GOVT POLYTECHNIC , AVARGUPPA

SIDDAPUR .

COMPUTER AIDED ENGINEERING, GRAPHICS.

FOR SECOND SEM DIPLOMA IN

CS , ENGINEERING COURSE.

REG NO:- **NAME :** **ROLL NO:-**

CERTIFICATE

THIS IS TO CERTIFY THAT _____ HAS SATISFACTORILY COMPLETED THE EXERCISES IN COMPUTER AIDED ENGINEERING, GRAPHICS, PRESCRIBED BY THE BOARD OF TECHNICAL EDUCATION /EXAMINATIONS, BANGALORE FOR SECOND SEM DIPLOMA IN COMPUTER SCIENCE ENGINEERING COURSE, DURING THE YEAR _____

DATE :- **SIGNATURE OF LECTURER**

1. **REG NO OF THE STUDENT :---**
2. **EXAMINATION CENTER :---**
3. **SIGNATURE OF THE HEAD OF THE DEPT:--**
4. **SIGNATURE OF THE EXAMINERS :-- 1.**

2.

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Course Assessment And Evaluation Chart

Date	Particulars	Max. Marks allotted 100 Marks [Except CAEG Practice (20)]	Marks Obtained for100 [Except CAEG Practice(20)]	Conversion for 20	Staff Initials
Entire SEM	CAEG Practice (20)	20			
	SKILL TEST -1	100			
	SKILL TEST -2	100			
	SKILL TEST -3	100			
	Total CIE Marks [for 60Marks]				
	Total CIE Marks In Words :--				

INTRODUCTION:

The graphical representation of an object is called drawing. A drawing of an object which contains all the necessary information like actual shape, accurate sizes, manufacturing methods, etc. required for the construction of the object is called Engineering drawing.

1. **Since the engineering drawing of an object is drawn well before the object is constructed, the person who draws the engineering drawing should have a clear picture of the shape of the object in his mind.**

2. **Since understanding of an engineering drawing involves much of imagination, a systematic method is to be followed for its drawing and reading.**

Drawing Instruments:

Since the engineering drawing shows the accurate sizes and actual shapes of the object, it should be drawn to scale with the aid of geometrical instruments. The accuracy and neatness of the drawing also depend on the selection and the proper method of using the instruments.

1. **Drawing board**
2. **T-square**
3. **Set square**
4. **Protractor**
5. **Clinograph**
6. **Drafting machine**
7. **Minidrafter**
8. **Compass**
9. **Bow compass**
10. **French curves**
11. **Drawing pins or Clips or tape**
12. **Eraser**
13. **Erasing shield**
14. **Scales**
15. **Drawing sheets**

1. Drawing board:

1. **Drawing board is made of well-seasoned pine wood.**
2. **Along the left edge of the board, a groove is cut and a perfectly straight ebony edge called working edge is inserted into the groove.**
3. **The working edge provides a guide for the T- square to slide on.**
4. **The drawing board should be placed on the table, always with the working edge at the left side.**
5. **The standard sizes of the drawing boards recommended by Bureau of Indian Standards IS: 1444-1963.**

Designation	Length x Width in mm	Recommended for use with sheet sizes
D0	1500 x 1000	A0
D1	1000 x 700	A1
D2	700 x 500	A2
D3	500 x 350	A3

2. T-Square:

1. **This is made of well-seasoned hard wood and consists of two parts, namely blade and stock.**
2. **The stock slides over the working edge of the drawing board.**
3. **The blade which moves on the surface of the board has a drawing edge.**
4. **The drawing edge is beveled and in some types of T- squares, it is made of transparent plastic edge so that the immediate lines hidden by it will be visible.**
5. **Its main use is to draw horizontal lines and it also serves as a support to place the set-square over it.**

3. Set square:

1. **These are two in number. One is 60°-30° set square and another is 45° set square.**
2. **These are made of celluloid.**
3. **These are used to draw vertical, inclined and parallel lines and for setting the combination of the angles in multiples of 15°.**

4. Protractor:

1. **This is made of celluloid.**
2. **Protractor will be either semicircular or circular.**
3. **It is used to set up or measure any given angle.**

5. Clinograph:

1. **The clinograph is a simple drawing instrument which is normally working together with a T-square and functions like a section liner or an adjustable set square.**
2. **To use, one of the two fixed edges is first placed against the T-square.**
3. **The adjustable edge is then rotated to the desired angle of inclination and it is held there by friction of the brass joint.**
4. **To draw parallel inclined lines to represent section areas or for shading, the whole device is slide along the edge of the T-square and stopped at every equal spaced position to draw the lines.**
5. **It can also be used for drawing parallel and perpendicular lines along the other fixed edge.**

6. Drafting machines:

1. **Drafting machines are used by professional draftsmen to produce the drawings with ease and rapidity.**
2. **A drafting machine combines all the functions of the T-square, set squares, protractor and clinograph.**

7. Mini drafter:

1. **A simplified miniature version of the drafting machine called mini drafter.**
2. **It consists of a clamp provided with a screw, two pairs of parallelogram of bars, a protractor head with a screwed knob and two metallic or celluloid scales with ruling edges.**
3. **The two pairs of parallelogram of bars are pivoted at right angles to a pivot plate.**
4. **One pair of parallelogram of bars is pivoted to the clamp while the other pair is pivoted to the protractor head and can be swiveled to any angular position.**
5. **The knob provided at the protractor head facilitates the clamping of the scales in any angular position.**

8. Compass:

1. **The compass is used for drawing circles and arcs of circles.**
2. **It consists of two legs hinged together at its upper end.**
3. **A pointed needle is fitted at the lower end of one leg, while a pencil lead is inserted at the end of the other leg.**

9. Bow compass:

A small pair of compasses, one leg of which carries a pencil or pen for drawing circles. Its legs are often connected by a bow shaped spring rather than a joint. Bow compasses are used to draw arcs and circles having a radius of less than 20mm.

10.French curves:

1. **French curves are templates of various curved shapes.**
2. **The French curve is used as a guiding edge for drawing non circular curves.**
3. **First sufficient numbers of points on the curve to be drawn are determined.**
4. **A smooth continuous free hand curve is drawn passing through all the points.**
5. **Then place the French curve selecting a part of it that will fit the portion of the curve, draw only that portion of the curve using the French curve guiding edge.**
6. **Next shift the French curve to find it's another portion that will fit to continue the next portion of the curve.**

11.Drawing pins or Clips or tape:

1. **These are used to secure the drawing sheets to the drawing board firmly.**
2. **The use of pins should be avoided as it will spoil the surface of the board.**

12.Eraser:

1. **Soft India rubber is the widely used form of eraser. Nylon rubber is also used.**

13.Erasing shield:

1. **It is a thin plastic or metallic plate cut with slots of different lengths, widths and shapes used to erase unwanted pencil lines without erasing the surrounding lines.**

14.Scales:

1. **Scales are measuring devices usually made either of cardboard or celluloid. Stainless steel scales are more durable.**
2. **15cm long and 2cm wide or 30cm long and 3cm wide flat scales are common in use.**

15.Drawing sheets:

1. **The standard sizes of the drawing sheets recommended by the Bureau of Indian Standards are given below**

Designation	Trimmed size in mm
A0	841 x 1189
A1	594 x 841
A2	420 x 594
A3	297 x 420
A4	210 x 297

Layout of drawing sheets:

1. **The layout of drawing sheet is the standard arrangement to ensure that all the necessary information is included in the drawing sheet and facilitate its quick reading and to make it possible for essential references to be located easily, especially when drawings are prepared by several offices.**
2. **The width of the grid reference system shall be approximately half the width of the border for a given size of sheet.**

3. **Border: the border is the space left all round in between the trimmed edges of the drawing sheet and the frame. It has been recommended that these borders have a minimum width of 20mm for A0 and A1 sizes, and a minimum width of 10mm for A2, A3 and A4 sizes.**
4. **Filing margin: a filling margin for taking perforations may be provided. This margin shall have a minimum width of 20mm with the border included therein.**
5. **Grid reference system: a grid reference system is provided for all the sizes of drawing sheets to facilitate easy location of any portion of the drawing within the frame. The length and width of the frames are divided into even number of divisions. The number of divisions chosen for a particular sheet depends on the complexity of the drawing.**
6. **Title block: a title block is an important feature provided on a technical drawing or with an associated document, for recording the technical and administrative details. It is placed on the right hand bottom corner of the drawing sheet. The space in the title block is divided into (i) identification zone and (ii) additional information zone.**

Types of lines and their applications:

For general engineering drawings the types of lines are recommended by the Bureau of Indian Standards.

Different types of lettering as per I.S.I:

An engineering drawing besides illustrating the shape of an object, is required to show the sizes of the object and some worded details to convey the technical information such as, the name of the material of the object, brief production instructions if any, title, name of the company, initials of the personnel designed, drawn and inspected, etc.,. Writing of these particulars and the sizes of the objects on an engineering drawing is called lettering.

Indian standard recommendations IS: 9609

Single stroke lettering is the style of letters adopted by the Bureau of Indian Standards, IS: 9609, for use on engineering drawings. The term single stroke means that the width of the stem of the letters and numerals is uniformly thick, equal to the thickness of lines produced by the tip of the pencil or pen.

The letters and numerals may be written either vertical or inclined.

An inclination of 15° to the right of the vertical is recommended for inclined letters and numerals.

DIMENSIONING :-INTRODUCTION:

Indicating on a drawing, the sizes of the object and other details essential for its construction and function using lines, numerals, symbols, notes, etc., is called dimensioning.

ELEMENTS OF DIMENSIONING:

The elements of dimensioning include the extension lines, dimension line, arrow heads, dimension figure, leader line etc.

1. **Extension lines: extension lines are drawn at the two end points of the edge or a line to be dimensioned. The extension lines are drawn at right angles to the line to be dimensioned. It is a TYPE B line as illustrated in figure.**
2. **Dimension lines: dimension lines are drawn between the two extension lines and are parallel to the line or edge to be dimensioned. The dimension line is originated and terminated by the arrow heads as shown in figure. It is also a TYPE B line.**
3. **Arrow heads: arrow heads are used to indicate the origin and termination of the dimension line as shown in figure. The included angle between the lines of the arrow heads is recommended as 15°. The length of the arrow is at least three times the width of the arrow head. In engineering drawing three types of arrow heads are preferred. They are open, closed and closed and filled as shown in figure.**
4. **Dimension figure: dimension figure is the number indicating the length of the line or the edge where it is placed. The unit preferred in engineering drawing is MILLIMETER. The unit of the number is generally not mentioned along with it. Instead a note mentioning ALL DIMENSIONS ARE IN MM is written at the selected place on the drawing sheet.**
5. **Leader line: leader line is drawn from a point at which a dimension figure or a note is to be placed. It is drawn at 30°, 45° or 60° to the horizontal or the main axis of the object. It is drawn at an angle first and then continued as a horizontal line and a dimension figure or a note is placed along with the horizontal line as shown in figure.**

SYSTEMS OF DIMENSIONING:

There are two systems of dimensioning. The aligned system and the unidirectional system.

1. **Aligned system: In aligned system of dimensioning the dimension figure or the number is placed above the dimension line and is written parallel to the dimension line. For the vertical dimension lines, the figure is placed by looking from the right side of the drawing sheet. The use of aligned system is shown in figure.**
2. **Unidirectional system: In the unidirectional system the dimension figure or the number is placed by breaking the dimension line at the middle. The use of unidirectional system is shown in figure.**

TYPES OF DIMENSIONING:

There are four types of dimensioning.

1. **Chain dimensioning: In chain dimensioning the dimensions of each edge or the line are shown separate.**
2. **Parallel dimensioning: In parallel dimensioning the dimensions of the different edges are shown from a datum or reference line.**
3. **Progressive dimensioning: In this type of dimensioning, dimensions of the edges or the lines are mentioned from a datum like in parallel dimensioning, but with a single dimension line. The dimension line is originated with a dot and is terminated with an arrow head.**
4. **Combined dimensioning: In chain, parallel and progressive dimensioning any of the two types of dimensioning are combined then it will be a combined dimensioning.**

INTRODUCTION TO SCALE:-The drawing drawn as same as the size of the object is called full scale drawing. The drawing drawn to the smaller size is called reduced drawing. The drawing drawn to the larger size is called enlarged drawing.

The proportion by which the drawing of a given object is enlarged or reduced is called the scale of the drawing. This is indicated by means of a ratio called Representative Fraction (R.F).

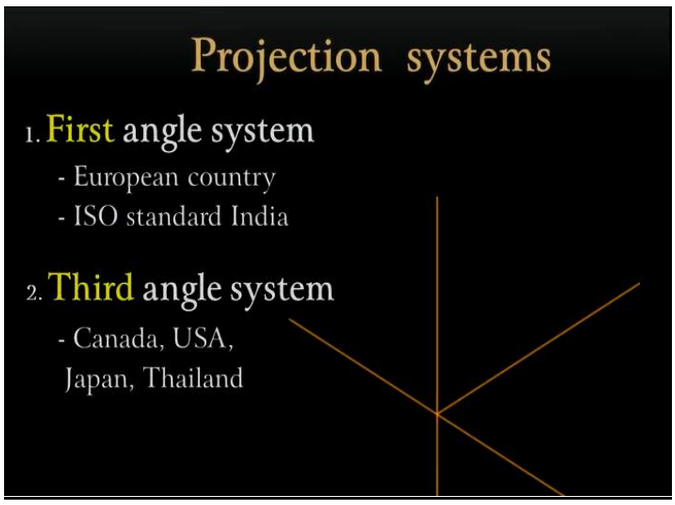
The ratio of length of an object on drawing to the actual length of the object is defined as Representative Fraction.

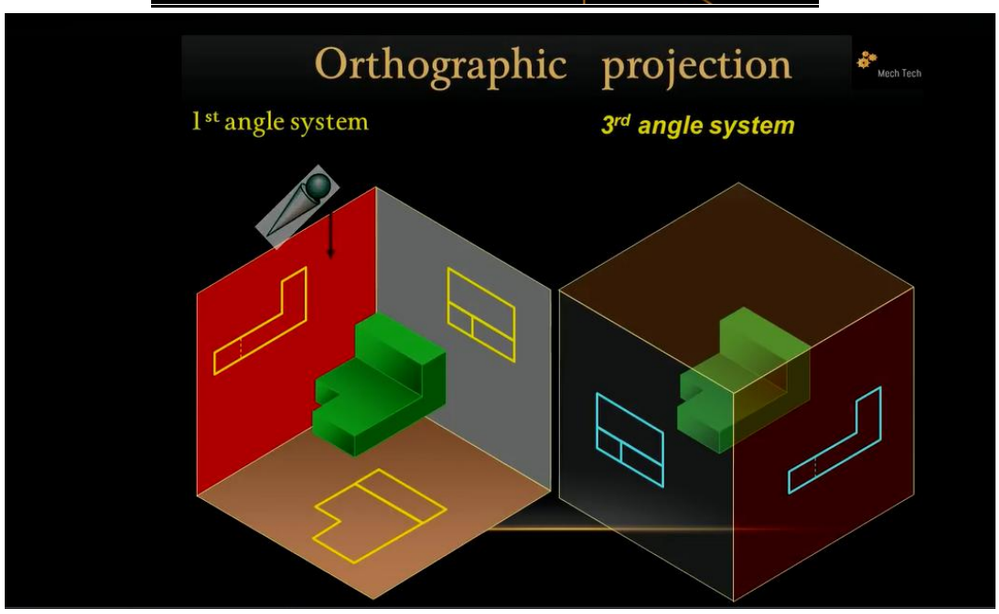
Representative Fraction = Drawing size / Actual size

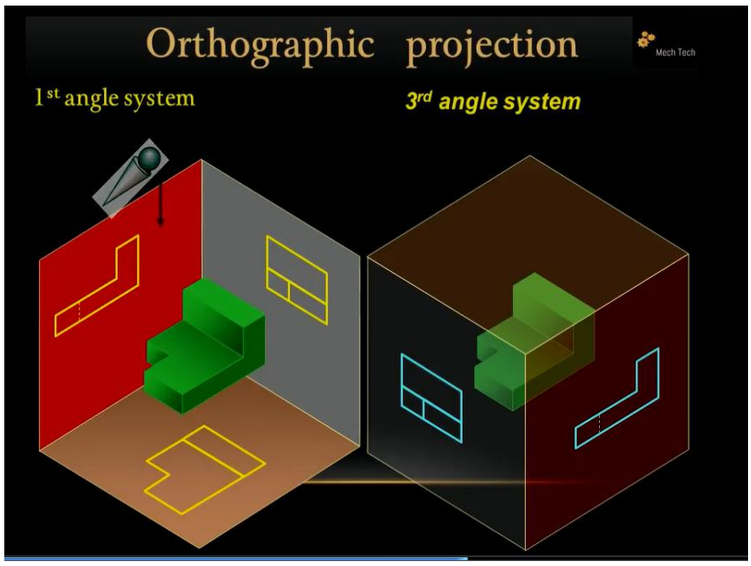
Types Of Lines

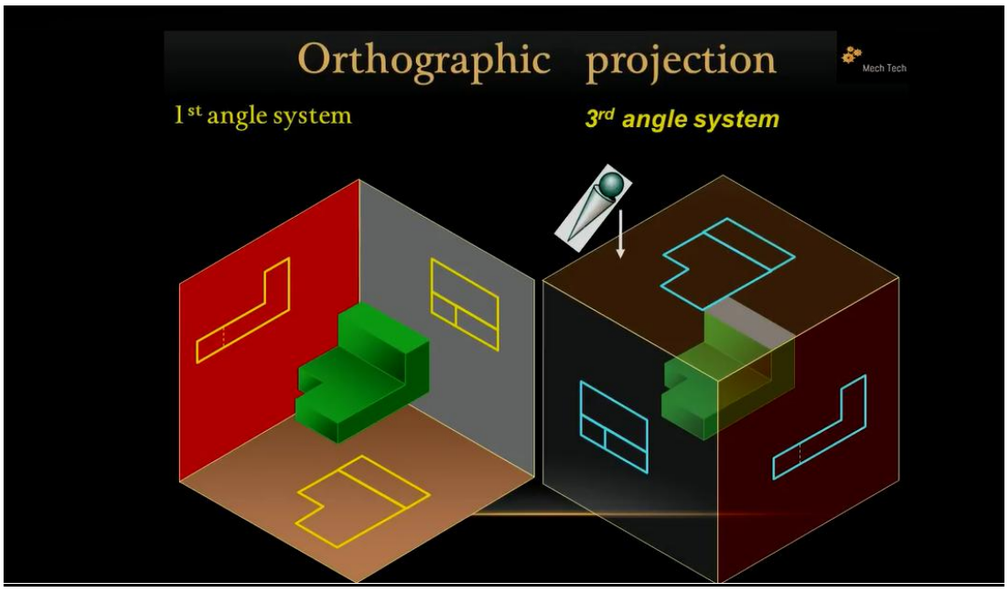
TYPE	ILLUSTRATION	APPLICATION
A	Continuous THICK	Visible outlines, visible edges.
B	Continuous THIN	Dimension lines, projection lines, leader lines, imaginary lines of intersections, outlines of revolved sections.
C	Continuous THIN freehand	Boundaries or limits of partial or interrupted views.
D	Continioous Thick Continuous THIN zig-zag	
E	Dashes THICK	Hidden outlines, hidden edges.
F	Dashes THIN	
H I	Chain THICK thin Thick	Cutting planes. Indication of surfaces to which specialrequirements applies.
J	Chain THIN double dash	Outlines of adjacent parts, alternate and extreme positions of movable parts, centroidal lines, initial outlines.

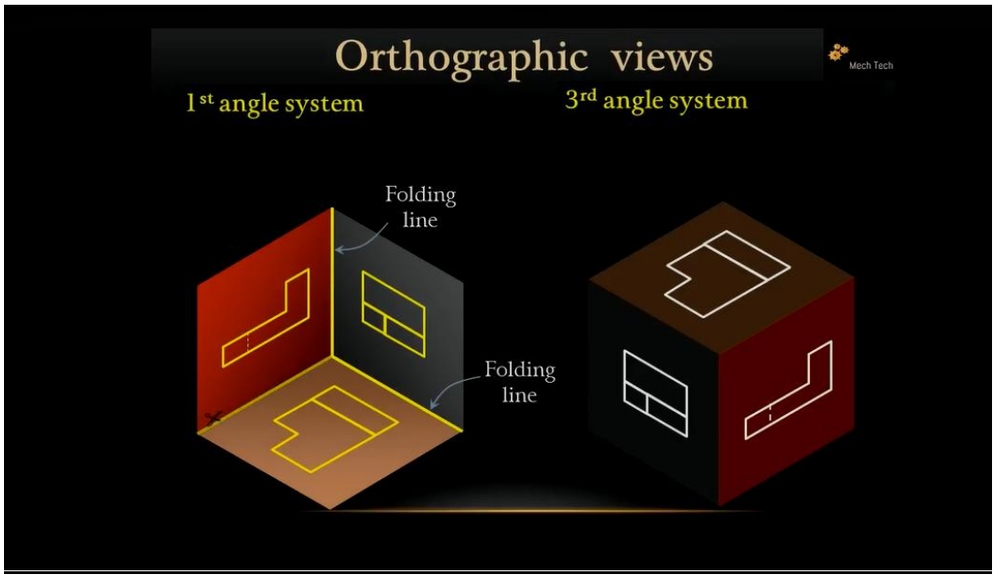
INTODUCTION TO ORTHOGRAPHIC PROJECTION

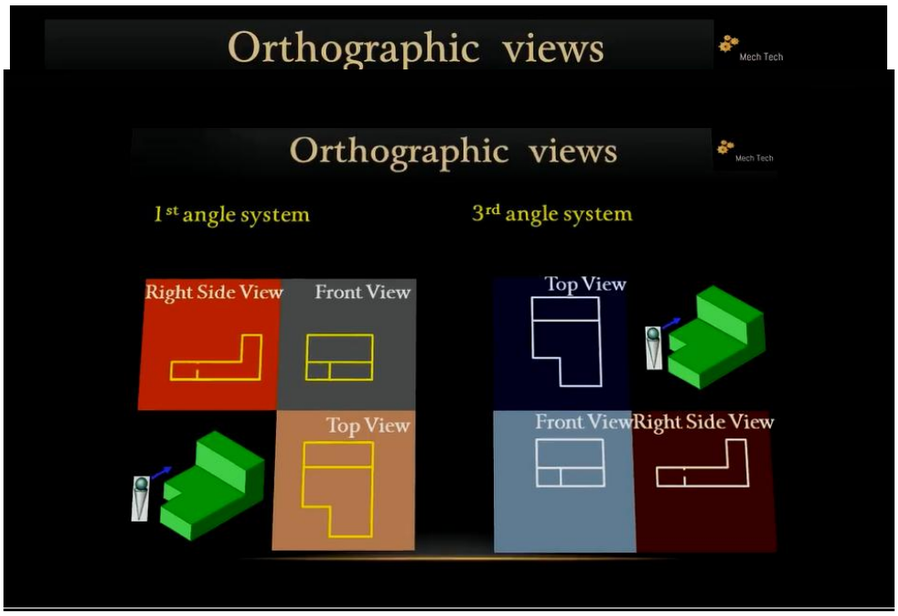


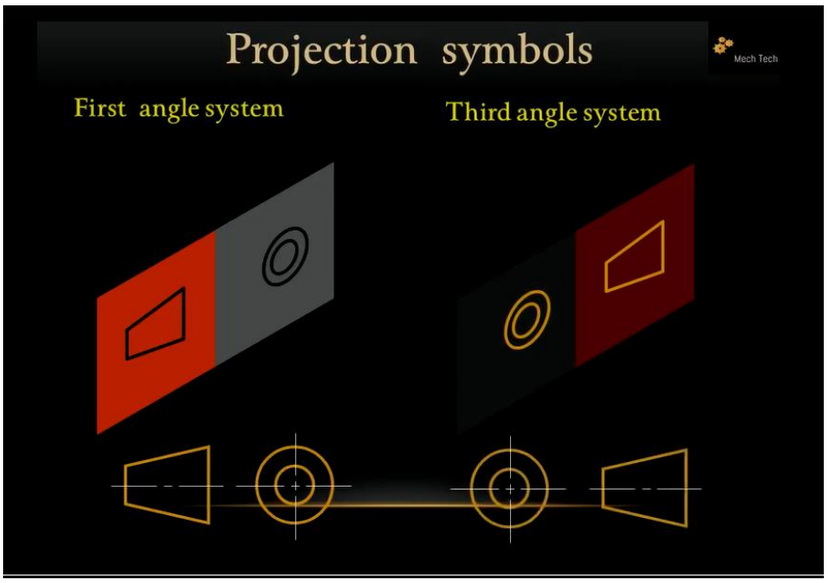





Orthographic projection
Mech Tech
1st angle system
3rd angle system


Orthographic views
Mech Tech
1st angle system
3rd angle system
Folding line
Folding line





INTRODUCTION TO COMPUTER AIDED DRAWING /GRAPHICS

INTTRODUCTION

Computers use different software's to perform tasks. With the help of special software's, computerscan be advantageously used to do the work of drafting. The process of constructing the drawings onthe computer screen with the help of specially developed software's and hardware's is called **"COMPUTER AIDED DRAFTING**". The drawings in CAD are clear and more exact than manual drawings. The CAD system is based on what is called interactive computer graphics (ICG). ICG helps to convert the data entered by the user in form of graphics.

AUTOCAD: AUTOCAD is very effective CAD software. It is used globally by CAD professionals. It supports 2D drafting and 3D modeling. AUTOCAD is user-friendly and easy to learn. Explaining AUTOCAD in 3D environment is beyond the scope.

SOFTWARE FOR CAD: Auto cad is one of the following CAD system software introduced in DEC 1982 July by Autodesk Ltd. Sausalito, California of USA. The latest one is auto cad 2022 released in 30 March 2021.The entire auto cad drawing created by using the old version can be read by the subsequent version.

CAD workstation: A CAD workstation, in its simplest form, consists of a computer with a keyboard, mouse and monitor and loaded with CAD software. The keyboard and mouse are essential input devices whereas monitor is a real time output device. All the three are integral parts ofcomputer and are always connected to central processing unit. For CAD applications, a computer with reasonably good processing unit is recommended.

ADVANTAGES OF CAD:

CAD offers the following advantages.

1. **Accuracy**: CAD helps to achieve very high degree of accuracy that is impossible to achieve manually.

2. **Speed**: With sufficient practice, a user can create the drawings specially. Similar objects can be copied (or) arrayed which saves time required for duplication. Easy editing: Drawings once constructed can easily be edited or modified as and when needed.

3. Space Effectiveness: A computer can store several thousand drawing files over a long period of time.

4. Standard Libraries: "CAD" software has libraries containing drawings of standard parts such as palettes panels.

5. **Scaling:** A drawing can be enlarged (or) reduced by any scale factor.

6. Better visualization: Use of different colors help avoiding confusion. 3D view of object can be easily created to boost imagination.

7. Freedom from using drawings instruments: A simple CAD system needs a computer with a mouse and keyboard to draw. The draftsmen need not use bulky drawing instruments like drawing board, drafter, set square, etc.

CAD PACKAGES

Some of the commonly used packages are:

1. Auto Cad.	2. Creo.	3. 3DS-max .	4. STADD
5. PRO-Engineer	6. Free Cad.	7. Solid Works.	8. Fusion 360^{o}.

AUTO CAD AN OVERVIEW :

Auto cad created by Autodesk is the most widely used technical drawing program anywhere with more than 2.4 million registered users. AutoCAD is the fifth largest software company in the world.

DRAWING EDITOR:

A typical auto cad drawing editors are:-

1. Drawing name (Title bar)
2. Menu bar
3. Tool bar
4. Graphic area (Drawing Area)
5. UCS icon
6. Cursor
7. Screen menu.
8. Horizontal and vertical scroll bar.
9. Command prompt area. 10. Status bar.

UCS ICON- WCS is the short form from of world co-ordinate system. This system represents X, Y and Z co-ordinates. X-axis is taken positive towards right, Y-axis is taken positive towards top and Z-axis is perpendicular to the screen. UCS is short form of User co-ordinate system.
UCS is displayed two styles. They are 2D STYLE and 3D STYLE.

SETTING LIMITS OF A DRAWING.

Drawing limits represents the boundaries of the drawing LIMIT command is used to set drawing boundaries. Assume that the word, command: appear at the command prompt area. Type

LIMIT at the command prompt area using keyboard and the Enter key now the command prompt area appears as shown below.

Command: LIMITS

SETTTING UNIT OF A DRAWING:

Unit command is used to set the units of measure. Angle measurement, direction and precision. Assume that the word command, direction and precision. Assume that the command: appears at the prompt area. Type units at the command prompt area by using keyboard and press the enter key now the command prompt area appears as:
Command: UNITS

Drafting setting:

Drafting setting include the commands for initial setting of a drawing. Some of the drawing settings are Snap, Grid, Object snap, polar tracking and ortho modes.

Grid:-

GRID command is used to display a reference grid of dots with any desired spacing. This visual display provides sense of the size of the drawing entities and their relationship.

Snap:-

SNAP command is used to lock the movement of cross hair of the pointing device to the nearest grid point.

Object snap:- Object snap settings are used to pick a geometric point on an object. There are various options for object snap settings such as End point, Midpoint, Center, Quadrant, Node, Intersection, perpendicular, tangent, Etc….

Ortho:- This Command is used to move the movement to Cross Hair. i.e. Horizontal / Isometric or Vertical direction.

Following are the functional keys in Auto CAD.

1. F1- Help
2. F2- Text /Graphic window
3. F3- Object snap ON/OFF
4. F4- tablet ON/OFF
5. F5- Isoplane Top / Left /Right
6. F6- Co-ordinates ON/OFF
7. F7- Grid ON/OFF
8. F8- Ortho ON/OFF
9. F9- Snap ON/OFF
10. F10- Polar tracking ON/OFF
11. F11- Object snap tracking ON/OFF

UTILITY COMMAND

1. **NEW**

Being a new drawing .The new command allows you to create a new drawing.

2. **SAVE**

This command can be used to save a unnamed drawing.

3. **SAVE AS**

This command request a file name, save and sets the current drawing to a new file name.

4. PRINT :- This allows To Take the Print Out.

5. **HELP or [?]**

The Help or '?' command can be used to find information about auto cad.

6. **QUIT**

The quit command exits from Auto Cad.

Important Commands in Cad

Limits	Zoom.	Line	Snap	Circle	Polygon	Copy
Mirror	Erase.	Trim	Extend	Rectangle	Osnap	Hatch
Offset	Arc	Move.	Rotate.	Scale.	Ellipse	Extend
Chamfer.	Fillet	Distance	Modify	Dimhorizontal		Dimalign
Dimcontinue	Array	Dimedit	DIM OBLIQUE (FOR ISOMETRIC)			

3D COMMANDS

Region Extrude. Subtract Union Explode. Vports

LINE:- (L) Is the command used to draw line series of line segment by using Auto Cad. Command Line : Specify first point (here specify the starting point of the line). Specify next point (here specify next point of line) There are four options for drawing a line. They are :-

1. Absolute Co-ordinates option.
2. Relative Co-ordinates.
3. Polar Co-ordinates.
4. Direct distance entry option.

POLYGON:- (POL) A polygon of sides from 3 to 1024 numbers can be drawn by clicking icon polygon.

RECTANGLE:- (REC)

With this command, you can specify the rectangle parameters (length, width, rotation) and control the type of corners (fillet, chamfer, or square). Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: Specify a point or enter an option.

ARC:- (A): This command is used to draw an arc by using Auto CAD. Command: - ARC specify start point of arc or (center): (Click start point) Specify Second point of an arc or (center/end): (Click second point) Specify end point of arc :(click end point)

CIRCLE: – (C): This command is used to draw a circle by using Auto Cad. Command: - specify center point of circle for (3p/2p/Ttr (Tan tan radius)) Located the center point by using the cursor. Specify radius of the circle or (diameter): (here specify the radius of circle or type D for entering diameter.)

ELLIPSE:- (ELI) : By this Command we can create an ellipse. Defines the first axis by its two end points. 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.

HATCH:- (H): Hatching is filling of a specified and enclosed area of a drawing with a predefine or user define pattern. HATCH is the command used for hatching the objects.

TEXT:- (DT): This is used for entering the related details on a drawing by using Auto Cad. There are two types of text command, Multiline text and Single line text.

ERASE :-(E): Erase command is used to remove the selected objects from a drawing.

COPY:-(CO): Copy command is used to create one or more copies of selected objects.

MIRROR:- (MI): Mirror command is used to create mirror image of an object or objects.

OFFSET:-(O): Offset command is used to create a new object ,which has shape of the object to be selected.

ARRAY:- (AR) : Array command is used to create multiple copies of an object or objects in a rectangular or circular pattern.

MOVE :-(M): Move command is used to move an object or objects from a position to any required position,without changing their size.

ROTATE:- (RO): Rotate command is used to rotate an object through any specified angle about its base.

SCALE :(SC): Scale command is used to change the size of an object according to a scale factor. If the scale factor is greater than one, the object is enlarged and if it is less than one, the object is in reduced size.

TRIM:-(TR): Trim command is used to cut and remove any drawn line at any required point.

EXTEND :-(EX): Extend command is used to extend an object up to the specified limit.

CHAMFER :-(CHA): Chamfer command is used to join two non-parallel lines by drawing a beveled line at a distance.

FILLET :(F): Fillet command is used to join two non-parallel lines,a line and an arc, circles, polylines with an arc of specified radius.

EXPLODE:-(X): Explode command is used to split compound objects like polygon, 3D Objects .etc.

EXTRUDE :- It allows to Extrude the any Regions Or Closed Object to 3D Solid Object.

SUBTRACT :- It allows to Subtract the any Regions Or 3D Solid Object from One to another.

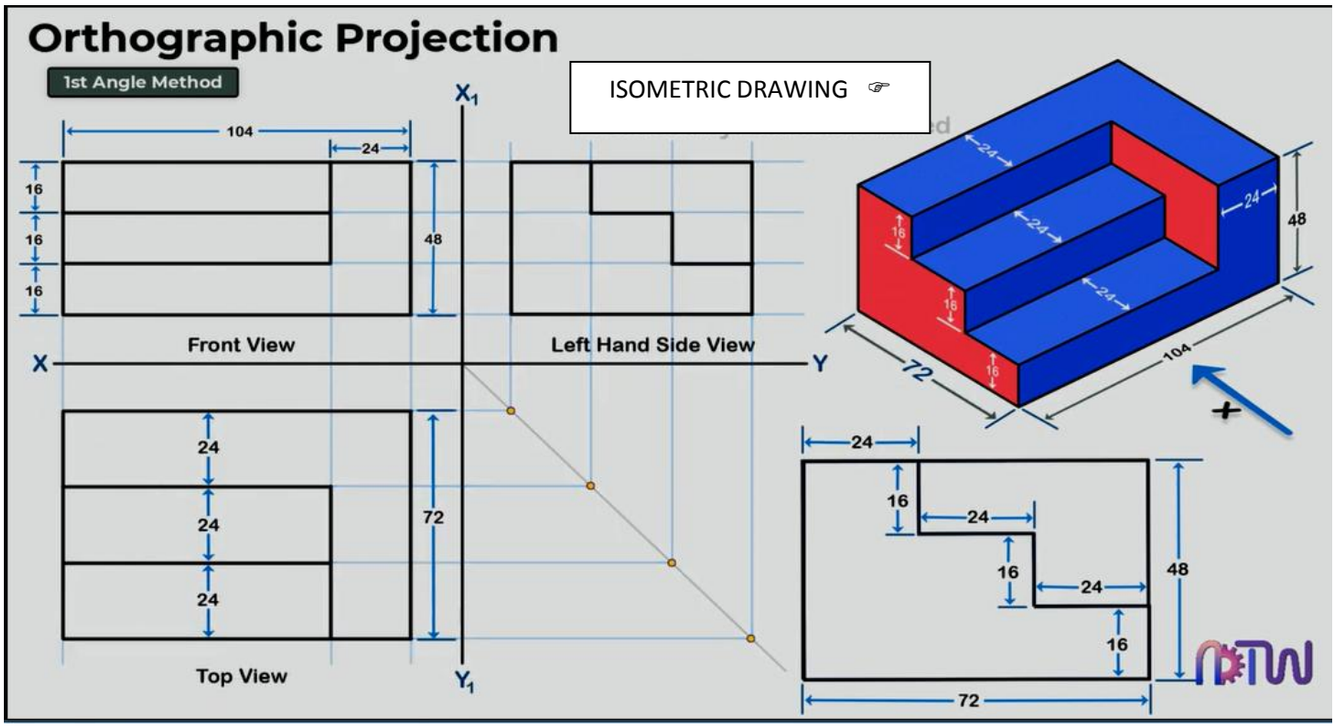
UNION:- It allows merge the several regions and solids.

VPORTS :-It creates the several viewports ,to view from different Directions, Like ,Front, Top, SW – ISOMETRIC ETC.

SECTION – I
[NOTE:-1. All Should Be Executed During Practical.
(Both Problem And Solution) 5 +5=10 DWGS]

DRAW THE ORTHOGRAPHIC VIEWS OF GIVEN ISOMETRIC DRAWING

DWG-1 [PROBLEM & SOLUTION]



Procedure to Execute Above Drawing

AIM: To draw the Orthographic Views Of Isometric Drawing With and Draw The Given Isometric Drawing Also. .

PROCEDURE:

1) Set The Drawing Area As per drawing area required. By The Command : ZOOM-----> ALL

2] Draw the **X** , **Y** axis, Front View, Top View , Side by line command and modify it by the command: Offset, trim, move, copy, Etc.,

3] Switch over to Isomeric Drawing By the Command SNAP----> STYLE----> ISOMETRIC

4] Then Draw the isometric dwg using, line, trim, ortho any other required command. 6)

5] After Completing Dwg Show the Title by **DTEXT -COMMAND**, Show The Dimensions By Dimensioning Commands like Dim horizontal, Dim aligned etc

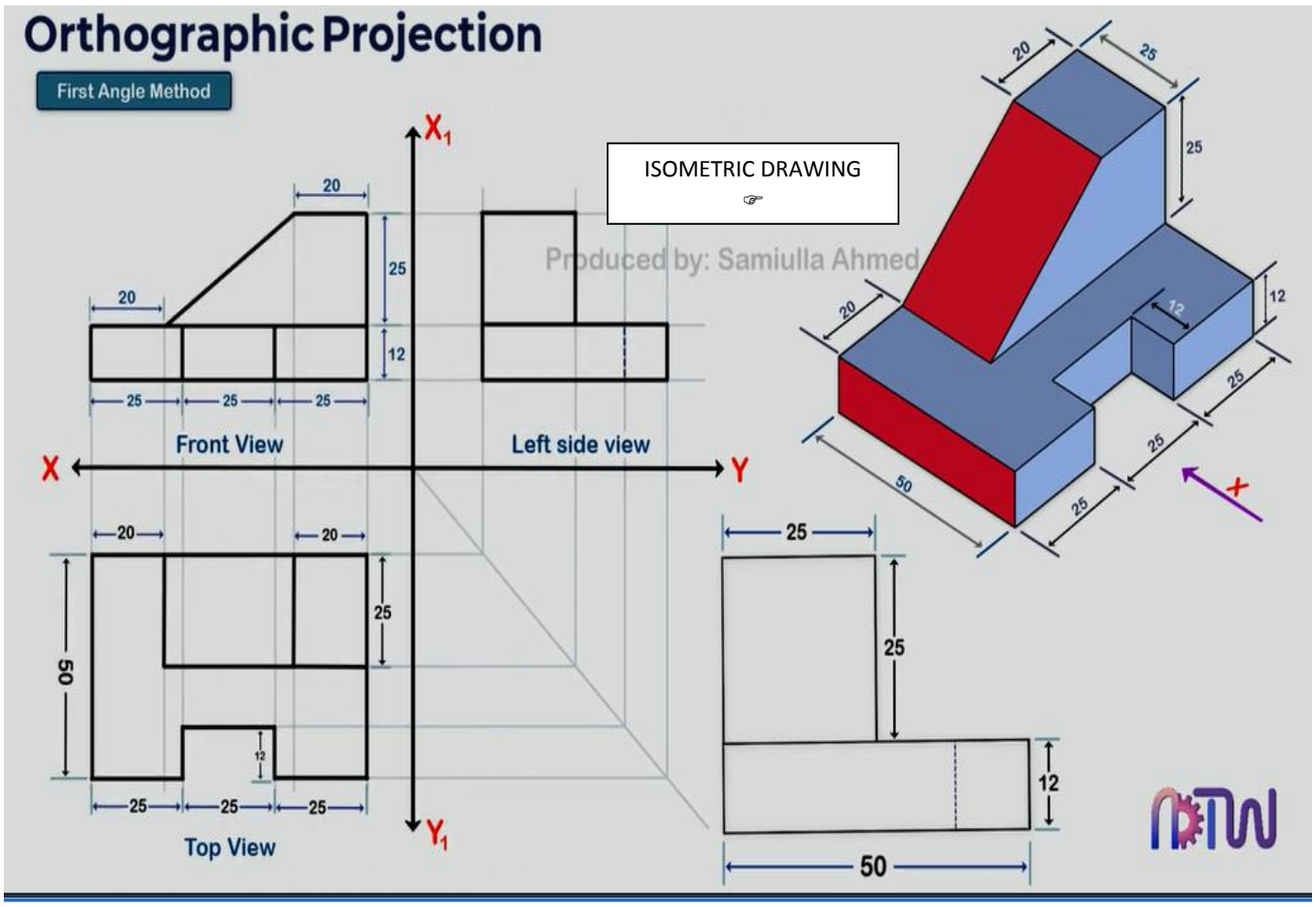
6] Save the file by menu →file →Save or control 'S'

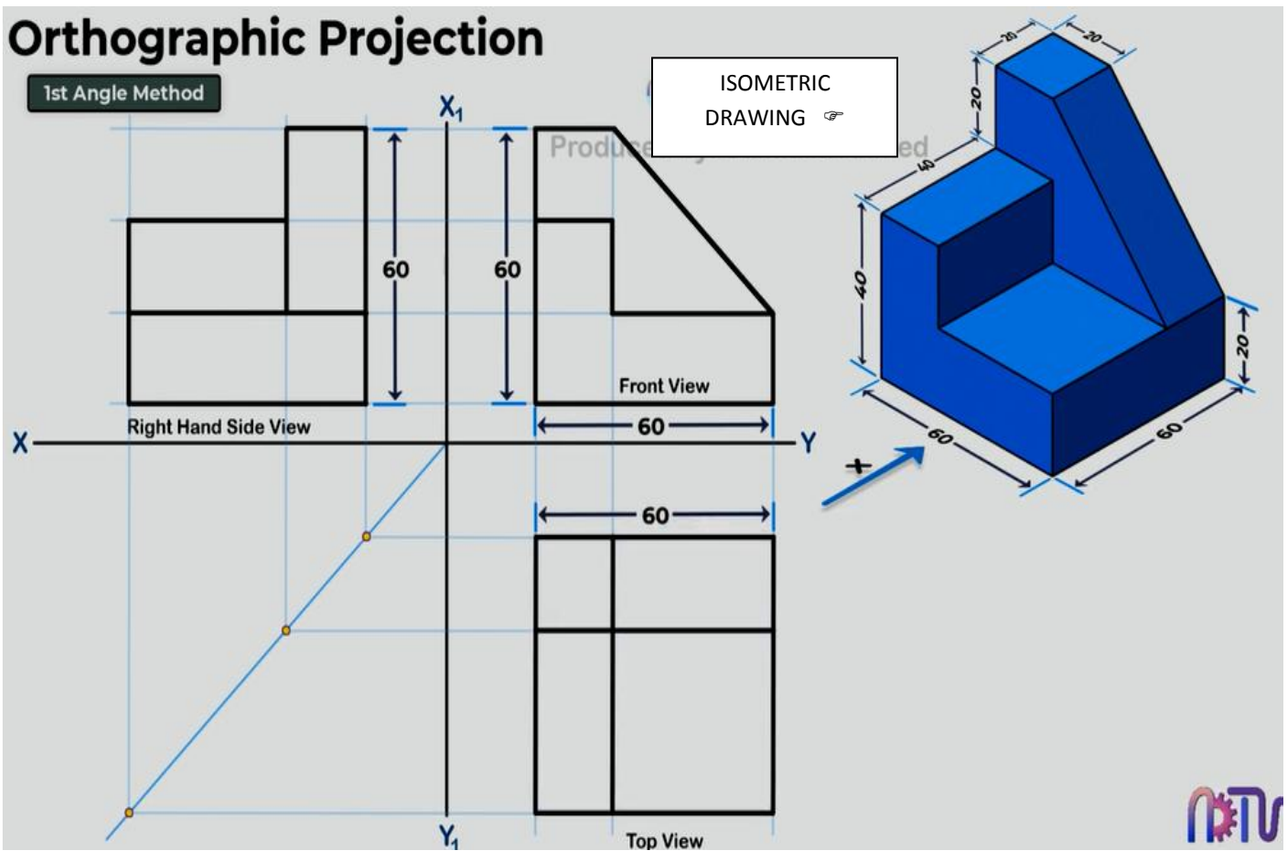
07] Take a print out by menu →file → plot or control 'P'.

Result:

The drawing is Executed .

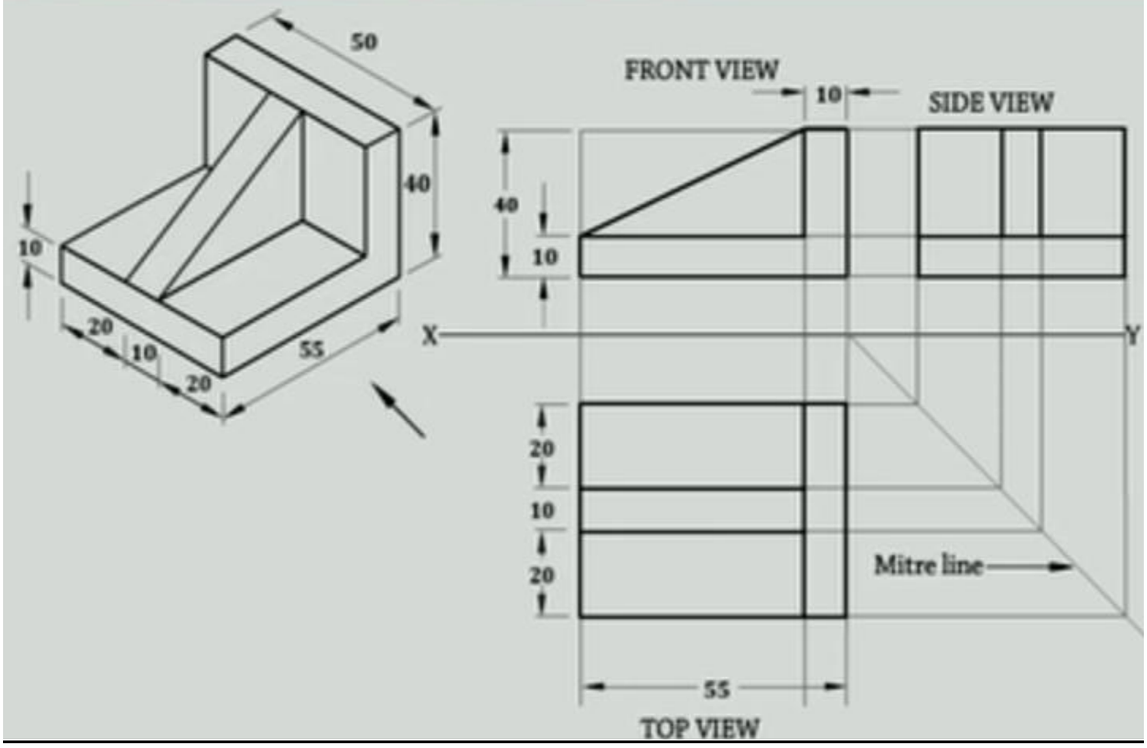
DWG-2 [PROBLEM & SOLUTION] [NOTE:- Write the Procedure , your self only]



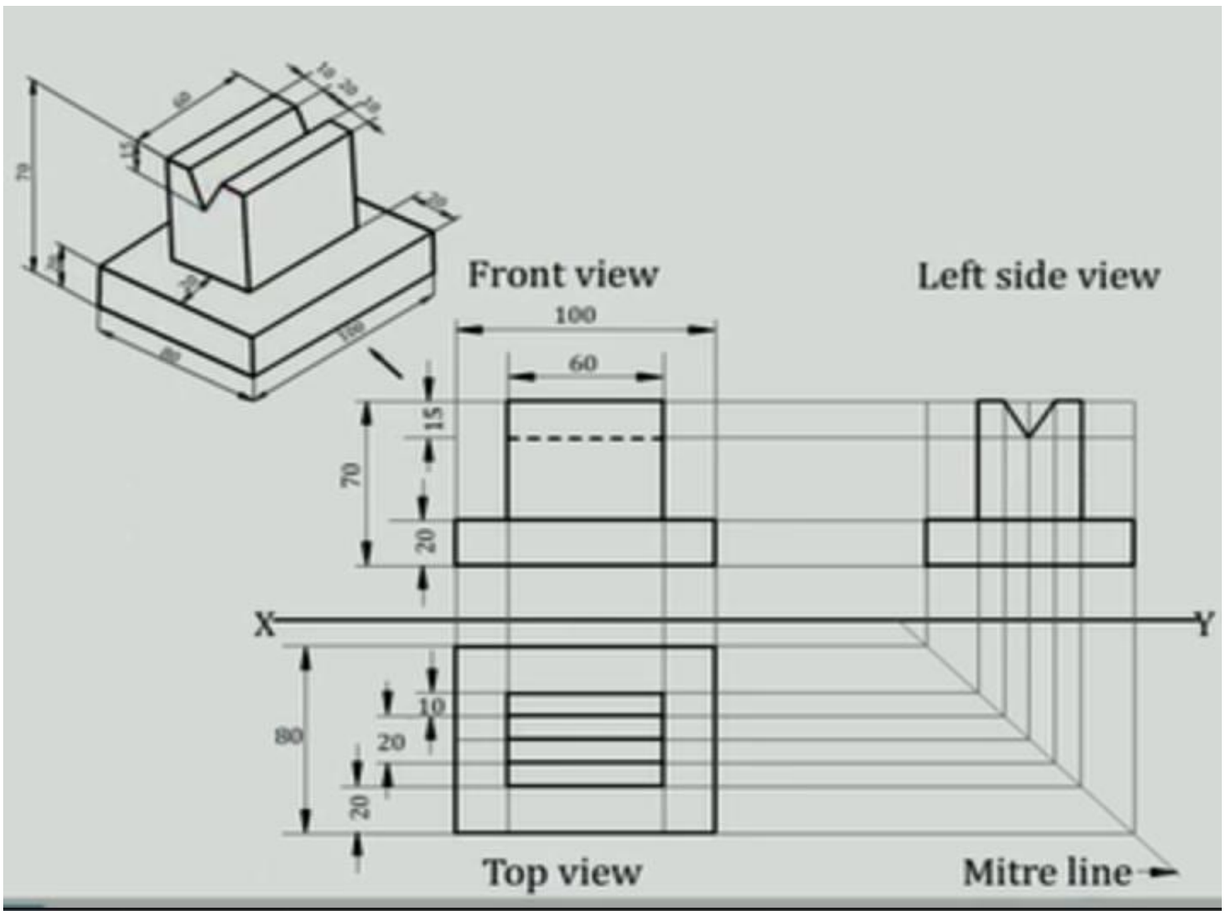

Orthographic Projection
1st Angle Method
ISOMETRIC
DRAWING
X1
60
60
Front View
Right Hand Side View
X
Y
60
60
Y1
Top View
20
20
20
40
20
60
60
+

DWG-4 [PROBLEM & SOLUTION] [NOTE:- Write the Procedure , Your self only]

In the top view the web portion is seen as rectangle of size 55mm × 10mm and In the side view, it will be seen as rectangle of size 30mm × 10mm.



DWG-5 [PROBLEM & SOLUTION] [NOTE:- Write the Procedure , Your self only]



Introduction to 3D.

Most of the drawing consists of two dimensional views of the object that ate essentially 3 dimensional. While creating these drawings different views are created independently and hence the possibilities of error ambiguity are high. Visualization of the object is also difficult as the reader has to imagine the object by mentally arranging the different views. In auto cad 3 dimensional objects are directly created and visualized from different angles. Further the view can be enhanced by shading or rendering.

Any 2 dimensional objects can be converted into 3 dimensional objects by giving height (thickness). UCS icon shows the direction of X, Y, Z axis in the auto-cad graphic screen. By default X direction is towards right. Y direction is upward and Z direction is towards the observer. Therefore whatever drawn in this view is the "PLAN" view of the object. So even if thickness is perpendicular to the view. It is possible to view the object from different direction.

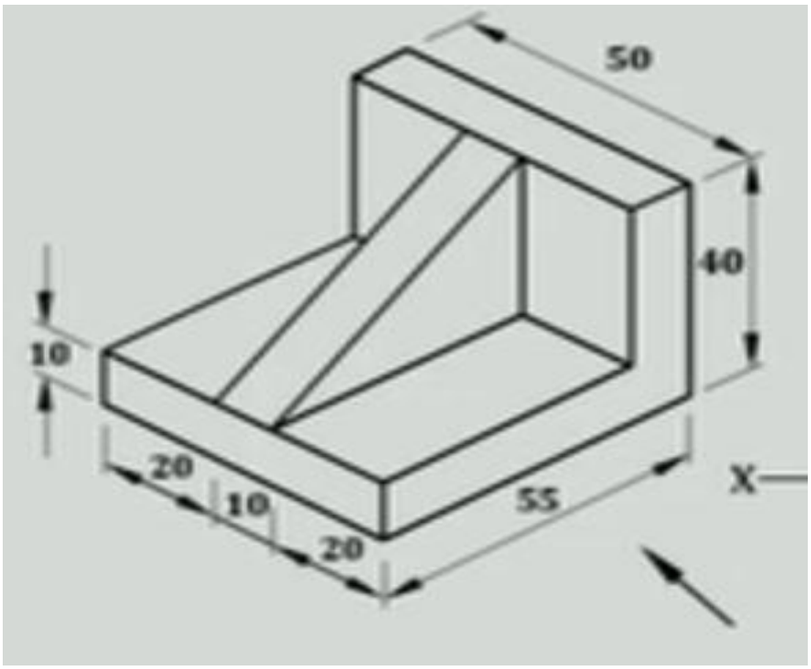
Following are the few common commands used in 3D drawings.

1) Command: 3d.
   Opens predefined objects and their 3d views. Eg: Box, Sphere, Cone, Dome ect.
2) VPORTS :- Generates the Different 3d views of the object with several window.
3) Shade: Provides shading so as to make the proper visualization of the 3d object.
4) Region: To make the view of an object into a single entity.
5) Extrude: To assign thickness or height to a 2d object. This can be also be done by giving a path along, which extrusion is desired.
6) Solid editing: This is used to edit the 3D face of an object.
7) Subtract: Creates a new region by subtracting one overlapping region from another.

SECTION –I I

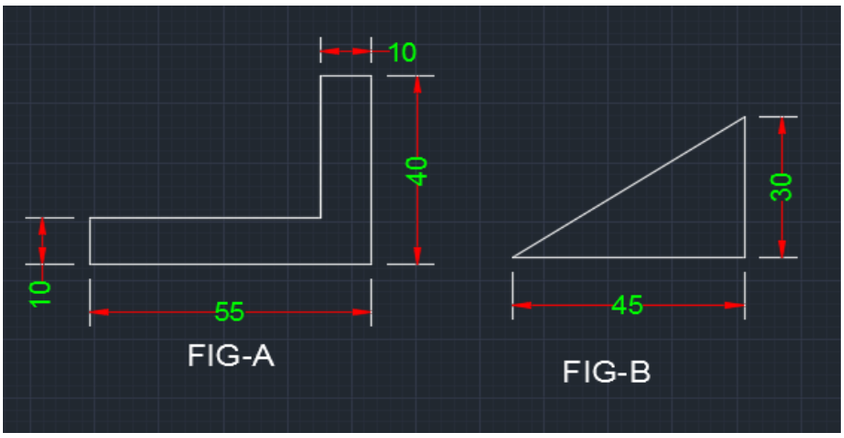
(3D SOLID MODELLING)

3DS-1. Create the Angle Block As Per Fig Shown Below.



Procedure :-

1. Draw the 2D Profile as Below using the **Command**:-LINE AND OTHERS .



2. Convert This Profile Into Closed Region, By The **Command :-** REGION

3. Extrude the Fig 1. By The **Command** :**EXTRUDE** & **Give the Height as 50**.

4. Extrude the Fig 1. By **The Command** :**EXTRUDE** & **Give the Height as 10.**

5. To See The 3D Effects **Give The Command** :-**VPORTS .**

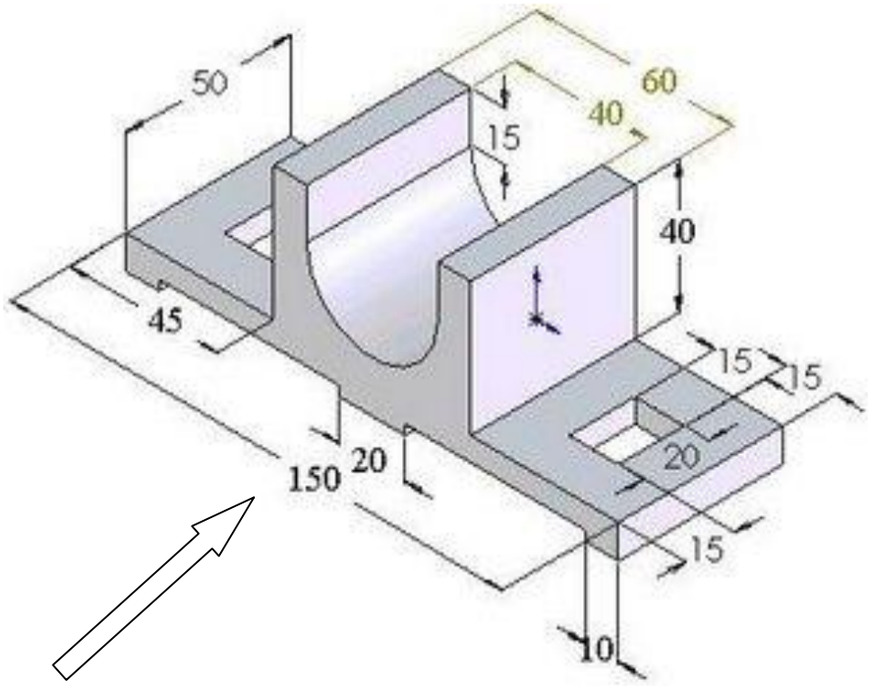
6. Rotate the 3d objects to the Proper Angle By **The Command :ROTATE3D & select the axis by edge/2point methods.**

7. For Shading Effect **Give The Command** :-**shademode------> choice required shade mode.**

8. Move **Part B** on **Part A** in Proper Position then Merge those by **The Command Union.**

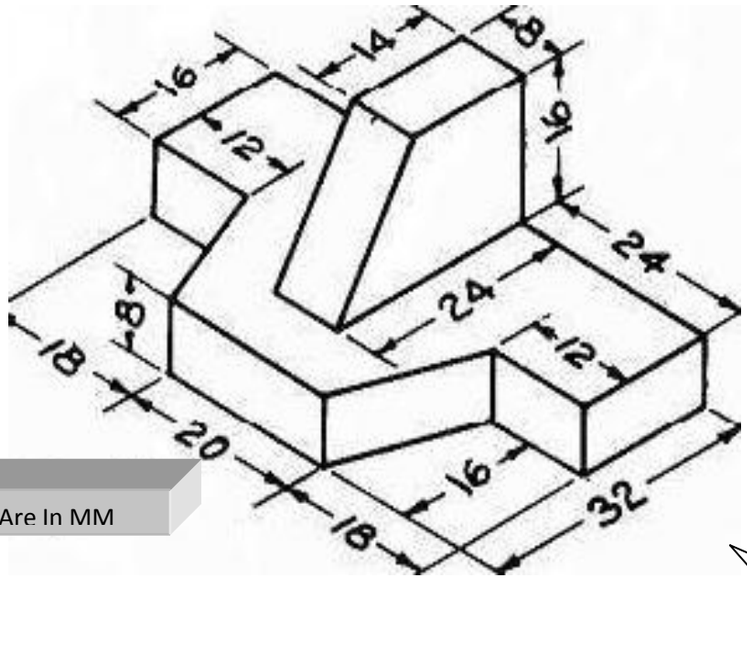
3DS-2. Create the 3D DRAWING AS PER Fig Shown Below.

[NOTE:- Write the Procedure , Your self only]



3DS-3. Create the 3D DRAWING AS PER Fig Shown Below.

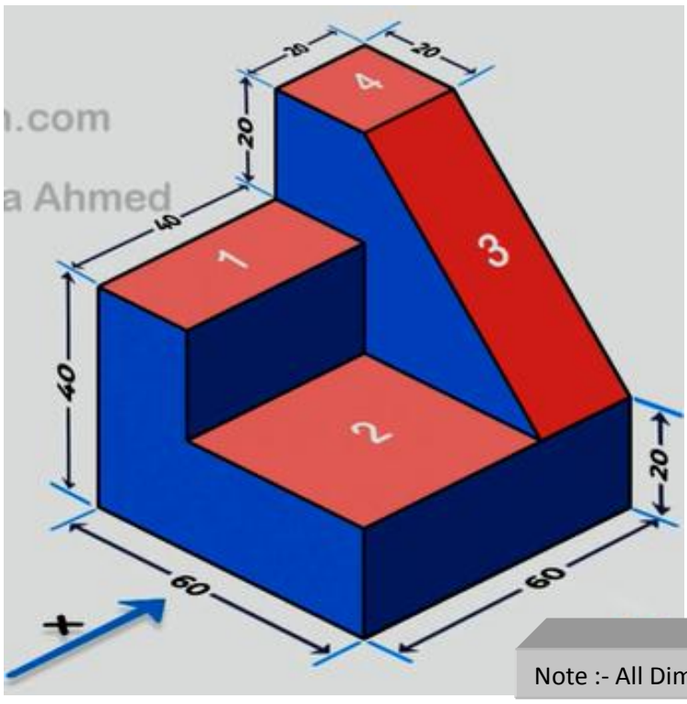
[NOTE:- Write the Procedure , Your self only]



Note: --All Dimensions Are In MM

3DS-4. Create the 3D DRAWING AS PER Fig Shown Below.

[NOTE:- Write the Procedure , Your self only]



Note :- All Dimensions Are In MM

3DS-5 Create the 3D DRAWING AS PER Fig Shown Below.

[NOTE:- Write the Procedure , Your self only]