LAB MANUAL
COMPUTER AIDED DRAFTING

MECHANICAL ENGINEERING DEPARTMENT

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LIST OF EXPERIMENTS

1. Learn the basic initial setting and viewing of the drafting software's Interface.

2. Learn the basic options of drawing aids like grid, snap, ortho etc. and other aids for distance and mass properties calculations

3. Learn and draw the basic entities in 2D

4. Learn and use the various modify commands of the drafting software

5. Learn and use the layers and blocks in drafting software

6. Use hatching and dimensioning to detail out a component drawings

7. Understand different coordinate system and do a exercise on drafting software

8. Draw the different types of 3D modeling entities using viewing commands to view them

9. Draw the different Surface model with different editing commands

10. Learn and use shading and rendering techniques for better visual appearance

11. Use and learn import/export techniques and customization of drafting software.

EXPERIMENT -1

AIM: - LEARN THE BASIC INITIAL SETTING AND VIEWING OF THE DRAFTING SOFTWARE'S INTERFACE

OBJECTIVE: - To learn initial setting and viewing of the drafting software interfaces.

THEORY: To find out Auto-CAD package and set its required setting for drawing in the auto-CAD window.

- How to Start your PC.
- Opening Auto-CAD Window.
- Description of Auto-CAD window.
- Setting GUI (Graphic User Interface) for drawing.

How to Start your PC.

Switch "ON" main switch. Switch "ON" UPS. Switch "ON" CPU and Monitor.

Opening Auto CAD Window.

First go to Start, then go to programme file and then go to Auto-CAD/Auto Desk. Select respective auto-CAD window, click ok.

Open an existing drawing:

To opens an existing drawing select from a list from the fore most recently open drawing. Also display brows button that you choose look for another file.

Start a drawing from scratch

Open a new drawing based on the measurement system you choose-English (inches) or Metric (millimeters) system.

Use a Template:

Open a new drawing based on a template you select from a list. The list display on a template files that exits in the drawing file locations specified in the option dialog box Template files stores all the setting for a drawing.

Setting Drawing Units: Drawing units are sets as per drawing units given in the drawing to be done. It may be in MKS or SI system such as millimeters or in inches in case of distance units and similarly for angular measurements it may be Degree, Radian or Grade e.t.c.

To set drafting Units: In the Drawing units dialog box set the units value for drawing. Type "units" at command window**.** Press ent. Select units in mm or inches. Select Precision, 0.00 or 0.000 or 0 itself. Ent. Select angular direction, clockwise, leave box blank,for anti clockwise, select/ click ok.

Command: Units

Setting the Limits of Drawing:

Limits of Drawing specify the area in which the drawing is to be drawn.

Command: Limits. Enter.

Specify the lower left corner of the area: say (00,00)

Specify the upper right corner of the area: (297, 420) o(420, 297) for land scape size. Press ent. Write "zoom" . Ent. Type "All" ent.

After specifying the limits we set Gid.

- Setting of Grid SpacingType grid, press ent. Clk on grid dialog box . set grid spacing 5/ 10 / 15 as desired. Press ent.

Setting of Snap Spacing. Click on snap . ent value as 5/10/ 15 as desired. Ent.

This will zoom all the limits of drawing. After this we can draw the required drawing

DRAFTING SOFTWARE INTERFACES

Menus, Toolbars and Tool Palettes..

You can use several menus, shortcut menus, palettes, and toolbars for access to frequently used commands, settings, and modes.

The Command Window.

You can display commands, system variables, options, messages, and prompts in a lockable and resizable window called the command window.

Design Center.

With Design Center, you can manage block references and other content such as layer, definitions, layouts, and text styles etc.

Customize the Drawing Environment.

Many elements of the working environment can be customized to fit your needs.

Pointing Devices:

You can use as your pointing device a mouse, a digitizing puck, and it may have more than two buttons. You can use several menus, shortcut menus, tool palettes, and toolbars for access to frequently used commands, settings, and modes.

Tool Palettes

Tool palettes are tabbed areas within the Tool Palettes window that provide an efficient method for organizing, sharing, and placing blocks and hatches. Tool palettes can also contain custom tools provided by third-party developers.

- **Insert Blocks and Hatches Using Tool Palettes**
  Tool palettes are tabbed areas within the Tool Palettes window. Drag blocks and hatches from a tool palette to place these objects quickly on a drawing.
- **Change Tool Palette Settings**
  The options and settings for tool palettes are accessible from shortcut menus in different areas on the Tool Palettes window.
- **Control Tool Properties** You can change the insertion properties or pattern/

Properties of any tool on a tool palette.

- **Customize Tool Palettes** You can add tools to a tool palette by veral methods.

Save and Share Tool Palettes: Save and share a tool palette by exporting it or importing it as a tool palette file.

Toolbars:

Use buttons on toolbars to start commands, display fly out toolbars, and display tool tips. Toolbars contain buttons that start commands. When you move the pointing device over a toolbar button, the tool tip displays the name of the button. Buttons with a small black triangle in the lower-right corner have fly outs that contain related commands. With the cursor over the icon, hold down the pick button until the fly out appears. The Standard toolbar at the top of the drawing area is displayed by default. This toolbars are Draw tools, Modify tools etc.

Shortcut Menus:

Display a shortcut menu for quick access to commands that are relevant to your current activity.

Menus are available from the menu bar at the top of the AutoCAD drawing area. You can choose menu options in the following ways:

- Click the menu name to display a list of options. Click the option to choose it, or press DOWN ARROW to move down the list and then press ENTER.

Press ALT and then enter the underlined letter in the menu name. For example, to open a new drawing, press ALT and press F to open the File menu. Then press ENTER to choose the highlighted option new.

EXPERIMENT-2

AIM: - LEARN THE BASIC OPTIONS OF DRAWING AIDS LIKE GRID, SNAP, ORTHO ETC. AND OTHER AIDS FOR DISTANCE AND MASS PROPERTIES CALCULATIONS

THEORY AND PROCEDURE: -

Drawing aids: These are the helping tools used to assist drafter to draw complicated drawing but not a draw tools. These are very helpful to make a drawing easy. These are SNAP, GRID, ORTHO, OSNAP, OTRACK etc.

Snap: It is useful for specifying precise points with the arrow keys or the pointing device.

Grid: It is a rectangular pattern of dots that extends over the area you specify as the drawing limits. Using the grid is similar to placing a sheet of grid paper under a drawing.

Ortho: - This aids to draw horizontal and vertical lines when it is on.

Polar: - It is used to draw lines at an angle to the reference line either clockwise or anticlockwise depending upon the requirement of drafter.

OSNAP (Object Snap) and OTRACK (Object Snap tracking): - This is used to make proper connection / attachment of lines with the other diagram or lines when it reach to the nearest point. It facilitates to identify the required co-ordinate points such as mid point, center, end point, tangents etc which are otherwise very difficult to track and make proper connection of lines or to complete a drawing. It has following features. End point, Mid point, Center, Node, Quadrant, Intersection, Extension, Perpendicular, Tangent, Nearest point, apparent intersection and parallel.

LWT: - Used to give required thickness of lines as defined in Engineering drawing. e.g. Border line, Title block line , Center line etc.

Model: -This is used to set the drawing sheet as required by the designer to follow standard procedure of drawing.

EXPERIMENT- 3

AIM: - LEARN AND DRAW THE BASIC ENTITIES IN 2D

OBJECTIVE: - To learn the basics of 2D entities.

THEORY: - In the 2D entities, we can draw different drawings by use of different draw commands.i.e.line arc, circle, rectangle, polygon, ellipse etc. In this exercise we will draw different types of drawing by using these commands with the help of different methods.

PROCEDURE:

The some exercises are drawn with the help of the following commands:

P- Line Command or PL Command: -

A polyline is a connected sequence of line and arc segments. It has feature to change the line width.

Command: P line or PL

Specify the start Pt: select the starting Pt by using mouse.

Select different Pts and press enter. Current line width is 0.000 or change if required. You can change the line to an arc and then to a line of same or different thickness. (See Fig 01)

Command: – Rectangle

A rectangle is a polyline based on two opposite Corner Pts called diagonal points. (See Fig 2)

Draw a rectangle defined by diagonal Pt (10, 10) and (30, 20)

Command: - Recta. Ent.

First corner: P_1 (10, 10). Ent.

Second corner: P_2 (30, 20). Ent. (See Fig 03)

Rectangles can be drawn by lines commands also. Line, Ent Pt.

P_1(10,10) Ent
P_2 (40,10) Ent
P_3 (40, 40). Ent
P_4 (10, 40). Ent.
C . Ent. (See Fig 04)

POLYGON:- It enables us to draw a polygon consists of more than four sides of regular size.

(a) Polygon Command: - Edge method

The polygon command is used to draw a regular polygon for a given length of the edge or side.

Command: Polygon

Enter no. Of sides<4>: 6

Specify <center of polygon> or edge: E

Specify first end Pt of edge: select using mouse

Specify second end PT of edge: @30< 0

(b) **Draw a polygon of 8 sides with center (50, 50) inscribed in a circle of radius 40 units.**

Command: Polygon

No. Of sides: 8

Edge / < center of Polygon>: 50, 50

Inscribed in a circle / circumscribed about circle (I /C) : I

Radius of Circle –40

(c) Draw a octagon with center (140, 50) circumscribed on a circle of radius 40 units.

Command: Polygon

No. Of Sides: 8

Edge / < center of Polygon>: 140, 50

Inc I/C : C

Radius of circle : 40

Circle: -

Circle is a locus point such that it remains at a fixed point from a fixed distance To obtain a circle a fixed point is taken as a center and at a fixed is taken as a radius.

Circle can be drawn by five methods

(a) Using center and radius

Command: circle

3P / 2P / TTR < center Point >: 6,6

Diameter / <Radius >: 5

(b) Using center and diameter

Command: circle

3P /2P / TTR / < center Point >: 6, 17

Diameter / <Radius >: D

Diameter: 10

(c) Using 3 given point 3P

Command: circle

3P /2P / TTR / < center Point >: 3P

First Pt: (5, 30). Second Pt (4, 6). Third Pt: (10, 25)

(d) Using two given points (2P)

Command : circle

: 2P

First Point on Diameter : (7, 35)

Second Point on Diameter : (7, 47)

(e) Using Tangent, Tangent and Radius (TTR)

You can draw a circle by specifying two lines or two circles or a line and circle and also radius of a circle.

Command**: Circle**: TTR

Select 1st Tangent: Line 1 (using mouse) (16, 4) to (19, 9)

Select 2nd tangent: Line 2 (using mouse) (20, 21) to (21, 7); Radius = 2

Command: circle: TTR

Select 1st Tangent: Point P1 on circle1 (cent 15, 15) radius 2

Select 2nd tangent: Point P2 on circle2 (cent 23, 14) radius 2.5); Radius = 2

Command: Circle: TTR

Select 1st Tangent: Point P1 on circle

Select 2nd tangent: Point P2 on line, Radius = 1.5

Ellipse: -

Two axes that define its length and width determine the shape of an ellipse. The longer axis is called the major axis, and the shorter one is the minor axis. Ellipse can be drawn with 4 methods:

(a) Using major axis end point (10, 20), (60, 20) and minor axis end Pt. (35, 35)

Command: Ellipse

< Axis end Pt> / center: (10, 20), Axis end Pt 2: (60, 20).

< Other axis distance > / Rotation: (35, 35)

(b) Using center of ellipse, end Pt and other axis distance ellipse center (100, 20) major axis end Pt (125, 20) minor axis end Pt (100, 35)

Command: Ellipse

< Axis end Pt 1 > / centre	: C
Centre of Ellipse	: (100, 20)
Axis end Point 2	: (125, 20)
<Other axis distance> / Rotation	: (100, 35)

(c) Using first axis end point and rotation angle of circle around the axis

Draw ellipse using major axis end points (8, 80), (58, 80) & 55^0 rotation around major axis Command: Ellipse

< Axis end Point 1 > / Centre	: (8, 80)
Axis end Point 2	: (58, 80)
< Other axis distance > / Rotation	: R
Rotation about major axis	: 55

A rotation angle of 0^0 produces a circle while a rotation angle of nearly 90^0 produces an ellipse that is almost flat. The system will not accept the entry of 90^0 .

(d) Using center, end point and rotation angle of circle around the axis. Draw ellipse with center point (35, 48), major axis end point (60, 48) and 65^0 rotation.

Command: ellipse

< Axis end point 1 > / center	: C
Centre of Ellipse	: (35, 48)
Axis end point 2	: (60, 48)
< cott axis distance> / ------	: R
65^0 rotation around major axis	: 65^0

Arc: - An arc (usually with arrows at each end) spanning the angle formed by the extension lines of an angle being measured. It can be drawn by different methods)

(a) Using 3 given points (75, 50) (55, 90) (105, 110), Command : Arc

Centre / < start Point > : (75, 50)

Centre / End / < second Point >: (55, 90)

End Point: (105, 10)

(b) Using start Point center and End Point: Command : Arc

Centre / < start Point > : (240, 20)

Centre / End / < second Point > : C

Centre Point : (250, 60)

Angle / length of Chord / < End Point > : (250, 100)

(c) Using start point , centre and included angle.
Command : Arc

Centre / < start Point > : 100, 190

Centre / End / < second Point > : C

Centre Point : 40, 190

Angle / length of Chord / < End Point >(40, 190) : A

Included Angle : 90

(+ve angle and –ve angle draws arc in anticlock wise and clockwise direction)

(d) Using start Point , centre and length of chord

Command : Arc

Centre / < start Point > : 140, 10

Centre / End / < second Point > : C

Centre Point : 100, 10

Angle / length of Chord / < End Point > : L

Length of Chord : 45

(These arcs are always drawn in anti clock wise direction)

(e) Using start Point , End Point and Radius; Command : Arc

Centre / < start Point > : 230, 80

Centre / End / < second Point > : E

End Point : 190, 80

Angle / Dir / Radius / < Centre Point > : R

Radius : 22

(f) Using start Point, End Point and included angle

Command : Arc

Centre / < start Point > : 300, 60

Centre / End / < second Point > : E

End Point : 340, 120

Angle / Dir / Radius / < Centre Point > : A

Included angle: 90

(This type of arc is normally drawn in anti clock wise direction from Point to end Point. If –ve sign is specified the arc is drawn clockwise.)

(g) Using start point , end point and starting direction.

Command : Arc

Centre / < start Point > : 40, 170

Centre / End / < second Point > : E

End Point : 70, 230

Angle / Dir / Radius / < Centre Point > : D

Direction from start point : 120

(h) Using Line / Arc continuation

Draw an arc with end point (200, 150) and to the existing line.

Take line from Point (150, 200) to Point (200, 200);

Command : Arc

Centre / < start Point > :

End Point : (200, 150)

REGION: -

Regions are two-dimensional enclosed areas you create from objects that form closed loops. Loops can be combinations of lines, polylines, circles, arcs, ellipses, elliptical arcs, and splines. The objects that make up the loops must either be closed or form closed areas by sharing endpoints with other objects.

Regions can be used for

- Applying hatching and shading
- Analyzing properties, such as area, using MASSPROP

Extracting design information, such as the centroid.

EXPERIMENT- 4

AIM: - LEARN AND USE THE VARIOUS MODIFY COMMANDS OF THE DRAFTING SOFTWARE

OBJECTIVE: - To learn various drafting modify commands.

THEORY: We can customize a drawing set for publishing using the Publish Drawing Sheets dialog box. We can erase, copy, mirror, offset, move, rotate, break, trim, fillet, chamfer the drawing. The use of these commands as:

ERASE: You can remove objects from your drawing using several methods, including

- Erasing them with ERASE
- Cutting them to the Clipboard
- Pressing DELETE

COPY: We can create duplicates of objects at a specified distance from the original. You specify the distance and direction by two points, a from point (1) and a to point (2), called the base point and the second point of displacement, respectively. These points can be located anywhere within the drawing. Select the objects to copy, right-click in the drawing area, and choose Copy Selection.

Command line: copy

Select objects: Select objects and press ENTER

Specify base point or displacement**, or [**Multiple**]:** Specify a point for a single copy or enter m for multiple copies.

Base Point or Displacement

Makes a single copy. Specify second point of displacement or <use first point as displacement>: Specify a point or press ET

Multiple

Makes multiple copies using one COPY command by typing <m>before selecting the base point. AutoCAD prompts for an insertion base point for the selection object.

Specify base point: Specify second point of displacement or <use first point as displace-

ment>: Specify a point or press ENTER

ARRAY:- You can create copies of objects in a rectangular or polar (circular) pattern called an array. For rectangular arrays, you control the number of rows and columns and the distance between each. For polar arrays, you control the number of copies of the object and whether the copies are rotated. To create many regularly spaced objects, arraying is faster than copying.

Command : Array

Select objects : select the circle using mouse

Select objects : ----

Enter type [Rec/ Pol] <R> : ----

Enter the no.of rows(---) <1> : 2

Enter the no.of columns(111) <1> : 3

Enter dist. below the rows or specify unit cell : 30;40

Command : Array

Select objects: -----

Enter type of array[R or P] : P

Specify center point of the array: select the end point of line at the center of the object

Enter the no. of terms in the array : 8

Specify the angle to fill (+ = CCW- = CW) <360^0> : ----

Rotate arrayed objects [Yes/No] <yes>: ----

Center point of array – the polar multiplying center

No. of items – no of copies

Angle of fill – the fill angle

Rotate objects as they are copied: -

Rotate the copied objects:-

MOVE: You can move objects without changing their orientation or size. By using coordinates and object snaps, you can move objects with precision.

To move an object using two points

1. From the Modify menu, choose Move.

2 Select the objects to move.

Specify a base point for the move; .Specify a second point, the point of displacement

ROTATE: You can rotate objects around a specified point. To determine the angle of rotation, you enter an angle value or specify a second point.

To rotate an object

1. From the Modify menu, choose Rotate.
2. Select the object to rotate.
3. Specify the base point for the rotation.
4. Enter the angle of rotation.
5. Drag the object around its base point and specify a point location to which you want to rotate.

TRIM:

We can shorten or lengthen objects to meet the edges of other objects. We also can trim objects so that they end precisely at boundary edges defined by other objects. Cutting edges can be lines, arcs, circles, polylines, ellipses, splines, regions, blocks, and rays.

To trim an object: :

1. From the Modify menu, choose Trim.
2. Select the objects to serve as **cutting edges.**

   To select all objects in the drawing as potential cutting edges, press ENTER without selecting any objects. Select the objects to trim.

AutoCAD converts a circle to an arc by removing a piece of the circle starting counterclockwise from the first to the second point.

FILLET:- You can change objects to meet in rounded or flattened corners. You can also create gaps in objects. Filleting connects two objects with a smoothly fitted arc of a specified radius.

To set the fillet radius

1. From the Modify menu, choose Fillet.
2. Enter r (Radius).
3. Enter the fillet radius

Select the objects to fillet:

CHAMFER: It is a fast way of creating a line between two nonparallel lines. It is usually used to represent a beveled edge on a corner. CHAMFER can also be used to bevel all corners of a polyline. You can chamfer lines, polylines, and rays

Command: chamfer

(TRIM mode) Current chamfer Dist1 = current, Dist2 = current

Select first line or [Polyline /Distance/Angle/Trim/Method/multiple]:

Select the second line:----------; The object will be chamfered at the given distance:

EXPERIMENT- 5

AIM: - LEARN AND USE THE LAYERS AND BLOCKS IN DRAFTING SOFTWARE

OBJECTIVE: - To learn about layers and blocks in drafting software

THEORY AND PROCEDURE:

BLOCK

You can use several methods to create blocks:

Combine objects to create a block definition in your current drawing.

Use the Block Editor to add dynamic behavior to a block definition in your current drawing.

Create a drawing file and later insert it as a block in other drawings.

Create a drawing file with several related block definitions to serve as a block library.

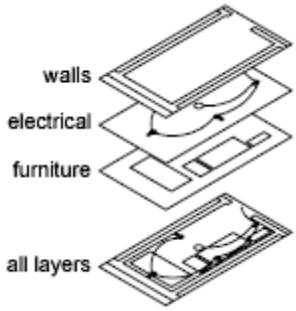
A block can be composed of objects drawn on several layers with various colors, linetypes, and lineweight properties. Although a block is always inserted on the current layer, the block reference preserves information about the original layer, color, and linetype properties of the objects that are contained in the block. You can control whether objects in a block retain their original properties or inherit their properties from the current layer, color, linetype, or lineweight settings.

A block definition can also contain elements that add dynamic behavior to the block. You add these elements to the block in the Block Editor. When you add dynamic behavior to a block, you add flexibility and intelligence to the geometry. When you insert a block reference with dynamic behavior in a drawing, you can manipulate the geometry of the block reference through custom grips or custom properties, depending on how the block was defined.

LAYERS

Layers are used to group information in a drawing by function and to enforce linetype, color, and other standards.

Layers are the equivalent of the overlays used in paper-based drafting. Layers are the primary organizational tool used in drawing. You use layers to group information by function and to enforce linetype, color, and other standards.



By creating layers, you can associate similar types of objects by assigning them to the same layer. For example, you can put construction lines, text, dimensions, and title blocks on separate layers. You can then control

Whether objects on a layer are visible in any viewports

Whether and how objects are plotted

What color is assigned to all objects on a layer

What default linetype and lineweight are assigned to all objects on a layer

Whether objects on a layer can be modified

Every drawing includes a layer named 0. Layer 0 cannot be deleted or renamed. It has two purposes:

Ensure that every drawing includes at least one layer

Provide a special layer that relates to controlling colors in blocks

EXPERIMENT –6

AIM: - USE HATCHING AND DIMENSIONING TO DETAIL OUT A COMPONENT DRAWINGS

OBJECTIVE: - An awareness about the hatching and dimensioning in Auto Cad.

THEORY: - You can add measurements to your drawing with several dimensioning commands. Use dimension styles to store your dimension format settings and maintain dimensioning standards.

- Understand Basic Concepts of Dimensioning
  You can create several types of dimensions, and you can control their appearance by setting up dimension styles or by editing individual dimensions.
- Use Dimension Styles
  You can control the appearance of dimensions by changing settings. For convenience and to help maintain dimensioning standards, you can store these settings in dimension styles.
- Set the Scale for Dimensions
  You can specify the size of dimensions in your drawing. How you set dimension size depends on the method you use to lay out and plot drawings.
- Create Dimensions
  You can create all standard types of dimensions.
- Modify Existing Dimensions
  You can modify all components of the existing dimension objects in a drawing either individually or by using dimension styles.

Type of Dimensioning:

- Linear Dimensioning

  (a) Horizontal Dimensioning (b) Vertical Dimensioning

- Angular Dimensioning
- Aligned Dimensioning
- Drawing leader Line

Dimensioning

Command : DIM or DIMLIN

DIM: HOR

Specify first text line origin < select object > : select with mouse

Specify second text line origin < select object > : select 2nd ext Pt.

Specify dim line location or [ntext / Text / Angle]:

select location away from the object

Enter dimtext <by default > : type a rounded dim or press enter

DIM : VER same as above

DIM : Press enter to complete dimensioning

Command: DIM or DIM ALIGNED

DIM: ALIGNED Same as above

Command : DIM: ANGULAR

Select arc, circular line or < specify vertex> : select 1st line

Select 2nd line : select 2nd

Specify dimension arc line location [M Text / Text / Angle]: specify location

Leader Line. If we want to draw a dimensioning of a number of object of same size, then we use the Leader Line. For example, there are 10 holes in a block of 10mm diameter, then the dimensioning of that case for a hole is considered for all holes.

DIM: LEADER or lea

Start point of a line

Endpoint of line

Type the text according to size of object.

HATCHING

You can hatch an area using a predefined hatch pattern, define a simple line pattern using the current linetype, or create more complex hatch patterns. One type of pattern is called solid, which fills an area with a solid color.

You can also create a gradient fill, which uses a transition between shades of one color or between two colors. Gradient fills can be used to enhance presentation drawings, giving the appearance of light reflecting on an object.

Define the Boundaries of a Hatch

You can choose from several methods to specify the boundaries of a hatch.

Specify a point in an area that is enclosed by objects.

Select objects that enclose an area.

Drag a hatch pattern into an enclosed area from a tool palette or DesignCenter.

When you hatch a drawing, whole or partial objects that are not part of the object boundary are ignored.

If a hatch line encounters an object such as text, an attribute, or a solid-fill object, and if the object is selected as part of the boundary set, HATCH hatches around the object.

To hatch areas

1. Click Draw menu » Hatch.
2. In the Hatch and Gradient dialog box, click Add: Pick points.
3. In your drawing, specify a point inside each area that you want hatched, and then press ENTER.

   This point is known as the internal point.
4. In the Hatch and Gradient dialog box, Hatch tab, in the swatch box, verify that the sample pattern is the pattern you want to use. To change patterns, select another pattern from the Pattern list.

   To see how the hatch pattern will look, click the [...] button next to Pattern. Click OK when you finish previewing.
5. In the Hatch and Gradient dialog box, make adjustments, if necessary.

   You can specify new hatch boundaries by clicking Add Boundaries or Remove Boundaries.
6. Under Draw Order, click one of the options.

   You can change the draw order of the hatch so that the hatch is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.
7. Click OK.

EXPERIMENT – 7

AIM :- UNDERSTAND DIFFERENT COORDINATE SYSTEM AND DO A EXERCISE ON DRAFTING SOFTWARE

OBJECTIVE :- An awareness about the different coordinate system

THEORY:-

There are four different types of coordinate systems used in AutoCAD to locate the point on screen. In AutoCAD considering the screen as XY plane does 2D drafting. X-value is consider horizontally and Y- value are taken vertically. By default ,the lower left corner is consider as origin(0,0) AUTOCAD uses the following coordinate systems:

(a) Absolute coordinate system

(b) Relative coordinate system

(C) Polar coordinate system

(d) Direct Distant Entry System

(a) Absolute coordinate system:

In this method ,the points are locate to draw with respect to the origin(0,0)To mark a point values is given in pairs for X-coordinate value followed by Y-coordinate.

Example:

Draw a line from point (5,5) to Pt (10,10)

Command:- Line

From Pt:- 5,5

To Pt:- 10,10

To Pt:-

(b) Relative Co-ordinates:-

In this method , the points are locate to draw a line with reference to the previous point.

Example:

Draw a line from Pt (2,2) to Pt 5 units in X-axis and 8 units in Y axis relative to first coordinate.

Command: Line

From Pt: 2, 2

To Pt: @ 5, 8

To Pt:

Draw the fig. with help of relative method

Specify First Pt:- 20, 20

Specify next Pt:- @ 60, 0

Specify next Pt:- @ 0, 30

Specify next Pt:- @ -40, 0

Specify next Pt:- @ 0, 20

Specify next Pt:- @ -20, 0

Specify next Pt:- @ 0, -50

©Polar Coordinates:-

The points are located to draw a line by defining the distance of the point from the current position and the angle made to that line.

Specify First Pt:- (20, 20)

Specify next Pt:- @ 60< 0

Specify next Pt:- @ 30< 90^0

Specify next Pt:- @ 40< 180

Specify next Pt:- @ 20< 90^0

Specify next Pt:- @ 20< 180 ;

Specify next Pt:- @ 50< 270^0

Direct DistanceEntry System:- In this method are locat to draw a line using the distance entry in the direction of cursor.Ortho mode is used here (F-8) to draw a drawing

To draw a line using the dist entry in the dir of the cursor.

First Pt. : 20, 20 move mouse horizontally right

Next Pt. : 60 move mouse vertically up

Next Pt : 30 moves mouse horizontally left

Next Pt : 40 moves mouse vertically up

Next Pt : 20 moves mouse horizontally left

Next Pt : 20 moves mouse vertically down

Next Pt : 50

EXPERIMENT – 8

AIM: - DRAW THE DIFFERENT TYPES OF 3D MODELING ENTITIES USING VIEWING COMMANDS TO VIEW THEM

THEORY AND PROCEDURE: - You can create a solid cone with a circular or an elliptical base tapering to a point. You can also create a cone frustum, which tapers to a circular or elliptical planar face that is parallel to its base.

Command entry: cone

Specify center point of base or [3P/2P/Ttr/Elliptical]: *Specify a point (1) or enter an option*

Specify base radius or [Diameter] <default>: *Specify a base radius, enter* **d** *to specify a diameter, or press* ENTER *to specify the default base radius value*

Specify height or [2Point/Axis endpoint/Top radius] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Use the Top Radius option to create a cone frustum.

Initially, the default base radius is not set to any value. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive.

Command entry: box

Specify first corner or [Center]: *Specify a point or enter* **c** *for center*

Specify other corner or [Cube/Length]: *Specify the other corner of the box or enter an option*

If the other corner of the box is specified with a *Z* value that differs from the first corner, then no height prompt is displayed.

Specify height or [2Point] <default>: *Specify the height or enter* **2P** *for the 2 Point option*

Entering a positive value draws the height along the positive *Z* axis of the current UCS. Entering a negative value draws the height along the negative *Z* axis.

Command entry: cylinder

Specify center point of base or [3P/2P/Ttr/Elliptical]: *Specify a point (1) or enter an option*

Specify base radius or [Diameter] <default>: *Specify a base radius, or enter* **d** *to specify a diameter, or press* ENTER *to specify the default base radius value*

Specify height or [2Point/Axis endpoint] <default>: *Specify a height (2), enter an option, or press* ENTER *to specify the default height value*

Command entry: pyramid

4 sides *(default)*

Circumscribed *(default)*

Specify center point of base or [Edge/Sides]: *Specify a point or enter an option*

One of the following prompts is displayed:

Specify base radius or [Inscribed] <default>: *Specify a base radius, enter* **i** *to change the pyramid to circumscribed, or press* ENTER *to specify the default base radius value*

Specify base radius or [Circumscribed] <default>: *Specify a base radius, enter* **c** *to change the pyramid to circumscribed, or press* ENTER *to specify the default base radius value*

Initially, the default base radius is not set to any value. During a drawing session, the default value for the base radius is always the previously entered base radius value for any solid primitive.

After specifying the base radius and whether the pyramid is inscribed or circumscribed, the following prompt is displayed:

Specify height or [2Point/Axis endpoint/Top radius] <default>: *Specify a height, enter an option, or press* ENTER *to specify the default height value*

Similiarly following commands are also part of 3D modeling.

Commands

EXTRUDE	Creates a 3D solid or surface by extruding an object or planar face a specified distance and direction
LOFT	Creates a 3D solid or surface by lofting through a set of two or more curves
POLYSOLID	Creates a 3D polysolid
REVOLVE	Creates a 3D solid or surface by revolving 2D objects about an axis
SPHERE	Creates a 3D solid sphere
SWEEP	Creates a 3D solid or surface by sweeping a 2D curve along a path
TORUS	Creates a 3D donut-shaped solid
WEDGE	Creates a five-sided 3D solid with a sloped face tapering along the X axis

EXPERIMENT –9

AIM: - DRAW THE DIFFERENT SURFACE MODEL WITH DIFFERENT EDITING COMMANDS

THEORY AND PROCEDURE:-

You can create surfaces from existing objects in your drawing.

With the CONVTOSURFACE command, you can convert any of the following objects into surfaces:

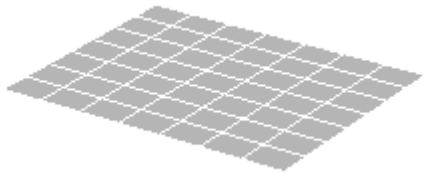
- 2D solids
- Regions
- Bodies
- Open, zero-width polylines with thickness
- Lines with thickness
- Arcs with thickness
- Planar 3D faces

You can create surfaces from 3D solids with curved faces, such as a cylinder, with the EXPLODE command.

You can use the PLANESURF command to create a planar surface. Use either of the following methods:

- Select one or more objects that form one or more enclosed areas
- Specify the opposite corners of a rectangle

When you specify the corners of the surface, the surface is created parallel to the workplane.



You can use the CONVTOSOLID command to convert the following objects into extruded 3D solids:

- Uniform-width wide polylines with thickness
- Closed, zero-width polylines with thickness
- Circles with thickness

Command entry: edgesurf

Current wire frame density: SURFTAB1=current SURFTAB2=current

Select object 1 for surface edge:

Select object 2 for surface edge:

Select object 3 for surface edge:

Select object 4 for surface edge:

You must select the four adjoining edges that define the mesh patch. The edges can be lines, arcs, splines, or open 2D or 3D polylines. The edges must touch at their end-points to form a topologically rectangular closed path.

Command entry: tabsurf

Select object for path curve:

The path curve defines the approximated surface of the polygon mesh. It can be a line, arc, circle, ellipse, or 2D or 3D polyline. The mesh is drawn starting at the point on the path curve closest to the selection point.

Select object for direction vector: Select a line or open polyline

Only the first and last points on a polyline are considered, and intermediate vertices are ignored. The direction vector indicates the direction and length of the shape to be extruded. The end selected on the polyline or line determines the direction of the extrusion. The original path curve is drawn with wide lines to help you visualize how the direction vector dictates the construction of a tabulated mesh.

EXPERIMENT –10

AIM: - LEARN AND USE SHADING AND RENDERING TECHNIQUES FOR BETTER VISUAL APPEARANCE

OBJECTIVE: - AN AWARENESS ABOUT SHADING AND RENDERING 3D IN AUTO CAD.

THEORY AND PROCEDURE: -

SHADING

The options for shaded viewport plotting give you a large degree of flexibility in conveying your three-dimensional designs to others. You can convey your design intent by choosing how viewports are plotted and by specifying resolution levels.

With shaded plotting options, you can choose whether to plot a set of shaded objects as displayed or in wireframe, hidden mode, a visual style, or rendered. Shaded and rendered viewports are plot-previewed, plotted, plotted to file, and published with full shading and rendering.

You can use realistic plots in your presentations by plotting viewports as they are displayed on the screen or otherwise.

Shaded viewport plotting options apply to all objects in viewports and model space. If you use the Shaded or Rendered options plot style tables included in the page setup do not affect plots. If you use the Render option, two-dimensional wireframe objects, such as lines, arcs, and text, are not plotted.

The Visual Styles Manager displays sample images of the visual styles available in the drawing. The selected visual style is indicated by a yellow border, and its settings are displayed in the panel below the sample images.

When the Dashboard is displayed, you can change some frequently used settings directly or open the Visual Styles Manager.

Five default visual styles are supplied with the product:

2D Wireframe. Displays the objects using lines and curves to represent the boundaries. Raster and OLE objects, linetypes, and lineweights are visible.

3D Wireframe (upper left in the illustration). Displays the objects using lines and curves to represent the boundaries.

3D Hidden (upper right). Displays the objects using 3D wireframe representation and hides lines representing back faces.

Realistic (lower left). Shades the objects and smooths the edges between polygon faces. Materials that you have attached to the objects are displayed.

Conceptual (lower right). Shades the objects and smooths the edges between polygon faces. Shading uses the Gooch face style, a transition between cool and warm colors rather than dark to light. The effect is less realistic, but it can make the details of the model easier to see.

RENDERING:

Creates a photo realistic or realistically shaded image of a three-dimensional wire frame or solid mode

To render a model

1. Display a 3D view of your model.

2. From the View menu, choose Render.
3. In the Render dialog box, set options or accept the defaults.
4. Under Rendering Options, select Smooth Shading to smooth the edges between the polygon faces.

Related to Smooth Shading is Smoothing Angle, which sets the angle at which AutoCAD interprets an edge. The default angle setting is 45 degrees. Angles less than 45 degrees are smoothed; angles greater than 45 degrees are considered edges.

4. To render the image to the screen, make sure that Destination is set to Render Window or View port.

5. Select a named scene or the current view.

6. Choose Render.

Depending on the size of the drawing, after a short or long pause AutoCAD displays a rendered image of your model.

Note: If your objects are zoomed out past the limits of the drawing and you are having rendering problems, try scaling the scene or zooming in to at least the limits of the drawing.

After creating a rendering, you can save the image for redisplay at a later time. Rendering can be a time-consuming process, but redisplaying a previously rendered image is instantaneous.

To save a rendered image, you can render directly to a file, or you can render to the screen and then save the image

EXPERIMENT –11

AIM: USE AND LEARN IMPORT/EXPORT TECHNIQUES AND CUSTOMIZATION OF DRAFTING SOFTWARE

THEORY AND PROCEDURE: -

The image or a drawing can be imported by using xref command. After that browse the file on the computer. You can perform several operations on referenced drawing (xref) files. Start with attaching, setting paths, and detaching.

- **Attach Drawing References (Xrefs)**
  When you attach a drawing as an *xref*, you link that referenced drawing to the current drawing; any changes to the referenced drawing are displayed in the current drawing when it is opened or reloaded.
- **Set Paths to Referenced Drawings**
  You can view and edit the file name and path used when locating a particular drawing reference (xref). Use this option if the referenced file has been moved to a different folder or renamed since it was first attached.
- **Detach Referenced Drawings**
  To completely remove DWG references (xrefs) from your drawing, you need to detach them rather than erase them

CUI:-

Using AutoCAD's customization tools, you can tailor your drawing environment to suit your needs.

Customization capabilities, including the CUI (Customize User Interface) file format and the Customize User Interface editor, help you to easily create and modify customized content. The XML-based CUI file replaces the menu files used in releases prior to AutoCAD 2006. Instead of using a text editor to customize menu files (MNU and MNS files), you customize the user interface from within AutoCAD. You can

- Add or change toolbars and menus (including shortcut menus, image tile menus, and tablet menus)
- Create or change workspaces
- Assign commands to various user interface elements
- Create or change macros
- Define DIESEL strings
- Create or change aliases
- Add tooltips

- Provide descriptive text on the status line