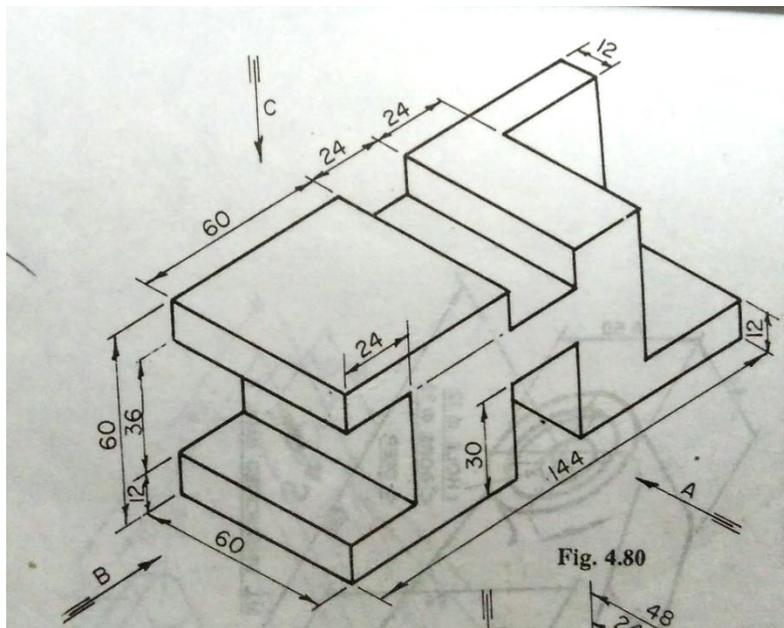
Q6-Define LINETYPE command?

### Experiment No. 3

**OBJECTIVE:** To perform 3-D modeling.

**DESCRIPTION:**

Make a 3-D Model as per given in the figure below.



**COMMANDS USED:**

Few previously used commands

**CO-ORDINATE SYSTEM**

The co-ordinate system can be modified in the AutoCAD. There are two types of co-ordinate systems used. The WCS (World co-ordinate system) is a universal system in which its origin is at the fixed position. The UCS (User co-ordinate system) is a system in which user can fix his origin at any point.

1. UCS : This manages user co- ordinate systems

At the command prompt enter **UCS**

Origin / z axis/ 3 point/ object/ view/ X/Y/Z / Prev/ Restore/Save/ Del/?/< world>: enter an option or press enter

2. WCS: This manages world co- ordinate system

**TRANSFORMATIONS:** These are the modifications in the drawn objects.

There are different types of transformations used

**1. MOVE:** This allows to move or displace objects a specified distance in a specified direction

At the command prompt, enter **move**

Select objects: use an object selection method

Base point or displacement: specify a base point (1)

Second point of displacement: specify a point (2) or press enter

**2. COPY:** This is used for producing a duplicate copy of the drawing.

At the command prompt, enter **copy**

Select objects: use an object selection method

<Base point or displacement >/multiple: specify a base point (1)

For a single copy or enter m for multiple copies

**3. ROTATE:** It moves objects about a base point

At the command prompt, enter **rotate**

Select objects: use an object selection method

<Rotate angle >/reference: specify an angle or enter r

**4. STRETCH:** This moves or stretches objects .AutoCAD stretches lines, arcs, elliptical arcs, splines, rays and polyline segments that cross the selection window.

At the command prompt, enter **stretch**

Select objects: use the CPOLYGON or cross object selection method (1,2)

Base point or displacement: specify a point (3) or press

Second point of displacement: specify a point (\$) or press

**5. EXTEND:** This extends an object to meet another object. Objects that can be extended

include arcs, elliptical arcs, lines, open 2D, and 3Dpolylines and rays.

At command prompt, enter **extend**

Select boundary edges

(projmode=UCS, edge mode=no extend)

Select objects: use an object selection method

**6. SCALE:** This enlarges or reduces selected objects equally in X and Y directions

At the command prompt, enter **scale**

Select objects: use an object selection method

Base point: specify a point (1)

<Scale factor>/reference: specify a scale or enter r

**7. TRACE:** This creates solid lines.

From the miscellaneous tool bar choose

At the command prompt, enter **trace**

Trace width<current>: specify a distance, enter a value, or press enter

From point: specify point (1)

To point: specify a point (2)

To point: specify a point (3) or press to end the command

**8. EXTRUDE:** This creates unique solid primitives by extruding existing two-dimensional objects extrudes also creates solids by extruding two-dimensional objects along a specified path .we can extrude multiple objects with extrude

At the command prompt enter, **extrude**

Select objects: use an object selection method

Path/<height of extrusion>: specify a distance or enter p

**9. MIRROR:** This is used to producing mirror image of the object

At the command prompt enter, **mirror**

Select objects: use an object selection method

First point of the mirror line: specify a point (1)

Second point: specify a point (2)

**10. OFFSET:** This creates concentric circles ,parallel lines and parallel curves, offset creates a new object at a specified distance from an existing object or through a specified point

At the command prompt enter, **offset**

Offset distance: specify a distance, enter t or press enter

**11. ARRAY:** This creates multiple copies of objects in pattern.

Each object in an array can be manipulated independently

At the command prompt enter, **array**

Rectangular or polar array<current>: enter an option or press enter specify a point

**12. UNION:**

At the command prompt, enter **union**

Select object: Use an object selection method

**13. SUBTRACT:**

At the command prompt, enter **union**

Select object: Use an object selection method

**EXPECTED OUTCOME:**

**COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1-What do you know about coordinate system?

Q2-What is the use of EXTRUDE command?

Q3-What is the difference between UNION and SUBTRACT command?

Q4-What is the use of OFFSET command?

Q5-Write syntax for MIRROR command?

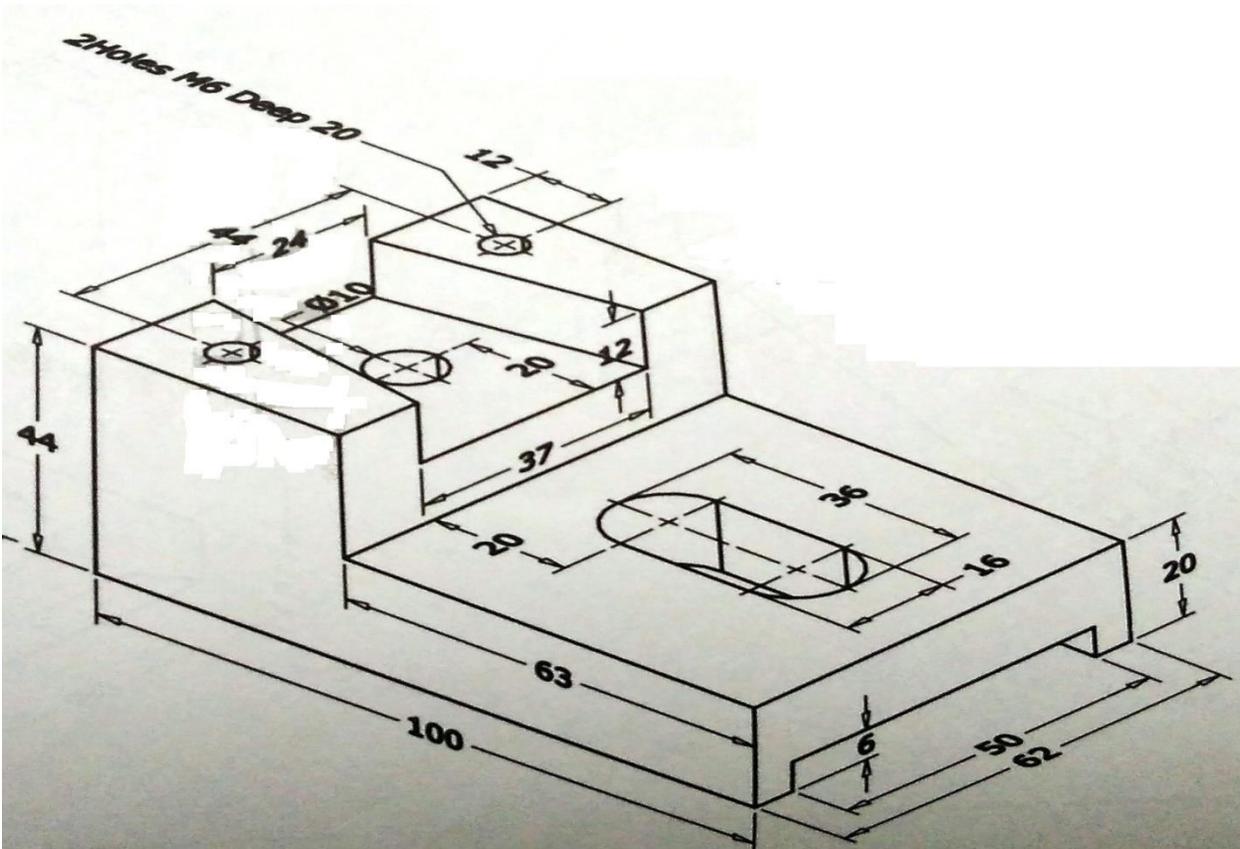
Q6-What is the use of ARRAY command?

**EXPERIMENT NO. 4**

**OBJECTIVE:** To perform advanced 3-D modelling.

**DESCRIPTION:**

Make a 3-D Model as per the given drawing.



**COMMANDS USED:**

Few previously used commands

### **CO-ORDINATE SYSTEM**

The co- ordinate system can be modified in the AutoCAD. There are two types of co- ordinate systems used. The WCS (World co- ordinate system) is a universal system in which its origin is at the fixed position. The UCS (User co- ordinate system) is a system in which user can fix its origin at any point.

1. UCS : This manages user co- ordinate systems

At the command prompt enter **UCS**

Origin / z axis/ 3 point/ object/ view/ X/Y/Z / Prev/ Restore/Save/ Del/?/< world>: enter an option or press enter

2. WCS: This manages world co- ordinate system

**TRANSFORMATIONS:** These are the modifications in the drawn objects.

There are different types of transformations used

**1. MOVE:** This allows to move or displace objects a specified distance in a specified direction

At the command prompt, enter **move**

Select objects: use an object selection method

Base point or displacement: specify a base point (1)

Second point of displacement: specify a point (2) or press enter

**2. COPY:** This is used for producing a duplicate copy of the drawing.

At the command prompt, enter **copy**

Select objects: use an object selection method

<Base point or displacement >/multiple: specify a base point (1)

For a single copy or enter m for multiple copies

**3. ROTATE:** It moves objects about a base point

At the command prompt, enter **rotate**

Select objects: use an object selection method

<Rotate angle >/reference: specify an angle or enter r

**4. STRETCH:** This moves or stretches objects .AutoCAD stretches lines, arcs, elliptical arcs, splines, rays and polyline segments that cross the selection window.

At the command prompt, enter **stretch**

Select objects: use the CPOLYGON or cross object selection method (1,2)

Base point or displacement: specify a point (3) or press

Second point of displacement: specify a point (\$) or press

**5. EXTEND:** This extends an object to meet another object. Objects that can be extended include arcs, elliptical arcs, lines, open 2D, and 3Dpolylines and rays.

At command prompt, enter **extend**

Select boundary edges

(projmode=UCS, edge mode=no extend)

Select objects: use an object selection method

**6. SCALE:** This enlarges or reduces selected objects equally in X and Y directions

At the command prompt, enter **scale**

Select objects: use an object selection method

Base point: specify a point (1)

<Scale factor>/reference: specify a scale or enter r

**7. TRACE:** This creates solid lines.

From the miscellaneous tool bar choose

At the command prompt, enter **trace**

Trace width<current>: specify a distance, enter a value, or press enter

From point: specify point (1)

To point: specify a point (2)

To point: specify a point (3) or press to end the command

**8. EXTRUDE:** This creates unique solid primitives by extruding existing two-dimensional objects extrudes also creates solids by extruding two-dimensional objects along a specified path .we can extrude multiple objects with extrude

At the command prompt enter, **extrude**

Select objects: use an object selection method

Path/<height of extrusion>: specify a distance or enter p

**9. MIRROR:** This is used to producing mirror image of the object

At the command prompt enter, **mirror**

Select objects: use an object selection method

First point of the mirror line: specify a point (1)

Second point: specify a point (2)

**10. OFFSET:** This creates concentric circles ,parallel lines and parallel curves, offset creates a creates a new object at a specified distance from an existing object or through a specified point

At the command prompt enter, **offset**

Offset distance: specify a distance, enter t or press enter

**11. ARRAY:** This creates multiple copies of objects in pattern.

Each object in an array can be manipulated independently

At the command prompt enter, **array**

Rectangular or polar array<current>: enter an option or press enter specify a point

**12. UNION:**

At the command prompt, enter **union**

Select object: Use an object selection method

**13. SUBTRACT:**

At the command prompt, enter **union**

Select object: Use an object selection method

**14. EXPLODE:**

This breaks a compound object into its component objects

At the command prompt enter **explode**

Select objects: use an object selection method.

**15. REGENERATION OF DRAWING:**

ACAD provides a facility of regenerating a drawing to clear the cross points or marks on the screen.

REDRAW

REGEN

REGENALL

REGENAUTO

**EXPECTED OUTCOME:**

**COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1- What is WCS and UCS?

Q2-Which command is use to breaks a compound object into its component objects?

Q3-What is regeneration of drawing?

Q4-Give some example for transformation?

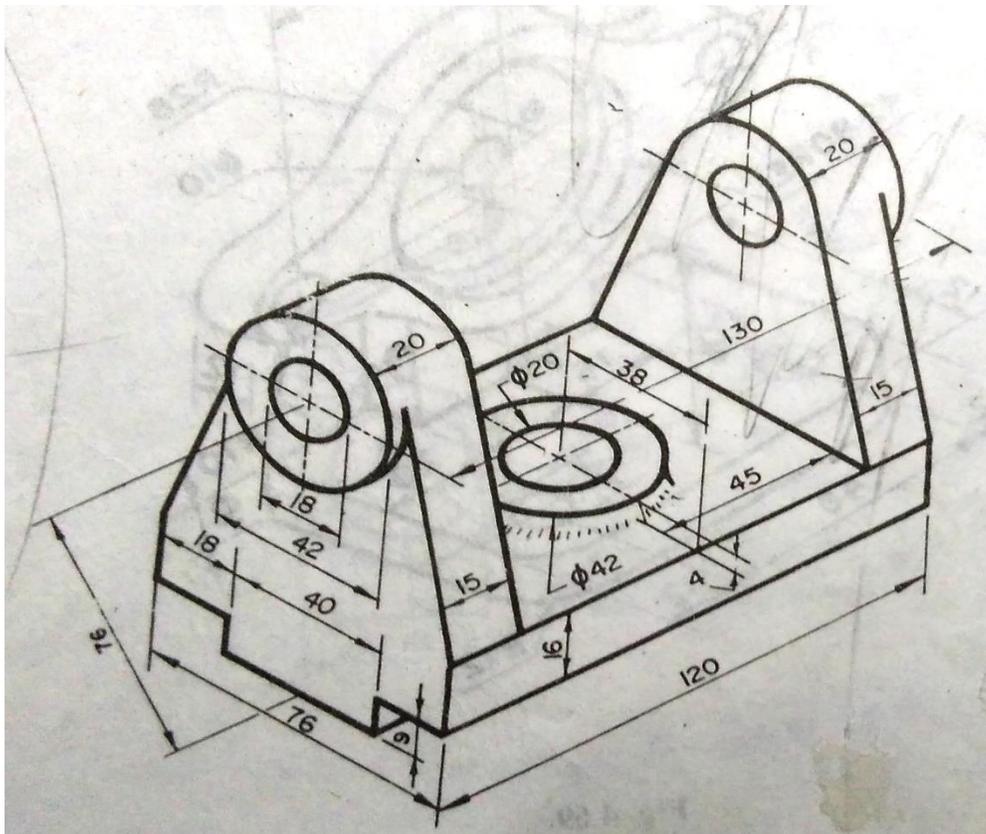
Q5-What do you know about 3-D modelling

### Experiment No. 5

**OBJECTIVE:** To make an assembly drawing using different parts.

### DESCRIPTIONS:

Make an assembly drawing using the drawing given below.



**COMMANDS USED:-**

**3D FUNCTIONS**

#### 1. BOX

This creates a three dimensional solid box.

At the command prompt enter **box**

Center/<corner of the box><0,0,0> :

Specify a point (1), enter c, or press enter

Corner of a box

Specifying a point or pressing defines the first corner of the box.

Cube/length /<other corner>: specify a point (2) or enter an option **center**

Creates the box by a specified center point

## 2. CONE

This creates a 3D solid cone. A cone is solid primitive with a circular or elliptical based tapering symmetrically to a point perpendicular to its base.

At the command prompt enter **cone**

Elliptical /<center point> <0,0,0>: specify a point, enter e or press enter

## 3. CYLINDER

This creates a 3D solid cylinder. A cylinder is solid primitive with a circular or elliptical based to a point perpendicular to its base without a taper.

At the command prompt enter **cylinder**

Elliptical /<center point> <0,0,0> : specify a point, enter e or press enter

## 4. SPHERE

This creates a 3D solid sphere. A sphere is positioned so that its central axis is parallel to the Z-axis of the current UCS. Latitudinal lines are parallel to the XY plane.

At the command prompt enter **sphere**

center of the sphere <0,0,0> : specify a point , enter e or press enter

## 5. WEDGE

This creates a three dimensional solid with a sloped face tapering along X axis.

At the command prompt enter **wedge**

Center <corner of the wedge> <0,0,0> : specify a point, enter e or press enter

Follow the prompting

## **6. ELEV**

This sets an elevation and extrusion thickness of new objects. The current elevation is the Z value that is used whenever a 3D point is expected but only X and y values are supplied.

At the command prompt enter **elev**

Follow the prompting

## **7. SHADE**

This displays a flat shaded image of the drawing in the current view port. SHADE removes hidden lines and displays a shaded picture of the drawing.

From the render toolbar, choose

At the command prompt, enter **shade**

## **8. REGION**

This creates a region object from a selection set of existing objects. Regions are 2Dimensional areas you create from closed shapes.

## **9. REINIT**

This reinitializes the input/output ports, digitizer, display and program parameters file.

## **10. REPLAY**

This displays a GIF, TGA or TIFF image.

From the tools menu, choose image, then view.

## **11. REVOLVE**

This creates a solid by revolving a two – dimensional object about an axis. From the solids toolbar, choose

At the command prompt, enter **revolve**

## **12. SHAPE**

This inserts a shape. Before inserting a shape, you must load the file containing the desired shape.

## **13. ROTATE 3D**

This moves objects about a three dimensional axis  
From the modify toolbar, choose the rotate flyout then  
Follow the prompting

#### **14. SECTION**

This uses the intersection of a plane and solids to create a region.  
AutoCAD creates regions on the current layer and inserts them at the location of the cross-section. Selecting several solids creates separate regions for each solid.

#### **15. SLICE**

This slices a set of solids with a plane.

#### **16. SHELL**

This accesses operating system commands.

#### **17. REVOLVE**

This creates a solid by revolving a two dimensional object about an axis.

#### **18. RENDER**

This creates a realistically shaded image of a three dimensional wireframe or solid model.  
RENDER produces an image using information from a scene, the current selection set, or the current view.

**TRANSFORMATIONS:** These are the modifications in the drawn objects.

There are different types of transformations used

**1. MOVE:** This allows to move or displace objects a specified distance in a specified direction

At the command prompt, enter **move**

Select objects: use an object selection method

Base point or displacement: specify a base point (1)

Second point of displacement: specify a point (2) or press enter

**2. COPY:** This is used for producing a duplicate copy of the drawing.

At the command prompt, enter **copy**

Select objects: use an object selection method

<Base point or displacement >/multiple: specify a base point (1)

For a single copy or enter m for multiple copies

**3. ROTATE:** It moves objects about a base point

At the command prompt, enter **rotate**

Select objects: use an object selection method

<Rotate angle >/reference: specify an angle or enter r

**4. STRETCH:** This moves or stretches objects. AutoCAD stretches lines, arcs, elliptical arcs, splines, rays and polyline segments that cross the selection window.

At the command prompt, enter **stretch**

Select objects: use the CPOLYGON or cross object selection method (1,2)

Base point or displacement: specify a point (3) or press

Second point of displacement: specify a point (\$) or press

**5. EXTEND:** This extends an object to meet another object. Objects that can be extended include arcs, elliptical arcs, lines, open 2D, and 3D polylines and rays.

At command prompt, enter **extend**

Select boundary edges

(projmode=UCS, edge mode=no extend)

Select objects: use an object selection method

**6. SCALE:** This enlarges or reduces selected objects equally in X and Y directions

At the command prompt, enter **scale**

Select objects: use an object selection method

Base point: specify a point (1)

<Scale factor>/reference: specify a scale or enter r

**7. TRACE:** This creates solid lines.

From the miscellaneous tool bar choose

At the command prompt, enter **trace**

Trace width<current>: specify a distance, enter a value, or press enter

From point: specify point (1)

To point: specify a point (2)

To point: specify a point (3) or press to end the command

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At the command prompt enter, **extrude**

Select objects: use an object selection method

Path/<height of extrusion>: specify a distance or enter p

**9. MIRROR:** This is used to producing mirror image of the object

At the command prompt enter, **mirror**

Select objects: use an object selection method

First point of the mirror line: specify a point (1)

Second point: specify a point (2)

**10. OFFSET:** This creates concentric circles ,parallel lines and parallel curves, offset creates a creates a new object at a specified distance from an existing object or through a specified point

At the command prompt enter, **offset**

Offset distance: specify a distance, enter t or press enter

**11. ARRAY:** This creates multiple copies of objects in pattern.

Each object in an array can be manipulated independently

At the command prompt enter, **array**

Rectangular or polar array<current>: enter an option or press enter specify a point

**EXPECTED OUTCOME:**

**COMMENT BY STUDENT:**

### **VIVA VOCE**

Q1- What is the use of TRACE command?

Q2-Which command is use for enlarge and reduce to any object?

Q3-Write syntax for following command

BOX.REVOLVE,RENDER,SLICE

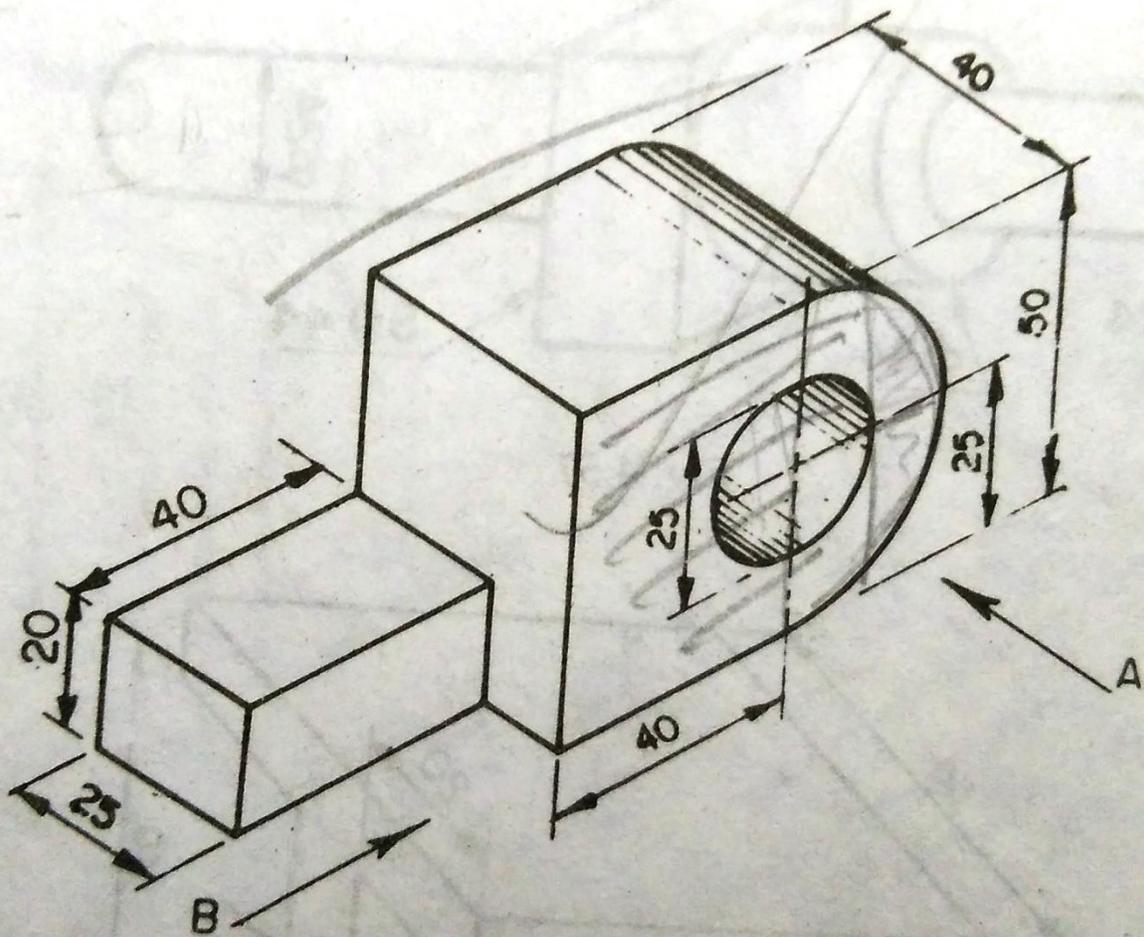
Q4-What is the use of SHAPE command?

**Experiment no. 6**

**OBJECTIVE:** Editing a drawing using feature modification and Manipulation.

**DESCRIPTION:**

Edit the given according to dimensions chosen by you.



**Commands used:**

Few previous commands.

**TRANSFORMATIONS:** These are the modifications in the drawn objects.

There are different types of transformations used

**1. MOVE:** This allows to move or displace objects a specified distance in a specified direction

At the command prompt, enter **move**

Select objects: use an object selection method

Base point or displacement: specify a base point (1)

Second point of displacement: specify a point (2) or press enter

**2. COPY:** This is used for producing a duplicate copy of the drawing.

At the command prompt, enter **copy**

Select objects: use an object selection method

<Base point or displacement >/multiple: specify a base point (1)

For a single copy or enter m for multiple copies

**3. ROTATE:** It moves objects about a base point

At the command prompt, enter **rotate**

Select objects: use an object selection method

<Rotate angle >/reference: specify an angle or enter r

**4. STRETCH:** This moves or stretches objects .AutoCAD stretches lines, arcs, elliptical arcs, splines, rays and polyline segments that cross the selection window.

At the command prompt, enter **stretch**

Select objects: use the CPOLYGON or cross object selection method (1,2)

Base point or displacement: specify a point (3) or press

Second point of displacement: specify a point (\$) or press

**5. EXTEND:** This extends an object to meet another object. Objects that can be extended include arcs, elliptical arcs, lines, open 2D, and 3Dpolylines and rays.

At command prompt, enter **extend**

Select boundary edges

(projmode=UCS, edge mode=no extend)

Select objects: use an object selection method

**6. SCALE:** This enlarges or reduces selected objects equally in X and Y directions

At the command prompt, enter **scale**

Select objects: use an object selection method

Base point: specify a point (1)

<Scale factor>/reference: specify a scale or enter r

**7. TRACE:** This creates solid lines.

From the miscellaneous tool bar choose

At the command prompt, enter **trace**

Trace width<current>: specify a distance, enter a value, or press enter

From point: specify point (1)

To point: specify a point (2)

To point: specify a point (3) or press to end the command

**8. EXTRUDE:** This creates unique solid primitives by extruding existing two-dimensional objects extrudes also creates solids by extruding two-dimensional objects along a specified path .we can extrude multiple objects with extrude

At the command prompt enter, **extrude**

Select objects: use an object selection method

Path/<height of extrusion>: specify a distance or enter p

**9. MIRROR:** This is used to producing mirror image of the object

At the command prompt enter, **mirror**

Select objects: use an object selection method

First point of the mirror line: specify a point (1)

Second point: specify a point (2)

**10. OFFSET:** This creates concentric circles ,parallel lines and parallel curves, offset creates a new object at a specified distance from an existing object or through a specified point

At the command prompt enter, **offset**

Offset distance: specify a distance, enter t or press enter

**11. ARRAY:** This creates multiple copies of objects in pattern.

Each object in an array can be manipulated independently

At the command prompt enter, **array**

Rectangular or polar array<current>: enter an option or press enter specify a point

**EXPECTED OUTCOME:**

**COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1- What do you know about manipulation?

Q2-What is the use of STRETCH command?

Q3- What is the use of EXTEND command?

Q4-What is the important feature for modification of drawing?

Q5- What are the use of ROTATE and MOVE command?



**Isocircle:** Creates an isometric circle in the current isometric drawing plane.

At the command prompt, enter **ellipse**

**Arc:** The arc is a curve specified by center and radius as well as the start angle and end angle.

There are seven methods used for drawing an arc.

1. Three point method
2. Start point-centre point –end point
3. Start point-centre point-length of chord
4. Start point-end point –angle of inclusion
5. Start point-end point-direction
6. Start point-centre point-angle of inclusion
7. Start point-end point-radius

These methods can be used by executing the arc command

**ARC:** creates an arc.

At the command prompt, enter arc

Center/<start point>: specify a point, enter c, or press enter

**Polyarc:** the second method of drawing the arc is poly arc by use of pline command.

This command allows drawing of filled arc of any width. It also allows for drawing of a regular or irregular curve.

**RECTANGLES:** draws a rectangular polyline

At the command prompt, enter **rectangle**

First corner: specify point (1)

Other corner: specify point (2)

**SOLID:** creates solid –filled polygons. Solids are filled only when fill system variable is set to on view is set to plan.

At the command prompt, enter **solid**

First corner: specify point (1)

Other corner: specify point (2)

The first two points define one edge of the polygon.

Third point: specify a point (3) diagonally opposite the second

Fourth point: specify a point (4) or press enter

### **DRAWING OF POLYGON**

Creates an equilateral closed polyline .A polygon is a polyline object. AUTOCAD draws polyline with zero width and no tangent information.

At the command prompt enter **polygon**

Number of sides <current>: enter a value between **3 and 1024** or press enter

Edge/<center of polygon>: **specify a point (1) or enter.**

### **POINT**

Creates a point object .points can act as nodes to which you can snap objects .you can specify a full 3D location for a point.

At the command prompt, enter **point**

Point: specify a point

### **HATCH**

This fills an area with a pattern.

HATCH fills the specified hatch boundary with non-associative hatch

A non –associative hatch is not updated when its boundaries are modified .a hatch boundary consists of an object or objects that completely enclose an area

At the command prompt, enter hatch

Pattern (? Or name/ U, style) <current>: enter a predefined pattern name, enter u, enter? Or press enter

### **EXPECTED OUTCOME:**

### **COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1- What is the use HATCH?

Q2-Explain drawing of polygon in CAD?

Q3-Explain the procedure to create an ELLIPSE?

Q4-What is the use of SOLID in detail drawing?

## Experiment No. 8

**OBJECTIVE:** To make an object using Sheet metal.

### **DESCRIPTION:**

Make a box of given dimensions using a sheet of 0.5 mm thickness.

Length = 60 mm      Width= 40 mm      Height= 50 mm

### **COMMANDS USED:**

#### **LINE**

A line is specified by giving its two end points or first point and the distance of line along with its angle of inclination. A line can be drawn by using two commands.

Command: **line**

Specify first point: Specify a point (1)

Specify next point or [Undo]: Specify a point (2)

The second point can be indicated by @**d<a**

Where **d** is the distance of line and **a** is the angle of inclination in degrees.

#### **PLINE**

This is a poly line which allows continuous segment of the line and it is drawn similar to the line command. The polyline allows changing the thickness of the line according to the requirement.

From the Draw tool bar choose the Polyline flyout.

Draw pull down menu: **Polyline**

At the command prompt, enter **pline**

#### **Syntax**

Specify start point: Specify a 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

## **LINETYPE**

Creates, loads, and sets linetypes. The LINETYPE command defines line characteristics consisting of dashes, dots, and spaces.

Format menu: Linetype or Command line: linetype

## **CIRCLE**

This command draws the circle by using four methods:

Center point and radius

Two point circle

Three point circle

Tangent circle

At the command prompt, enter **circle**

**Specify center point for circle or [3P (Three Points)/2P (Two Points)/Ttr]: Specify a point or enter an option**

## **TRIM**

Trims objects at a cutting object defined by other objects. Objects that can be trimmed include arcs, circles, elliptical arcs, lines, open 2D and 3D polylines, rays and splines

At the command prompt, enter **trim**

Select cutting edges:

Select objects: use object selection method

<Select object to trim>/project/edge/undo: select an object, enter an option, or press enter

## **BREAK**

This erases an object or splits the object in to two parts

From the modify toolbar select break flyout

At the command prompt, enter **break**

## **ERASING OF OBJECT**

The object can be removed or erased by use of erase command

### **ERASE**

This removes object from drawing

At the command prompt, enter **erase**

**Select objects: use an object selection method.**

### **EXTRUDE**

This creates unique solid primitives by extruding existing two-dimensional objects extrudes also creates solids by extruding two-dimensional objects along a specified path .we can extrude multiple objects with extrude

At the command prompt enter, **extrude**

Select objects: use an object selection method

Path/<height of extrusion>: specify a distance or enter p

### **EXPECTED OUTCOME:**

### **COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1- What is the use EXTRUDE?

Q2-Which command is use to erase any part,write it's syntax?

Q3-Expalin sheet meatl work?

Q4-Which commands are used in sheet metal work?

### Experiment No. 9

**OBJECTIVE:** To perform surface modeling.

**DESCRIPTION:-**

To make a hollow cylindrical object of the given dimensions.

Length : 50 mm

Outer Dia: 25 mm

inner Dia: 20 mm

**COMMANDS USED:**

**CIRCLE**

This command draws the circle by using four methods:

Center point and radius

Two point circle

Three point circle

Tangent circle

At the command prompt, enter **circle**

**Specify center point for circle or [3P (Three Points)/2P (Two Points)/Ttr]: Specify a point or enter an option**

**TRIM**

Trims objects at a cutting object defined by other objects. Objects that can be trimmed include arcs, circles, elliptical arcs, lines, open 2D and 3D polylines, rays and splines

At the command prompt, enter **trim**

Select cutting edges:

Select objects: use object selection method

<Select object to trim>/project/edge/undo: select an object, enter an option, or press enter

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From the modify toolbar select break flyout

At the command prompt, enter **break**

### **ERASING OF OBJECT**

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#### **ERASE**

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### **EXTRUDE**

This creates unique solid primitives by extruding existing two-dimensional objects extrudes also creates solids by extruding two-dimensional objects along a specified path .we can extrude multiple objects with extrude

At the command prompt enter, **extrude**

Select objects: use an object selection method

Path/<height of extrusion>: specify a distance or enter p

### **EXPECTED OUTCOME:**

### **COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1- Explain surface modelling?

Q2- What is the use of BREAK command?

Q3-Write syntax for TRIM command?

Q4-Explain CIRCLE command with example?

### Experiment No. 10

**OBJECTIVE:** To solve a one dimensional problem on finite element method.

**DESCRIPTION:-**

Analyse the effect of forces and stresses on the given object.

**Software used:**

Creo 2.0

**COMMANDS USED:**

**LINE**

A line is specified by giving its two end points or first point and the distance of line along with its angle of inclination. A line can be drawn by using two commands.

Command: **line**

Specify first point: Specify a point (1)

Specify next point or [Undo]: Specify a point (2)

The second point can be indicated by @**d**<**a**

Where **d** is the distance of line and **a** is the angle of inclination in degrees.

**PLINE**

This is a poly line which allows continuous segment of the line and it is drawn similar to the line command. The polyline allows changing the thickness of the line according to the requirement.

From the Draw tool bar choose the Polyline flyout.

Draw pull down menu: **Polyline**

At the command prompt, enter **pline**

**Syntax**

Specify start point: Specify a 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

## **LINETYPE**

Creates, loads, and sets linetypes. The LINETYPE command defines line characteristics consisting of dashes, dots, and spaces.

Format menu: Linetype or Command line: linetype

## **CIRCLE**

This command draws the circle by using four methods:

Center point and radius

Two point circle

Three point circle

Tangent circle

At the command prompt, enter **circle**

**Specify center point for circle or [3P (Three Points)/2P (Two Points)/Ttr]: Specify a point or enter an option**

## **TRIM**

Trims objects at a cutting object defined by other objects. Objects that can be trimmed include arcs, circles, elliptical arcs, lines, open 2D and 3D polylines, rays and splines

At the command prompt, enter **trim**

Select cutting edges:

Select objects: use object selection method

<Select object to trim>/project/edge/undo: select an object, enter an option, or press enter

## **BREAK**

This erases an object or splits the object in to two parts

From the modify toolbar select break flyout

At the command prompt, enter **break**

## **ERASING OF OBJECT**

The object can be removed or erased by use of erase command

### **ERASE**

This removes object from drawing

At the command prompt, enter **erase**

**Select objects: use an object selection method.**

### **EXTRUDE**

This creates unique solid primitives by extruding existing two-dimensional objects extrudes also creates solids by extruding two-dimensional objects along a specified path .we can extrude multiple objects with extrude

At the command prompt enter, **extrude**

Select objects: use an object selection method

Path/<height of extrusion>: specify a distance or enter p

### **EXPECTED OUTCOME:**

### **COMMENT BY STUDENT:**

## **VIVA VOCE**

Q1- Which software is use to analyze forces on any object?

Q2-What is the use of BREAK command?

Q3What is the use of LINETYPE command?

Q4-What is the use of PLINE command?

Q5-Whrere will we use EXTRUDE command?