AUTO CAD

LABORATORY MANUAL

DEPARTMENT OF

CIVIL ENGINEERING

MANDATORY INSTRUCTIONS

1. Students should report to the labs concerned as per the timetable.

2. Record should be updated from time to time and the previous experiment must be signed by the faculty in charge concerned before attending the lab.

3. Students who turn up late to the labs will in no case be permitted to perform the experiment scheduled for the day.

4. After completion of the experiment, certification of the staff in-charge concerned in the observation book is necessary.

5. Students should bring a notebook of about 100 pages and should enter the readings/observations/results into the notebook while performing the experiment.

6. The record of observations along with the detailed experimental procedure of the experiment performed in the immediate previous session should be submitted and certified by the staff member in-charge.

7. Not more than FIVE students in a group are permitted to perform the experiment on a set up.

8. The group-wise division made in the beginning should be adhered to, and no mix up of student among different groups will be permitted later.

9. The components required pertaining to the experiment should be collected from Lab-in-charge after duly filling in the requisition form.

10. When the experiment is completed, students should disconnect the setup made by them, and should return all the components/instruments taken for the purpose.

11. Any damage of the equipment or burnout of components will be viewed seriously either by putting penalty or by dismissing the total group of students from the lab for the semester/year.

12. Students should be present in the labs for the total scheduled duration.

13. Students are expected to prepare thoroughly to perform the experiment before coming to Laboratory.

14. Procedure sheets/data sheets provided to the students groups should be maintained

neatly and are to be returned after the experiment.

15. DRESS CODE:

 1. Boys -   Formal dress with tuck in and shoes.

 2. Girls -   Formal dress (salwarkameez).

 3. Wearing of jeans is strictly prohibited

COMPUTER AIDEDRAFTING OF BUILDINGS

Experiment No. 1 INTRODUCTION TO COMPUTER AIDED DRAFTING

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 colors, fonts and other aesthetic features.

1.1 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

6. **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.

1.2 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

1.3 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. Large investment.

1.4 Pre-lab questions:

1. Full form of AUTO CAD
2. What are the benefits of auto cad?
3. What are the reasons for implementing of auto cad?

1.5 Post lab questions:

1. Briefly explain application of auto cad?
2. What are the benefits of auto cad in civil engineering point of view?
3. What are the limitations of auto cad?

Experiment No. 2 SOFTWARE FOR CAD- INTRODUCTION TO DIFFERENT SOFTWARE'S

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 software's are available for drafting

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

The above software's are used depending upon their application.

AUTO CAD

Auto CAD package is suitable for accurate and perfect drawings of engineering designs. The drawing of machine parts, isometric views and assembly drawings are possible in AutoCAD. The package is suitable for 2D and 3D drawings.

2.1 STARTING WITH AUTO CAD

CAD uses four basic elements for preparation of any drawing:

1. Line
2. Curves
3. Text
4. Filling point.

Computer Aided Drafting is done by the operator by placing the mouse pointer by placing the mouse pointer at the desired location and then executing the command to draw the graphic elements using different methods.

Advanced computer aided drafting packages utilize four areas on the screen.

1. Drawing Area
2. Command Area

3. Menu Area
4. Tool Boxes.

2.2 Layout and Sketching

The package provides various facilities for layout, sketching and borders for preparing a drawing. It provides facilities for display co-ordinates and measurement units.

a. **Units:** The format for display co – ordinates and measurement can be selected according to the requirement.

Several measurement styles are available in ACAD. The main methods are engineering and architectural, having specific base unit assigned to them.

i. Decimal: select to enter and display measurements in decimal notation

ii. Engineering: Display measurements in feet and decimal inches.

iii. Architectural: Display measurements in feet, inches and fractional inches

iv. Fractional: Display measurements in mixed numbers notation

v. Scientific: Display measurements in scientific notation.

The precision that is specified controls the number of decimal places or fractional size to which we want linear measurements displayed.

b. **Angles**: Select the format in which we want to enter and display angles.

i. Decimal Degrees: Display partial degrees as decimals

ii. Deg/Min/Sec: Display partial degrees as minutes and seconds.

iii. Grades: Display Angles as grades

iv. Radians: Display angles as radians.

v. Surveyor: Displays angles in surveyor units.

c. **Angle measure:** Select the direction of the zero angle for the entry of angles:

i. East: Select to specify the compass direction east as the zero angles.

ii. North: Select to specify the compass direction north as the zero angles.

iii. West: Select to specify the compass direction west as the zero angles.

iv. South: Select to specify the compass direction south as the zero angles.

v. Other: Select to specify a direction different from the points of the compass as the zero angles.

d. **Area:** Enter the approximate width and length which is planned to draw in full scale units. This limits the area of the drawing covered by grid dots when the grid is turned on. It also adjusts several default settings, such as text height, line type scaling and

snap distance to convenient values. It is possible to adjust these settings.

e. **Title block:** Select the description of an ACAD drawing file of a title block to insert as a symbol in the new drawing. It can add or remove drawing files of title blocks from the list with the Add or Remove buttons

f. **Layout:** Paper space is often used to create complex multiple view drawings. There are three types of paper spaces:

1. Work on the drawing while viewing the layout.
2. Work on the drawing without the layout visible
3. Work on the layout of the drawing.

The following procedure is used for this purpose

1. From the File menu or from the standard tool bar, choose New
2. In the startup dialog box, choose Use a wizard, and select Advanced wizard
3. Choose OK
4. In the Advanced Setup Dialog box, select Title Block.
5. Select Title Block Description and Title Block file Name from the lists and then choose Add.
6. In the Select Title Block File dialog box, Select a title block, then choose open
7. In the Advanced Setup dialog box, a sample of that title is displayed.
8. Choose Done.

2.3 DRAWING ENVIRONMENT

ACAD provides two drawing environments for creating and laying out the drawing.

i.   Model Space
ii.   Paper Space.

ACAD allows creating drawing, called a model, in full scale in an area known as model space without regard to the final layout or size when the drawing is plotted on the paper.

In the space opened for the first time, it is possible to create floating viewports to contain different views of the model. In the paper space, floating viewports are treated as objects which can be moved and resized in order to create a suitable layout.

LIMITS

This sets and controls the drawing boundaries.

At the command prompt, enter **limits**

ON/OFF/<LOWER LEFT CORNER> <current>: Specify a point, enter on or off, or press

enter.

LTSCALE

This sets the line type scale factor. Use LTSCALE to change the relative length of the dash – dot line types per drawing unit

At the Command prompt, enter **ltscale**

New scale factor <current>: Enter a positive real value or press enter

Changing the line type scale factor causes the drawing to regenerate.

MEASURE

This places point objects or blocks at measured intervals on an object.

At the command prompt, enter **measure**

Select object to measure: Use an object selection method / Block: Specify a distance.

PAN

This moves the drawing display in the current viewport.

At the command prompt, enter **pan**

Displacement: Specify a point (1)

The point which specify indicates the amount to move the drawing or the location of the drawing to be moved.

Second point: Press or specify a point (2)

If pressed, ACAD moves the drawing by the amount which is specified in the

Displacement prompt. If we specify a point, ACAD moves the location of the drawing to that point.

2.4 ELEMENTS OF DRAWING

2.4.1 DRAW COMMANDS

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/Half width/Length/Undo/Width]: Specify a point (2) or enter an option

LINETYPE

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

Format menu: Line type or Command line: line type

1. CURVES

Following are the various types of

i.   Circle

ii.   Ellipse

iii.   Arc

iv.   Regular or any other type.

i.   Circle: The circle can be drawn by using two types of commands

a. Circle

b. Donut

a) **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

   b) **DONUT:** This draws filled circles and rings.

Donuts are constructed of a closed polyline composed of wide arc

segments. At the command prompt, enter **donut**

Specify inside diameter of donut <current>: Specify a distance or press ENTER

If you specify an inside diameter of 0, the donut is a filled circle.

Specify outside diameter of donut <current>: Specify a distance or press ENTER

   Specify center of donut or <exit>: Specify a point (1) or press ENTER to end the command

ii.   ELLIPSE: It is a curve having major and minor axis with a center. The

ellipse can be prepared by four methods.

   ELLIPSE

   Creates an ellipse or an elliptic arc.

   Axis end point: Defines the first axis by two specified endpoints. The angle of the first
   axis determines the angle of the ellipse. The first axis can define either the major or the
   minor axis of the ellipse.
   Arc: Creates an elliptical arc. The angle of the first axis determines the angle of the
   elliptical arc. The first axis can define either the major or the minor axis of the elliptical
   arc.
   Center: Creates the ellipse by a specified center point.

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

At the command prompt, enter **ellipse**

iii. **Arc:** The arc is a curve specified by center and radius as well as the start angle and end angle. There are seven method used for drawing an arc.

1. Three point method

2. Start point-center point –end point

3. Start point-center point-length of chord

4. Start point-end point –angle of inclusion

5. Start point-end point-direction

6. Start point-center 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 the 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.

2. Drawing of Rectangle: A rectangle can be drawn by LINE command or by Rectangle command. The **PLINE** command also allows for drawing of hollow or filled rectangle .A **SOLID** command is also used for drawing of filled rectangles.

1. **RECTANGLES:** draws a rectangular polyline

   At the command prompt, enter **rectangle**

   First corner: specify point

   (1) Other corner: specify

   point (2)

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

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

3. **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**.

4. **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

5. **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.

6. COLOURING OF OBJECT:

The object can be drawn with any variety of colour which ranges from 0 to 256. The setting of colour can be done by color command

COLOR

Sets the colour for new objects.

At the command prompt, enter color <current>: enter a value (1-255), color name, by block, or by layer

7. FILLING OF OBJECT: the object can be filled with different colors and patterns by use of hatch command

This command allows selection of various patterns, scale of pattern and angle of pattern.

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.

8. SCALING OF DRAWING: zoom command displays the object at a specified scale factor. the value entered is relative to the limits of the drawing .for example ,entering 2 doubles the apparent display size of any objects from what it would be if it were zoomed to the limits of the drawing

If you enter a value followed by xp, auto CAD specifies the scale relative to paper scale unit's .for example, entering 0.5xp displays model space at half the scale of paper space unit's .the following illustration shows a number of viewports arranged in paper space. the view in each view port is scaled relative to paper space .the first view is scaled 1=1 relative to paper space (1xp) ,the second is scaled 0.5=1 relative to paper space (0.5xp),and so on.

ZOOM

This increases or decreases the apparent size of objects in the current view port

At the command prompt, enter zoom

All/center/dynamic/extents/left/previous/vmax/window/<scale(x/xp)>: enter an option or value, specify a point, or press enter.

9. **TEXT:** The text in software is indicated by font's .the fonts define the shapes of the text characters that make up each character set. In AUTOCAD, you can use true type fonts in addition to AUTOCAD's own compiled shape (SHX) fonts.

   A font is indicated by various parameters like

   i.   Style :these are four types: normal,bold,italic,underline

   ii.   Size: this is the size of characters

   iii.   Colour: there are facilities to colour the characters selecting layer.

   iv.   Type: different types of fonts may be used:

Mono text: COMPUTER AIDED DESIGN

Romans: COMPUTER AIDED DESIGN

Romand: **COMPUTER AIDED DESIGN**

 Dtext: This displays text on the screen as it is entered .AutoCAD can create text with a variety of character patterns, or fonts. These fonts can be stretched, compressed, oblique, mirrored, or aligned in a vertical column by applying a style to the font .text can be rotated, justified, and made any size.

At the command prompt, enter text

Justify/style/<start point>: specify a point or enter an option

TEXT: This creates a single line of text .AutoCAD can create text with a variety of character patterns, or fonts. These fonts can be stretched, compressed, oblique, mirrored, or aligned in a vertical column by applying a style to the font.

At the command prompt, enter **text**

Justify/style/<start point>: specify a point or enter an option

QTEXT: This controls the display and plotting of text and attribute of objects.

At the command prompt, enter **text**

ON/OFF <current>: enter on or off, or press enter

10. **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

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. CUTTING OF OBJECTS

The drawn objects can be cut or trimmed by using following commands

1. **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 3Dpolylines,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

2. **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**

   Select objects: use an object selection method

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

   Enter second point: specify the second break point (2) or enter F

13. 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:

This allows diametrical dimensions of the circle

For dimensioning of objects, the first point and second point has to be specified. The dimension text must be written and then the position of dimension must be specified
At the command prompt, enter **dim**

Dim: Enter a dimensioning mode command

14. AREA:

This allows calculation of the area and perimeter of objects or of defined areas

From the object properties toolbar, choose the inquiry flyout, then

At the command prompt, enter **area**

<First point>/object/add/subtract: specify a point or enter option

15. **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

16. **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

17. **UNION:**

This measures the distance and angle between two points. At the command prompt, enter **union**

Select object: Use an object selection method

18. **DIST:** This measures the distance and the angle between two

points. At the command prompt area enter **dist**

First point: Specify a point (1)

Second point: Specify a point

(2) Distance = calculated

distance

Angle in XY plane = angle from XY plane = angle

Delta X = change **in X**

Delta Y = change in

Y Delta Z = change

in Z.

19.**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

21. **TOLERANCE**

This creates geometric tolerances. Geometric tolerances define the maximum allowable variations of form or profile, orientation, location and runout from the exact geometry in a drawing. They specify the required accuracy for proper function and fit the objects drawn in AutoCAD

22. SKETCH

This creates a series of free hand line segments. From the miscellaneous toolbar, choose

At the command prompt enter **sketch**

Follow the prompting

2.5 3D FUNCTIONS

1.　BOX

This creates a three dimensional solid box.

At 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.

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 fly out 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.

2.6 STARTING THE DRAWING

The figures we do in engineering are fitted into a template. In ACAD draw a template known as *Drawing sheet* in two different formats.

The size of the drawing sheet is
ISO A4 210 X

Pre-lab questions:

1. what are different types of line?
2. What are different AUTO-CAD software?
3. What is drawing environment?

Post lab questions:

1. Wirte any 5 types of line command?
2. List out any 5 2d command?
3. List out any 5 3d command?

Experiment No. 3   PRACTICE EXERCISES ON CAD SOFTWARE

Launching AutoCAD from operating System.

- Click on Start.
- Go to Programs
- Go to Auto Desk
- Go to Auto CAD 2012- English
- Click on Auto CAD 2012- English.

Using your pointing device

One of the key means of controlling commands & inputting information into your drawing file will be by using your mouse.

Using three button mouse:

- Left button: data / accept – used to select command & enter points.
- Middle button: It is used as Pan and scroll zooming
- Right button: Reset / reject / repeat last command – used to end a command or to display

    o a small pull down menu or to repeat last command.
- Using the Key boards.
- Typing Command name is "command line". Can run all the commands in auto CAD.
- Enter button and Space bar can be used to accept a command or repeat last command.
- The Function Keys can be used as toggle buttons to switch ON/ OFF for some of functions. The Important ones are

Function key	Function
F1	Help on
F2	Toggle text windows
F3	Toggle object snap setting
F4	3D o snap on/off
F5	Isoplane Top / Left/ Right
F6	Dynamic UCS off/ on
F7	Toggle Grid points
F8	Ortho on/off
F9	Toggle Snap on/off
F10	Toggle Polar on
F11	Object snap tracking off
F12	Dynamic on/off

User coordinate system:

You can enter pts directly on the command line using 3 different systems. The one you use is depend on which is more applicable for the situation. The first assignment will get you used to this the 3 systems are as follows.

a.   Absolute Coordinate system (x, y)
b. Relative rectangular Coordinate system @ (x, y)

c. Relative Polar Coordinate system : You would use this System if you know that you want to draw a line a certain distance at a ⊥ Lar Angle. We would enter this as at D <

A. In This case D is the distance and A is the angle.

Example: At 10 Angle 90^0 (10 < 90)

d. Creating a new AutoCAD file:

Go to file menu
Select new & Click on drawing
Then select template in that selects ACAD and then click on open.

e. Command in AutoCAD to DRAW, MODIFY and DIMENSION: Before going to

start draw any drawing must set Units, Limits & Grid.

Unit limits:
Command : units (enter)
Type Decimal
Millimeters
Click on ok

Limits:
Command: limits (enter)
Specify lower left corner – (0, 0)
Specify upper right corner (100, 100)

Grid on:

Click on F7

Zoom Command:

Command : Z enter
Options: (All / center/ dynamic/ extents. Previous/ scale/ window/ object) (real Time)

Line command:

Command: L (enter)

Using absolute ordinate system

L (enter)
(0,0) enter
(100, 0) enter
(100, 00) enter
(10, 100) enter

Using relative rectangular coordinate system

L (enter)
(0,0) enter
@100,0
@0,100
@-100,0
C enter

Using relative polar Coordinate System:

L enter
Click on screen
@ 100 < 0 enter
@ 100< 90 enter
@ 100 < 180 enter

Construction Line:

Command: XL enter
XLINE Specify a point or [Hor/ ver/ Ang/ Bisect/ offset] : h

Polyline:

Command: PL (enter)
Specify Start point:
Specify next point (or) [Arc/ Half width/ length/ undo/ width] : click on screen
        For arc in the polyline enter option 'A'

Polygon:

Command: POL (enter)
Polygon enter no. of sides : 5 (enter)
Specify center of polygon or edge : Click on Screen
Enter an option [Inscribed circle/ circumscribed about circle] {I}:C

Circle:

Command: C (enter)
Circle Specify center pt for circle or [3p / 2p/ Ttr]: click on screen
Specify radius of circle : 50 (enter)
    second click

Rectangle:

Rec (enter)

Specify 1st Corner pt: click on Screen first pt

Specify 2nd

Corner pt: click on Screen second pt

Option for specifying other corner points
Using dimension option

Specify other corner pt: [Area/ dimension]: D (enter)
Specify length: 100 (enter)

Specify width : 50 (enter)
 Click on screen

Using Area option

A (enter)
Enter area of rectangle : 100
L (enter)
10 (enter)

Arc:

Command : Arc **(**enter)
Specify 1st pt: click on screen
Specify 2nd pt: click on screen
Specify end pt: click on screen

Revision cloud:

Command : Revcloud(enter**)**
Specify start pt or [arc/length/object/ style]
s (enter)

Spline:

Command : Spl (enter)
Click on screen

Ellipse:

Command : El(enter)
Click on screen (3pts)

Ellipse Arc:

Command : El (enter)
A (enter)
Click on screen (3pts)

Hatch:

Command :h (enter)
Scale – 50 (properties)
Click on pick pts.
Click on rectangle

Command :H (enter)
Scale – 0.5
Pick pts
Click on rectangle

Point & Point style:

6. Points are very simple objects and the process of creating them is very simple.

7. Points are rarely used as drawing components although there is no reason why they could not be.

For Example: Points are automatically created when you measure and divide commands and to set act distance along lines

Point

$^\vee$ Command: po (enter)
Click on Screen

Multi Line commands

command: mline (enter)

Click on Screen

Multiline Style Commands:

Go to format menu
Select multiline style and click
Keyboard: ML style.

Object snap.

command: OS (enter)
Select the required node otherwise (select all)

Text command

Command: t (enter)

Specify two corners
Enter the text
Click on ok

Copy

Command: co (or) cp (enter)
Select the Object (enter)
Select the base point and place the point

Mirror

Command: mi (enter)
Select the object (enter)
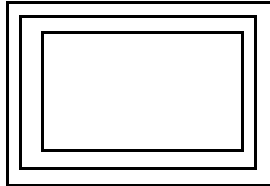Specify the first pt of mirror line
Specify the second pt of mirror line
Erase the source object (yes/No) <N>

Offset

Command: O (enter)
Enter the offset distance:
example: 10 (enter)

Select the object and direct the offset

Rectangular Array

Command: ar (enter)
 Select Tick The rectangular array
 Enter the no of rows
 Enter the no of columns
 Click on the selected objects button

Polar array:

Command: ar (enter)
Tick polar array
Select the objects
Enter the total no of items
Specify the angle
Specify the center point

Move

Command: m (enter)
Select the object
Specify the base point
Click wherever required

Rotate

Command: ro (enter)
Select the object (enter)
Specify the base point
Enter the angle

Example: 45^0 enter

Scale

    Command: sc (enter)

      Select the object (enter)

      Specify the base points

      Enter   the   scale factor Example: 2 (enter)

Stretch

    s (enter)

    Select the object (enter)

    Specify Base point and drag the mouse

  towards the required direction $^\vee$ 25

Trim

    Command: tr (enter)

    Select cutting edges (enter)

    Select the objects to trim

Extend command

    Command: Ex (enter)

    Select boundary edges (enter)

    Then select the object to extend

Break command:

    Command: br (enter)

    Select 1st point and 2nd point

Chamfer command:

(Draw a rectangle of 100 x 50)

    Command: Cha (enter)

    d (enter)

    Enter the distance 15 (enter)

    Enter the distance 10 (enter)

    Select two adjacent line on screen

Fillet

    Command: f(enter)

    r (enter)

    example: 10 (enter)

Explode:

    x (enter)

Select the object (enter)

Pre lab questions

1. What is user co-ordinate system?
2. What is polar co-ordinate system
3. What is absolute co-ordinate system?

Post lab questions
1. What is offset?
2. What is rectangular array?
3. What is fillet?

Experiment No. 4   DRAWING OF PLANS OF BUILDINGS USING SOFTWARE

a)   SINGLE STOREYED

b)   MULTI STOREYED

a) SINGLE STOREYED

4.1 <u>Aim:</u>

To draw the plan of single storeyed building using the various commands in AutoCAD

4.2 <u>Software Used:</u>

AutoCADD 2016

4.3 <u>Command Used And Their Description:-</u>

Zoom – It is used to zoom the object created.

Units – Used to set the current format for units of measure.

Line –Line commands allows creating a line where the end points allow creating a line where the end points are dimensional co-ordinates.

Line type – using this command different type of lines can be used to draw object.

Offset – create a news object at a specified distance from an existing object or through a specified

point.
Fillet - This command is basically used for rounding off edges

Trim – trims off an object using cutting edges defined by other objects.2

Break – removes only a part of an object.

Arc – Used to create an arc segment. Methods are:

1. 3 Points,

2. Start, Center, End

3. Start, Center, Angle

4. Center, Start, End

5. Center, Start, Angle

6. Start, Center, Length

7. Center, Start, Length

8. Start, End, Angle

9. Start, End, Radius

10. Start, End, Diameter

11. Continue

Copy- Moves the selected objects from a given square to destination, learning a copy at the originally selected location.

Rotate- rotate objects around a specified point

Move- moves object to the destination place from the source place

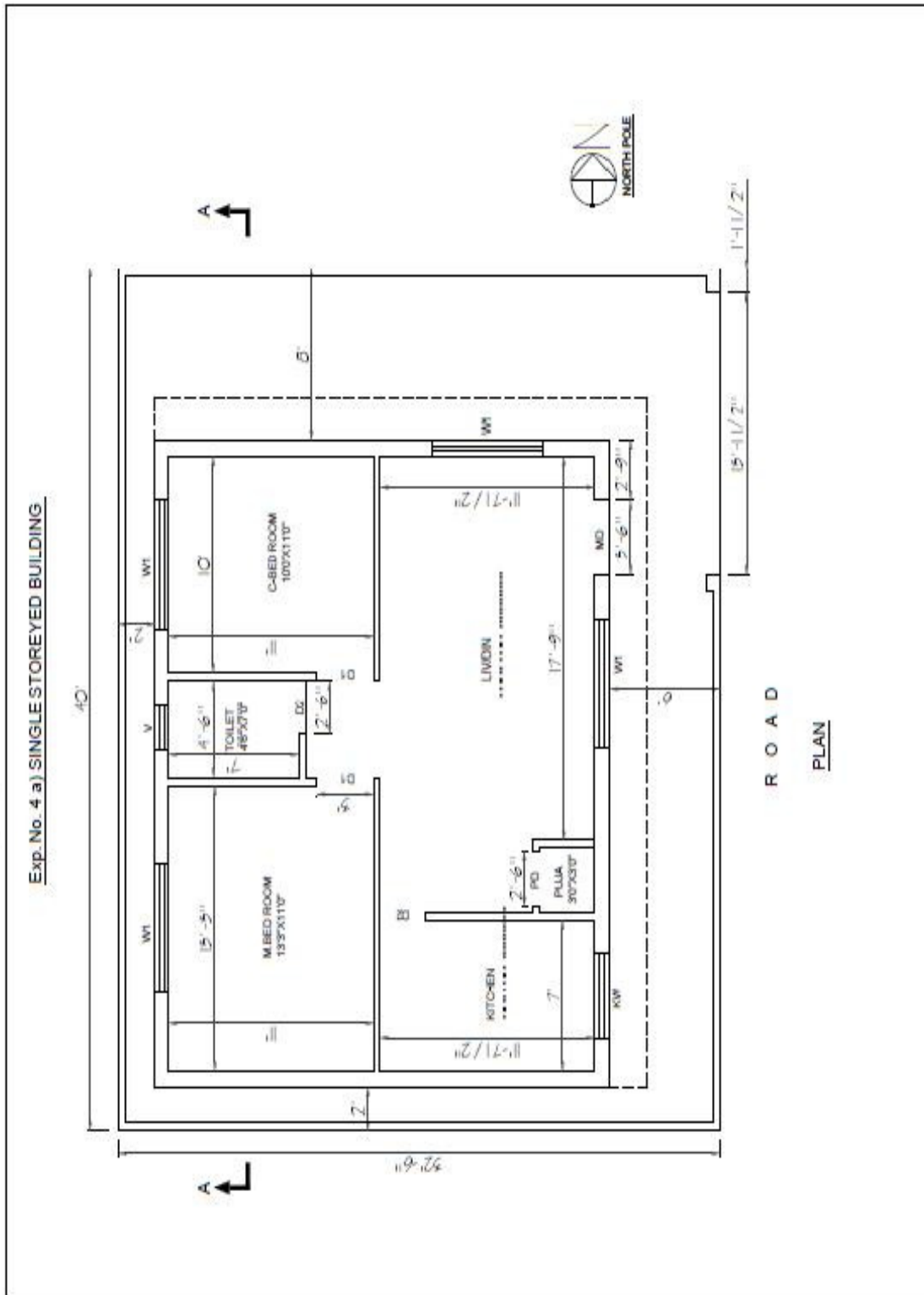BHatch – makes shaded patterns as matter of few picks and clicks away

Extend – elongates an object to a boundary defined by other objects

Erase – used to erase the unwanted objects

Text – creates text object with specified height and orientation

osnap – AutoCAD displays the object snaps tab in drafting setting dialog box. If we enter

osnap at command prompt it presents options on the command line.

4.4 Procedure:

Exp. No. 4 a) SINGLE STOREYED BUILDING

PLAN

R O A D

NORTH POLE

M BED ROOM
13'3"X11'0"

C-BED ROOM
10'0"X11'0"

TOILET
4'6"X7'0"

LIVING

KITCHEN

PUJA
3'0"X3'0"

1. Set units type-Architectural and precision-0'-0½"
2. Create layers, text style, Dimension style
3. Draw a rectangle required template size using rectangular command, then Z 'enter' E 'enter' to zoom extend
4. Start drawing plan with line command one horizontal line and vertical line-while drawing line keep **Ortho on**
5. With offset command wall thickness and room dimensions are drawn
6. Use Trim command at every junction and excess lines
7. Provide doors and windows openings according to the sizes- use offset and trim command
8. Text and Dimensioning – (common text height 9"- Main text height 14")-Dimensioning commands preferable Linear, Aligned and leader.
9. Draw section mark –use polyline command
10. Schedule of Door and windows- use text command
11. Using save command, the file having plan is saved by giving the Corresponding path name.

4.5 Results:

The functional requirements of the single storeyed building is planned and the plan drawn in AutoCAD 2013

4.6 Pre lab questions

1. What is osnap?
2. What is plan?
3. List various commands used in this building plan?

4.7 Post lab questions

1. What is polyline?
2. What is common text height?
3. Thickness of wall?

4.8 Assignment question

1. Draw single storeyed building plan in Auto Cad 2016

Experiment No. 4. Drawing of plans of Buildings using software

b) MULTI STOREYED

4.1 Aim:

To draw the plan of single storeyed building using the various commands in AutoCAD

4.2 Software Used:

AutoCADD 2016

4.3 Command Used And Their Description:-

Zoom – It is used to zoom the object created.

Units – Used to set the current format for units of measure.

Line –Line commands allows creating a line where the end points allow creating a line where the end points are dimensional co-ordinates.

Line type – using this command different type of lines can be used to draw object.

Offset – create a news object at a specified distance from an existing object or through a specified point.

Fillet - This command is basically used for rounding off edges

Trim – trims off an object using cutting edges defined by other objects.2

Break – removes only a part of an object.

Arc – Used to create an arc segment. Methods are:

1. 3 Points,

2. Start, Center, End

3. Start, Center, Angle

4. Center, Start, End

5. Center, Start, Angle

6. Start, Center, Length

7. Center, Start, Length

8. Start, End, Angle

9. Start, End, Radius

10.Start, End, Diameter

11.Continue

Copy- Moves the selected objects from a given square to destination, learning a copy at the originally selected location.

Rotate- rotate objects around a specified point

Move- moves object to the destination place from the source place

BHatch – makes shaded patterns as matter of few picks and clicks away
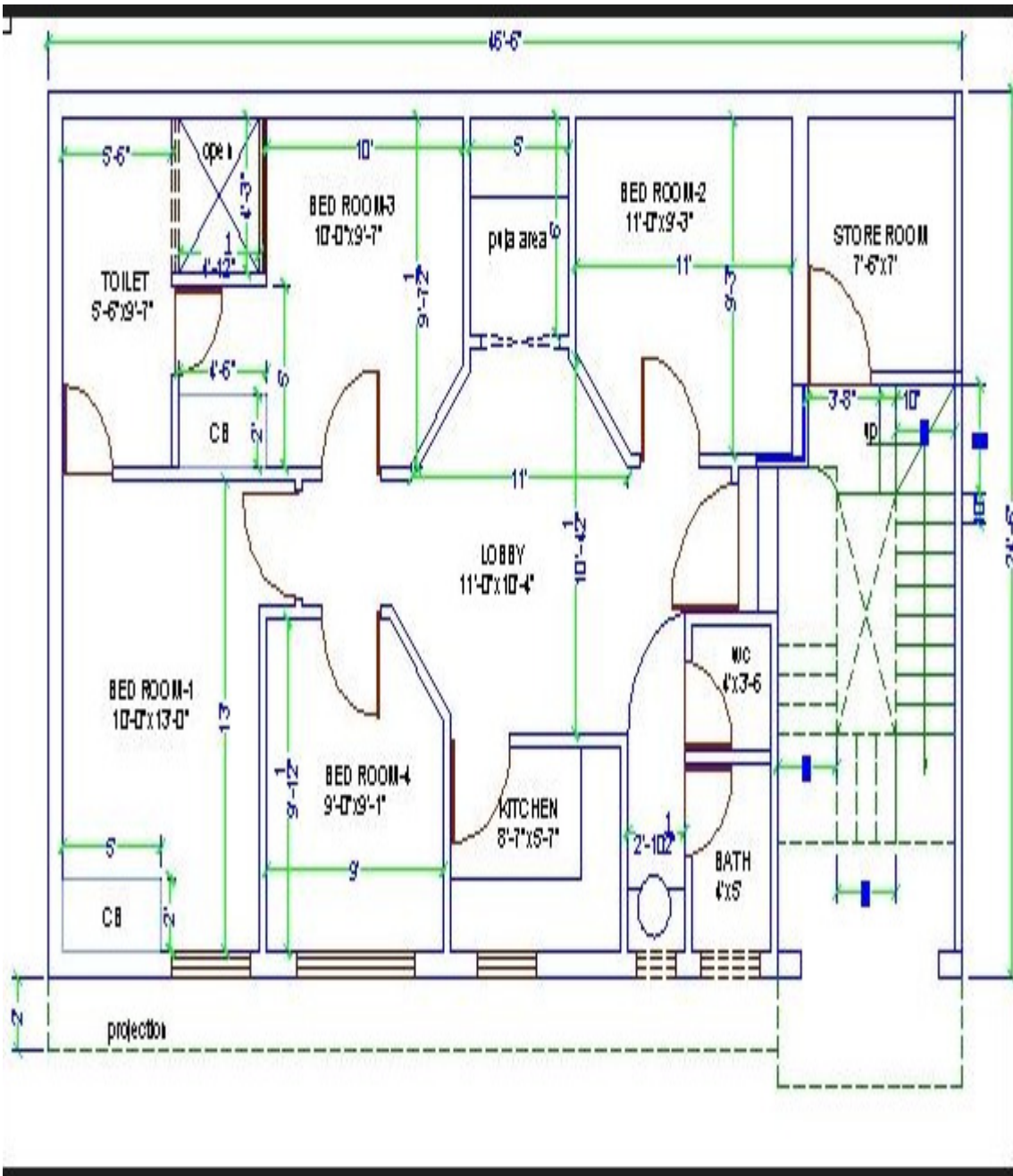
Extend – elongates an object to a boundary defined by other objects

Erase – used to erase the unwanted objects

Text – creates text object with specified height and orientation

osnap – AutoCAD displays the object snaps tab in drafting setting dialog box. If we enter

osnap at command prompt it presents options on the command line.

4.4 Procedure:

4.5 <u>Results</u>:

The functional requirements of the multi storeyed building is planned and the plan drawn in AutoCAD 2016

4.6 <u>Pre lab questions</u>

1. What is meant by multi storyed building?
2. Define trim?
3. How to set page dimensions?

4.7 <u>Post lab questions</u>

1. Define break command
2. Different types of line commands?

4.8 <u>Assignment questions</u>

1. Draw multi storeyed building plan in Auto Cad 2016

Experiment No. 5 DEVELOPING SECTIONS AND ELEVATIONS FOR
a) SINGLE STOREYED
b) MULTI STOREYED

a) Single storeyed Building

5.1 <u>Aim:</u>

To draw the plan of single storeyed building using the various commands in AutoCAD

5.2 <u>Software Used:</u>

AutoCADD 2016

5.3 <u>Command Used And Their Description:-</u>
Zoom – It is used to zoom the object created.

Units – Used to set the current format for units of measure.

Line –Line commands allows creating a line where the end points allow creating
a line where the end points are dimensional co-ordinates.

Line type – using this command different type of lines can be used to draw object.

Offset – create a news object at a specified distance from an existing object or through a specified

point.
Fillet - This command is basically used for rounding off edges

Trim – trims off an object using cutting edges defined by other objects.2

Break – removes only a part of an object.

Arc – Used to create an arc segment. Methods are:

1. 3 Points,

2. Start, Center, End

3. Start, Center, Angle

4. Center, Start, End

5. Center, Start, Angle

6. Start, Center, Length

7. Center, Start, Length

8. Start, End, Angle

9. Start, End, Radius

10. Start, End, Diameter

11. Continue

Copy- Moves the selected objects from a given square to destination, learning a copy at the originally selected location.

Rotate- rotate objects around a specified point

Move- moves object to the destination place from the source place

BHatch – makes shaded patterns as matter of few picks and clicks away

Extend – elongates an object to a boundary defined by other objects

Erase – used to erase the unwanted objects

Text – creates text object with specified height and orientation

osnap – AutoCAD displays the object snaps tab in drafting setting dialog box. If we enter

osnap at command prompt it presents options on the command line.

5.4 <u>Procedure:</u>

1. The elevation is drawn by extending the outer line of the plan using extend command and unwanted lines are erased using erase command.
2. The doors, windows, sunshade and parapet are drawn in same distance in elevation as drawn in same distance in the plan.
3. The section is drawn on the adjacent side of the elevation by extending the lines.
4. The various represtations of brickwork, sand filling and concrete are completed in the section using Bhatch command.
5. Using save command, the file having plan, elevation and section is saved by giving the Corresponding path name.
6. Using save command, the file having plan, elevation and section is saved by giving the Corresponding path name.

5.5 <u>Results</u>:

The functional requirements of the multi storeyed building is planned, the section and elevation are drawn in AutoCAD 2016

5.6 <u>Pre lab questions</u>

1. How to use Hatch?
2. Define trim?
3. How to set page dimensions?

5.7 <u>Post lab questions</u>

1. Define break command
2. Define section?
3. Define elevation?

5.8 <u>Assignment questions</u>
1. Draw the plan of single storeyed building

Experiment No. 5 DEVELOPING SECTIONS AND ELEVATIONS FOR

b)Multi storeyed Building

5.1 <u>Aim:</u>

To draw the plan of multi storeyed building using the various commands in AutoCAD

5.2 <u>Software Used:</u>

AutoCADD 2016

5.3 <u>Command Used And Their Description:-</u>

Zoom – It is used to zoom the object created.

Units – Used to set the current format for units of measure.

Line –Line commands allows creating a line where the end points allow creating a line where the end points are dimensional co-ordinates.

Line type – using this command different type of lines can be used to draw object.

Offset – create a news object at a specified distance from an existing object or through a specified point.

Fillet - This command is basically used for rounding off edges

Trim – trims off an object using cutting edges defined by other objects.2

Break – removes only a part of an object.

Arc – Used to create an arc segment.

Methods are:

1. 3 Points,

2. Start, Center, End

3. Start, Center, Angle

4. Center, Start, End

5. Center, Start, Angle

6. Start, Center, Length

7. Center, Start, Length

8. Start, End, Angle

9. Start, End, Radius

10. Start, End, Diameter

11. Continue

Copy- Moves the selected objects from a given square to destination, learning a copy at the originally selected location.

Rotate- rotate objects around a specified point

Move- moves object to the destination place from the source place

BHatch – makes shaded patterns as matter of few picks and clicks away

Extend – elongates an object to a boundary defined by other objects

Erase – used to erase the unwanted objects

Text – creates text object with specified height and orientation

osnap – AutoCAD displays the object snaps tab in drafting setting dialog box. If we enter osnap at command prompt it presents options on the command line.

5.4 Procedure:

1. The elevation is drawn by extending the outer line of the plan using extend command and unwanted lines are erased using erase command.
2. The doors, windows, sunshade and parapet are drawn in same distance in elevation as drawn in same distance in the plan.
3. The section is drawn on the adjacent side of the elevation by extending the lines.
4. The various repetitions of brickwork, sand filling and concrete are completed in the section using Bhatch command.
5. Using save command, the file having plan, elevation and section is saved by giving the Corresponding path name.
6. Using save command, the file having plan, elevation and section is saved by giving the Corresponding path name.

5.5 <u>Results</u>:

The functional requirements of the multi storeyed building is planned, the section and elevation are drawn in AutoCAD 2013

5.6 <u>Pre lab questions</u>

1. How to use Hatch?
2. Define trim?
3. Define Multi storey building?

5.7 <u>Post lab questions</u>

1. How to use text command?
2. Define section?
3. Define elevation?

5.8 <u>Assignment questions</u>

1. Draw the plan of multi storeyed building?

Experiment No. 6   DETAILING OF BUILDING COMPONENTS LIKE DOORS, WINDOWS, ROOF TRUSSES ETC. USING CAD SOFTWARE

a)DOORS

6.1 Aim:

To draw the plan, section and elevation of doors using the various commands in AutoCAD

6.2 Software Used:

AutoCADD 2013

6.3 Command Used And Their Description:-

Zoom – It is used to zoom the object created.

Units – Used to set the current format for units of measure.

Line –Line commands allows creating a line where the end points allow creating a line where the end points are dimensional co-ordinates.

Line type – using this command different type of lines can be used to draw object.

Offset – create a news object at a specified distance from an existing object or through a specified point.

Fillet - This command is basically used for rounding off edges

Trim – trims off an object using cutting edges defined by other objects.2

Break – removes only a part of an object.

Arc – Used to create an arc segment. Methods are:

1. 3 Points,

2. Start, Center, End

3. Start, Center, Angle

4. Center, Start, End

5. Center, Start, Angle

6. Start, Center, Length

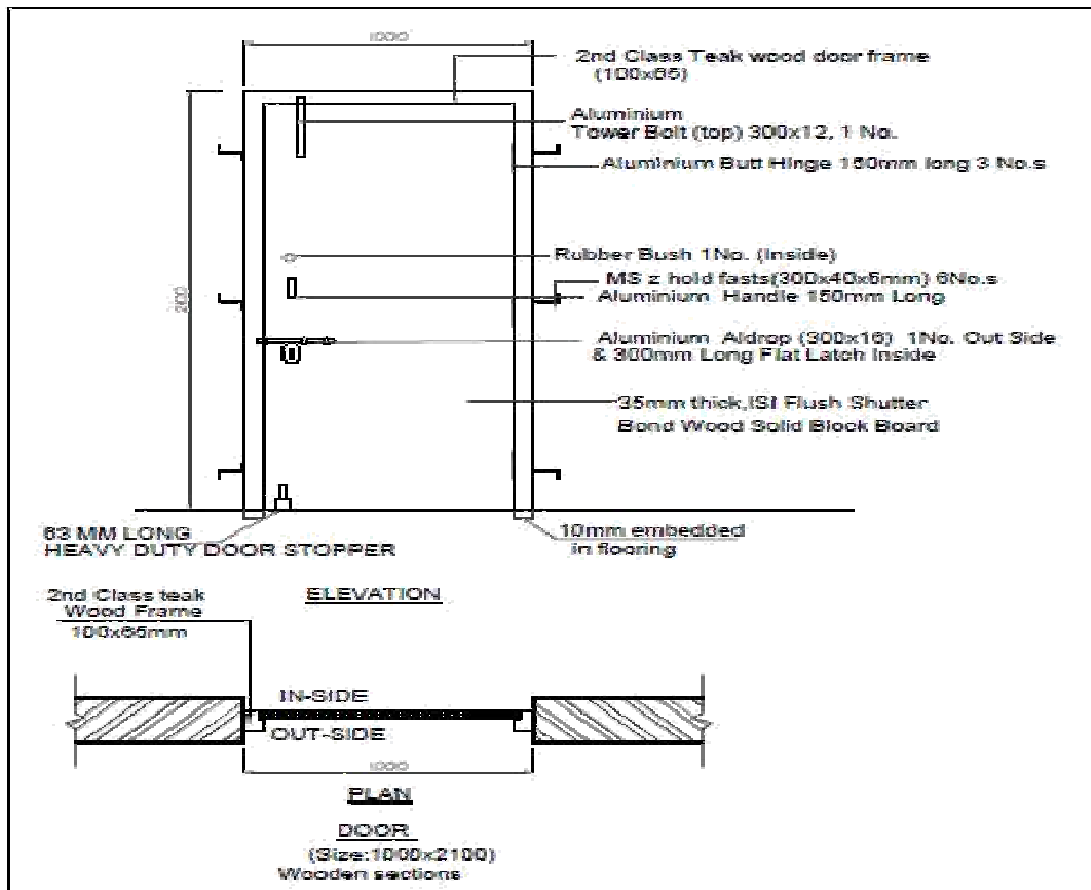7. Center, Start, Length

8. Start, End, Angle

9. Start, End, Radius

10. Start, End, Diameter

11. Continue

ᵛ Copy- Moves the selected objects from a given square to destination, learning a copy at the originally selected location.

ᵛ Rotate- rotate objects around a specified point

ᵛ Move- moves object to the destination place from the source place

ᵛ BHatch – makes shaded patterns as matter of few picks and clicks away

ᵛ Extend – elongates an object to a boundary defined by other objects

ᵛ Erase – used to erase the unwanted objects

ᵛ Text – creates text object with specified height and orientation

ᵛ osnap – AutoCAD displays the object snaps tab in drafting setting dialog box. If we enter osnap at command prompt it presents options on the command line.

6.4 **Procedure:**

DOOR
(Size:1000x2100)
Wooden sections

1. The plan is drawn as per the door size and shown in above fig.
2. The elevation is drawn by extending the outer line of the plan using extend command and unwanted lines are erased using erase command.
3. The section is drawn on the adjacent side of the elevation by extending the lines.
4. The various represtations of frames, Aldrop, Door stopper, Handles etc.., are completed in the section using line and Bhatch commands.
5. Using save command, the file having plan, elevation and section is saved by giving the Corresponding path name.
6. Using save command, the file having plan, elevation and section is saved by giving the Corresponding path name.

6.5 <u>Results</u>:

The planned, the section and elevation of the door is drawn in AutoCAD 2013

6.6 <u>Pre lab questions</u>

1. How to use Hatch?
2. Define trim?
3. define Multi storey building?

6.7 <u>Post lab questions</u>

1. How to use text command?
2. Define section?
3. Define elevation?

6.8 <u>Assignment questions</u>

1. Draw the section and elevation of the door?

Experiment No. 6   DETAILING OF BUILDING COMPONENTS LIKE DOORS, WINDOWS, ROOF TRUSSES ETC. USING CAD SOFTWARE

b) WINDOW

6.1 Aim:

To draw the plan, section and elevation of doors using the various commands in AutoCAD

6.2 Software Used:

AutoCADD 2013

6.3 Command Used And Their Description:-

ᵛ   Zoom – It is used to zoom the object created.

ᵛ   Units – Used to set the current format for units of measure.

ᵛ   Line –Line commands allows creating a line where the end points allow creating a line where the end points are dimensional co-ordinates.

ᵛ   Line type – using this command different type of lines can be used to draw object.

ᵛ   Offset – create a news object at a specified distance from an existing object or through a specified

point.

ᵛ   Fillet - This command is basically used for rounding off edges

ᵛ   Trim – trims off an object using cutting edges defined by other objects.2

ᵛ   Break – removes only a part of an object.

ᵛ   Arc – Used to create an arc segment. Methods are:

1. 3 Points,

2. Start, Center, End

3. Start, Center, Angle

4. Center, Start, End

5. Center, Start, Angle

6. Start, Center, Length
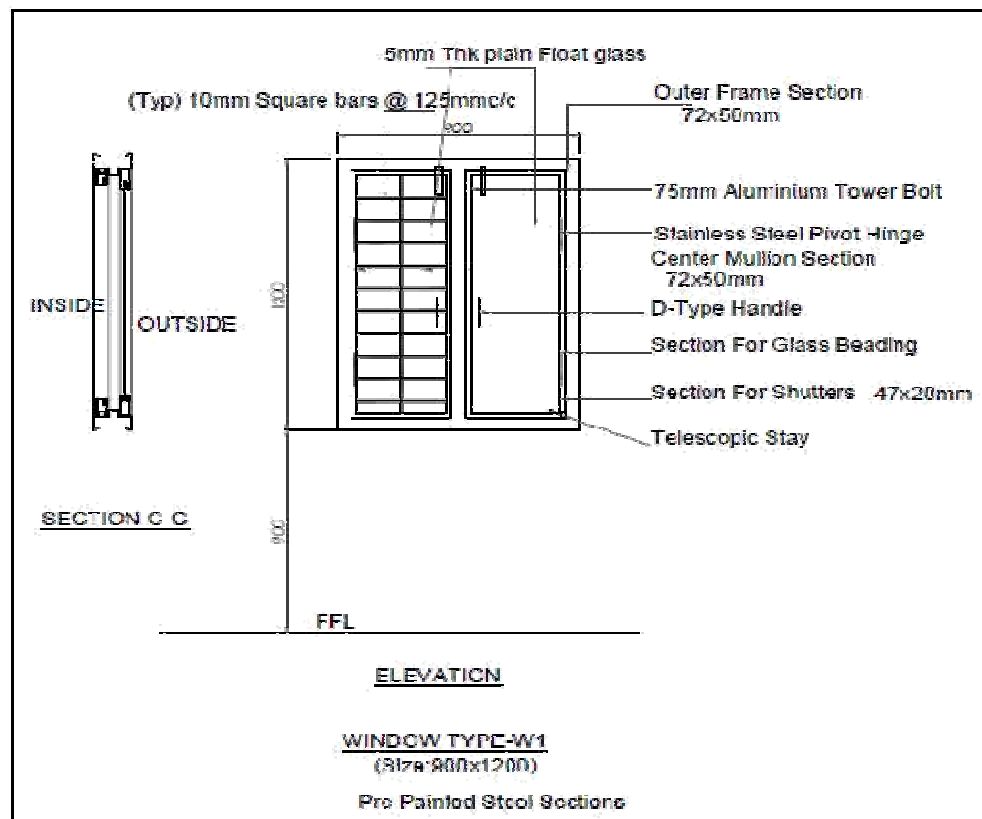
7. Center, Start, Length

8. Start, End, Angle

9. Start, End, Radius

10. Start, End, Diameter

11. Continue

ᵛ Copy- Moves the selected objects from a given square to destination, learning a copy at the originally selected location.

ᵛ Rotate- rotate objects around a specified point

ᵛ Move- moves object to the destination place from the source place

ᵛ BHatch – makes shaded patterns as matter of few picks and clicks away

ᵛ Extend – elongates an object to a boundary defined by other objects

ᵛ Erase – used to erase the unwanted objects

ᵛ Text – creates text object with specified height and orientation

ᵛ osnap – AutoCAD displays the object snaps tab in drafting setting dialog box. If we enter

ᵛ osnap at command prompt it presents options on the command line

6.4 **Procedure:**

WINDOW TYPE-W1
(Size 900x1200)
Pre Painted Steel Sections

1. The plan is drawn as per the window size and shown in above fig.
2. The elevation is drawn by extending the outer line of the plan using extend command and unwanted lines are erased using erase command.
3. The section is drawn on the adjacent side of the elevation by extending the lines.
4. The various representations of frames, tower bolts, railings, Handles etc.., are completed in the section using line and B hatch commands.
5. Using save command, the file having plan, elevation and section is saved by giving the Corresponding path name.
6. Using save command, the file having plan, elevation and section is saved by giving the Corresponding path name.

6.5 <u>Results</u>:

The planned, the section and elevation of the window is drawn in AutoCAD 2013

6.6 <u>Pre lab questions</u>

1. How to use Hatch?
2. Define trim?
3. Define Multi storey building?

6.7 <u>Post lab questions</u>

1. How to use text command?
2. Define section?
3. Define elevation?

6.8 <u>Assignment questions</u>

1. Draw the section and elevation of the window?

Experiment No. 7 EXERCISES ON DEVELOPMENT OF WORKING OF BUILDINGS.

7.1 <u>Aim:</u>

To draw the working drawing building using the various commands in AutoCAD

7.2 <u>Software Used:</u>

AutoCADD 2013

7.3 <u>Command Used And Their Description:-</u>

Zoom – It is used to zoom the object created.

Units – Used to set the current format for units of measure.

Line –Line commands allows creating a line where the end points allow creating a line where the end points are dimensional co-ordinates.

Line type – using this command different type of lines can be used to draw object.

Offset – create a news object at a specified distance from an existing object or through a specified point.

Fillet - This command is basically used for rounding off edges

Trim – trims off an object using cutting edges defined by other objects.2

Break – removes only a part of an object.

Arc – Used to create an arc segment. Methods are:

1. 3 Points,

2. Start, Center, End

3. Start, Center, Angle

4. Center, Start, End

5. Center, Start, Angle

6. Start, Center, Length

7. Center, Start, Length

8. Start, End, Angle

9. Start, End, Radius

10. Start, End, Diameter

11. Continue

Copy- Moves the selected objects from a given square to destination, learning a copy at the originally selected location.

Rotate- rotate objects around a specified point

Move- moves object to the destination place from the source place

BHatch – makes shaded patterns as matter of few picks and clicks away
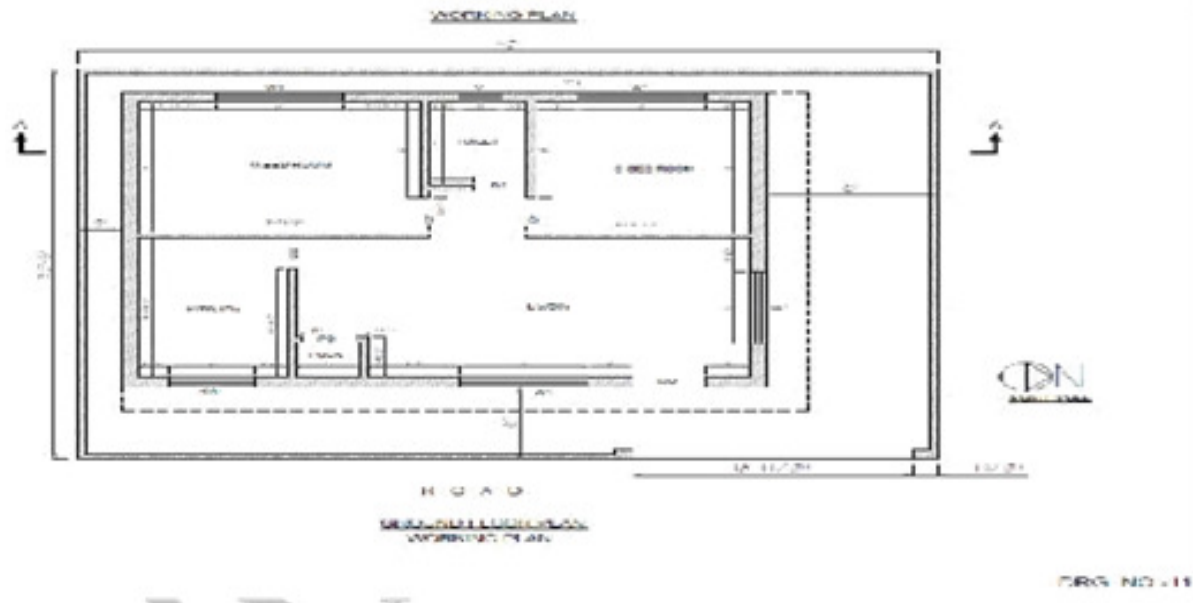
Extend – elongates an object to a boundary defined by other objects

Erase – used to erase the unwanted objects

Text – creates text object with specified height and orientation

osnap – AutoCAD displays the object snaps tab in drafting setting dialog box. If we enter osnap at command prompt it presents options on the command line

7.4 <u>Procedure:</u>

1. Start drawing working plan to Architectural plan with of hatch, dim etc… commands
2. Remove all room sizes and do dimensioning each and every detail.
3. Using save command, the file having plan is saved by giving the Corresponding path name.

7.5 <u>Results</u>:

The working drawing for the given plan is drawn in AutoCAD 2013

7.6 <u>Pre lab questions</u>

1. How to use Hatch?
2. Define trim?
3. Define Multi storey building?

7.7 <u>Post lab questions</u>

1. How to use text command?
2. Define section?
3. Define elevation?

7.8 <u>Assignment questions</u>

1. Draw a working plan of building using Auto CAD?

S.NO	LIST OF EXPERIMENTS
1	Introduction to computer aided drafting
2	Software for CAD- Introduction to different software's
3	Practice exercises on CAD software
4	Drawing of plans of buildings using software c) Single storeyed d) Multi storeyed
5	Developing sections and elevations for c) Single storeyed d) Multi storeyed
6	Detailing of building components like doors, windows, roof trusses etc. using CAD software's
7	Exercises on development of working of buildings.