Course File

Name of the Lab: ENGINEERING GRAPHICS LAB

Name of the Faculty: N.Harini

Year/Sem : I-I &II

Academic Year: 2019-2020

Department of Mechanical Engineering

MALLA REDDY ENGINEERING COLLEGE (Autonomous)

(Approved by AICTE & Affiliated to JNTUH)

Maisammaguda, Dhulapally (Post via Kompally), Secunderabad-500 100

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Faculty Name & Sign Verified By HOD PRINCIPAL

INSTITUTE VISION

To establish a reputable professional education centre, to impart high quality trend setting technologies in an ambience of humanity, wisdom, intellect and innovation to nurture the students to become competent and committed professionals with disciplined ethical values.

INSTITUTE MISSION

Commitment to progress in mining new knowledge by adopting cutting-edge technologies to promote academic growth by offering state-of-the-art undergraduate and postgraduate programmes based on well-versed perceptions of global areas of specialization to serve the nation with advanced technical knowledge.

DEPARTMENT VISION

To provide world class platform for Education, Research, Knowledge and Technical skill in Mechanical Engineering and to create leaders with passion for innovation to ensure environment friendly development needs of the society.

DEPARTMENT MISSION

Create innovative learning atmosphere with superior and environment friendly infrastructure for better understanding of the technical knowledge in practical situations, so as to make them effective ethical and global leaders.

PROGRAMME EDUCATIONAL OBJECTIVES (PEOs)

PEO1: To develop the ability among students to synthesize data and technical concepts for applications related to product design and development in industry that meet the global needs.

PEO2: To provide students with sound foundation in the mathematical, scientific and engineering fundamentals necessary to formulate, solve and analyse engineering problems and to prepare them to work as part of teams on multi disciplinary projects.

PEO3: To promote student awareness of the lifelong learning and to create them with professional ethics and code of practice.

PROGRAMME OUTCOMES (POs) (for the regulations MR15-MR18)

PO 1	Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO 2	Problem analysis: Identify, formulate, review research literature and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.
PO 3	Design/development of solutions: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO 4	Conduct investigations of complex problems: Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO 5	Modern tool usage: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO 6	The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
PO 7	Environment and sustainability: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
PO 8	Ethics: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO 9	Individual and team work: Function effectively as an individual and as a member or leader in diverse teams, and in multidisciplinary settings.
PO 10	Communication: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO 11	Project management and finance: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
PO 12	Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.
PSO1	Understand the problem and apply design and analysis tools to find solution in the domains of Structural, thermal and Fluid Mechanics.
PSO2	Engage professionally in industries or as an entrepreneur by applying Manufacturing concepts.
PSO3	Systemize the Engineering and manufacturing practices using TQM concepts and Optimization techniques

Syllabus

2018-19 Onwards (MR-18)	MALLA REDDY ENGINEERING COLLEGE (Autonomous)	B.Tech. II Semester		
Code: 80302	**ENGINEERING GRAPHICS LAB** (Common for ME,CIVILEEE,ECE, CSE and IT)	**L**	**T**	**P**
Credits: 1		**-**	**-**	**2**

Prerequisites: NIL

Course Objectives:

To develop student's skill in Computer graphics for communicating the concepts and ideas in engineering products by using drafting software.

List of Exercises

Any 12 exercises out of fourteen should be done by using drafting software

1. Drawing of basic drawing elements and Regular polygons.
2. Drafting projections of lines- parallel, perpendicular, inclined to one reference plane.
3. Drafting projections of lines - inclined to both reference planes.
4. Drafting projections of lines inclined to both reference planes - obtaining true length.
5. Drafting Projections of planes – Surface inclined to one reference plane.
6. Drafting of regular solids - cube, prism, pyramid, cylinder and cone.
7. Drafting projection of solids inclined to one plane.
8. Drafting projection of section of solids cutting plane inclined to one plane.
9. Drafting development of surface of regular solids - prism
10. Drafting development of surface of regular solids - cylinder, cone
11. Drafting Isometric Projection – Isometric Views- Plane Figures
12. Drafting Isometric Projection – Isometric Views- Simple Solids.
13. Conversion of Isometric Views to Orthographic Views.
14. Conversion of Orthographic Views to Isometric Views.

Course Out comes:

At the end of the course, students will be able to

1. Draft basic drawings elements.
2. Draw the projection of points, lines and planes on Cartesian coordinates using drafting software.
3. Draw the projection solids on Cartesian coordinates using drafting software.
4. Develop surfaces of regular solids, sectional solids and solids inclined to one axis using drafting software.
5. Convert and develop the isometric views on to orthographic projections using drafting software.

CO- PO Mapping (3/2/1 indicates strength of correlation) 3-Strong, 2-Medium, 1-Weak												
COs	**Programme Outcomes(POs)**											
	PO1	**PO2**	**PO3**	**PO4**	**PO5**	**PO6**	**PO7**	**PO8**	**PO9**	**PO10**	**PO11**	**PO12**
CO1	3		1		3				2	3		2
CO2	3		1		3				2	3		2
CO3	3		1		3				2	3		2
CO4	3		1		3				2	3		2
CO5	3		1		3				2	3		2

TIME TABLE

MALLA REDDY ENGINEERING COLLEGE(Autonomous)

Department of Mechanical Engineering

I B.Tech. II Semester

Name of the Lab: Engineering Graphics

Academic Year: 2019-20

	11:05-12:45	**02.20-4:00**
Monday		**ECE-B**
		Mr.T.Naresh
		Mrs.B.Charitha(s)
Tuesday	**CSE-D**	**CSE-A**
	Mr.M.Sandeep	**Mrs. P. Shanti Priya**
	Mrs.R.V.S.Madhuri(s)	Mr.M.Sandeep(s)
Wednesday	**ECE-A**	**EEE-A**
	Mr.T.Naresh	**Ms.N.Harini**
	Mrs.P.Santhi Priya(s)	MrsS.Bhanu Teja(s)
Thursday		**CSE-C**
		Ms.N.Harini
		Mrs.P.Santhi Priya(s)
Friday	**CSE-B**	**IT**
	Mr.E.Rajesh	**Mr.M.Sandeep**
	Mr.M.Sandeep(s)	Mr.V.Indrakanth(s)
Saturday	**1.EEE-B (First 2 hours)2.ECE-C(Last 2 hours)**	
	1. Mr.E.Rajesh, Mrs.S.Bhanu Teja(s)	
	2.Mrs.A.Aruna Jyothi, Mr.P.Praveen Babu(S)	

Lesson plan

Department: ME | Academic Year: 2019-20

Year of Branch: I | Semester: I &II

Subject Description: Engineering Graphics Lab | Subject Code: 80302

Name of Faculty Member: Ms.N.Harini | No of lectures per Week: 2

Number of Total Number of Periods: 32 | Number of Working Weeks: 16

.No.	List of Experiments	Number of hours
1	Drawing of basic drawing elements and Regular polygons	2
2	Drafting projections of lines- parallel, perpendicular, inclined to one reference plane.	2
3	Drafting projections of lines - inclined to both reference planes.	2
4	Drafting projections of lines inclined to both reference planes - obtaining true length.	2
5	Drafting Projections of planes – Surface inclined to one reference plane.	2
6	Drafting of regular solids - cube, prism, pyramid, cylinder and cone.	2
7	Drafting projection of solids inclined to one plane.	2
8	Drafting projection of section of solids cutting plane inclined to one plane.	2
9	Drafting development of surface of regular solids - prism	2
10	Drafting development of surface of regular solids - cylinder, cone	2
11	Drafting Isometric Projection – Isometric Views- Simple Solids.	2
12	Conversion of Isometric Views to Orthographic Views.	2
13	Conversion of Orthographic Views to Isometric Views.	2
14	Drawing of basic drawing elements and Regular polygons.	2

INTRODUCTION TO GRAPHICS PACKAGE

The engineering drawing has been and is an integral part of industry and it is a link between engineering design and manufacturing. Information is quickly communicated in the form of drawings prepared according to prescribed drafting standards.

What is Computer Aided Design?

The use of interactive graphics programs to develop 2D and 3D models, assemblies, part lists, and dimensional drawings of various components, structures or objects with the help of a computer is called Computer-Aided Design(CAD). The data generated by a CAD system can be directly utilized by a CAM (Computer Aided Manufacturing) system. Thus CAD and CAM are interrelated to each other.

Advantages of CAD:

1. Reduction of drafting labor.
2. Direct cost savings.
3. High accuracy (up to one millionth of a unit)
4. Improvement in the general flow of information through the company.
5. Evaluation of alternative designs.
6. Use of common parts in multiple products.

Before we design and manufacture any component, the object must be presented in the form of drawings along with different views and dimensions. Further, the bill of materials etc. must also be supplied. With the advent of computers and relevant software packages, the drafting drifted from manual to computer aided drafting. The networking between the shop floor sections and design sections makes it easier to share the data and information. The modifications (in the drawings, materials or process of operation) if required, can be discussed on line and can be implemented without any time lag. An attempt is made here to introduce the graphics package in the following topics.

Starting Up AutoCAD

Select AUTOCAD from *Start* menu or picking the AutoCAD icon with a double click will open the AutoCAD. If you choose to start AUTOCAD from *start* menu, select Start/ Programs/ AutoCAD.

AUTOCAD 2015 SCREEN LAYOUT

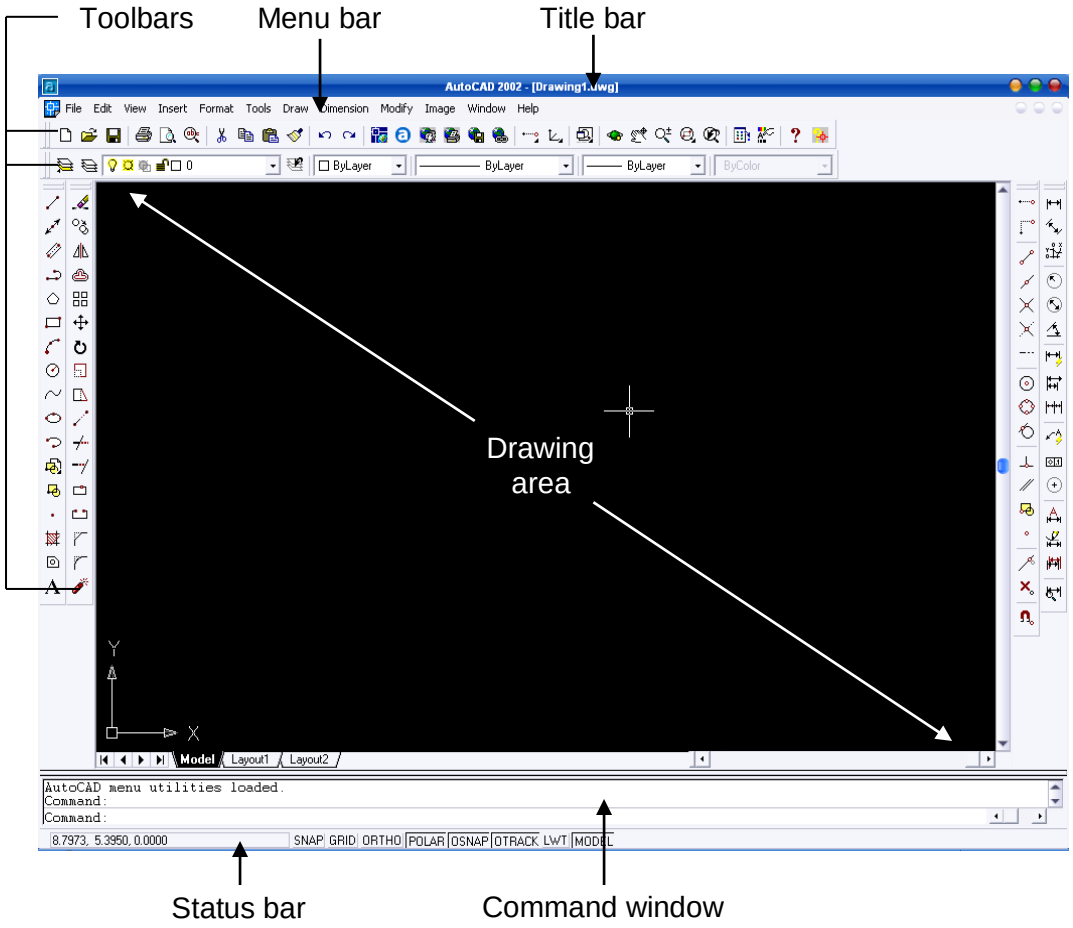


Figure 1: AutoCAD screen layout

AutoCAD screen layout is divided into six major parts:

- Title bar
- Menu bar
- Toolbars
- Document window or drawing area
- Command window
- Status bar

'Limits'

The Drawing Limits command is used to define the extent of the drawing and the grid display. The command is also used to turn Limits mode on or off. A limit is turned off by default which means that there is no restriction as to where points can be picked and objects drawn. When Limits is on, AutoCAD will not allow points to be picked or coordinates

entered at the command line which falls outside of the specified drawing limits. Usually the scale of 1:1 (full scale) is used to represent the actual drawing size.

Example:

Command: limits <enter>

Reset model space limits.

ON/OFF/ lower left corner <0.0000,0.0000> : <enter>

Upper right corner <12.0000,9.0000> : 420,300<enter>

‘Snap’

Snap command allows picking points which lie on a regular grid. When Snap mode is turned, the crosshairs will jump from one grid point to another as you move across the screen. This makes it very easy to draw objects which have a regular shape. The Snap command is used to set the snap spacing and to toggle Snap mode. Snap command can be used to turn Snap mode on and off, but it is much more efficient to use the **F9** function key on the keyboard or to double-click "SNAP" on the status bar. Snap style can be set to either *Isometric* or *Standard* (the default) using the "Style" option. The Standard style is used for almost all drawing situations including detail drawings in *Orthographic Projection*. The Isometric style is specifically to aid the creation of drawings in *Isometric Projection*.

Command: snap <enter>
Snap spacing or ON/OFF/Aspect/Rotate/Style: 10 <enter>

Snap spacing – default value for snap spacing in x and y direction are 1 unit. However it can be changed to any number by setting the snap spacing.

Grid

The Grid command can be used to turn Grid mode on or off and to set the grid spacing (in drawing units). When Grid mode is on, AutoCAD displays a regular pattern of dots on the screen as a visual aid; it is equivalent to having a sheet of graph paper behind your drawing on a drawing board. If the size of a drawing is A3 size, the suitable grid spacing is about 10 units.

Example:

The grid command will produce;

Command: Grid <enter>

Grid spacing <x> or ON/OFF/Aspect/ <0.000>: 10 <enter>

Units

Units command can be used to control the accuracy of drawing. There are several formats that available for determination of spacing and measurement style.

Command: **Units** <enter>
Measurement unit system

1. Scientific 1.55E + 01
2. Decimal 15.50
3. Engineering 1' – 3.50"
4. Architectural 1 – ¾"
5. Fractional 15 ½

Enter selection :< choose the coordinate format (1-5) and press enter>
Numbers of digits to right of decimal point (0-8) <4> : <value><enter>

After define the unit spacing unit, determine the angle measurement system.

Angle measurement system

1. Decimal degree 45.0000
2. Degree/minute/second 45d0'00"
3. Grads 50.0000g
4. Radian 0.7854r
5. Surveyor's unit N45d0'0"E

Enter selection: <Value> <Enter>

Number of fractional places for displaying angle <8> : <Enter>

Exercises of using limits, snap and grid in a drawing

Command: limits <enter>
ON/OFF/ lower left corner <0.0000, 0.0000>: 0,0 <enter>
Limits at upper left corner is setting to A4 size paper
Command: grid <enter>
Grid spacing: 10 <enter>
Command: snap <enter>
Snap spacing: 10 <enter>
Command: Zoom <enter>
All/Extend/……/Scale: All <enter>
'Zoom All' command will change the drawing limit to A4 size paper.

THE FUNCTION KEYS

Many of the modes in AutoCAD can be controlled quickly using the keyboard function keys. In most cases this is quicker than using a pull-down or the command line.

FI (dos) / F2 (windows) – Graphic screen or text screen
This key clears half of the screen and displays the HELP menu. Pressing F2 in the middle of a drawing will remove the drawing from the screen and display a written list of the command that were used to generate the drawing.

F6 – Coordinate mode
The coordinate display can be seen on the status of the line. Key F6 will switch the coordinated display on or off.

F7 – Grid mode
The F7 key is used to toggle grid mode on and off. When grid mode is on a grid of dots is shown on the screen as a drawing aid.

F8 – Ortho mode
The F8 key on the keyboard can be used to toggle Ortho (orthogonal) mode on and off. When Ortho mode is on AutoCAD will only allow to draw either vertical or horizontal lines.

F9 – Snap mode
The **F9** key can be used to toggle Snap mode on and off. Snap makes the *crosshairs* jump to points on a defined grid.

COORDINATE SYSTEMS

(1) Absolute Coordinates

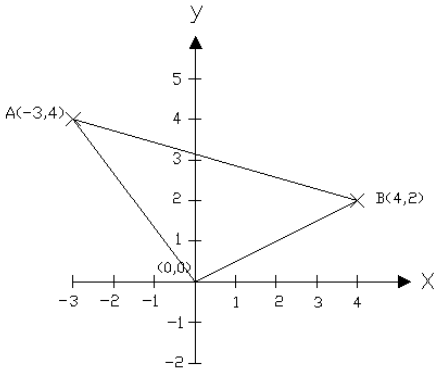
The Absolute coordinate method is based on specifying the location of a point by providing its distances from two intersecting perpendicular axes in a 2D plane or from three intersecting perpendicular planes for 3D space. Points are located by absolute coordinates in relation to the origin. In AutoCAD, by default the origin (0,0) is located at the lower left corner of the drawing.

(2) Relative Coordinates

In the relative coordinate method of construction each of the coordinates are entered relative to the last set – i.e. including the distance from the last set. To differentiate between absolute and relative entry, @ is entered before the figures of the coordinate point relative to the last point. Example: @4,2

(i) Absolute Coordinates

Command: Line<Enter>
Specify first point: 0,0<Enter>
Specify next point or [Undo]: -3,4<Enter> (Point A)
Specify next point or [Undo]: 4,2<Enter> (Point B)
Specify next point or [Close/Undo]: 0,0

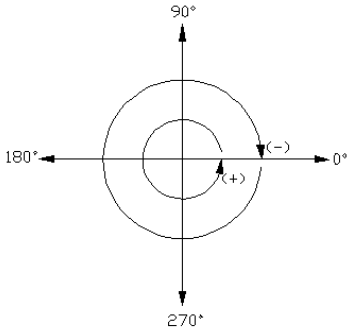


(ii) Relative Coordinates

Command: Line<Enter>
Specify first point: 0,0 <Enter>
Specify next point or [Undo]: @-3,4<Enter> (Point A)
Specify next point or [Undo]: @7,-2<Enter> (Point B)
Specify next point or [Close/Undo]: 0,0

(3) Polar Coordinates

Polar coordinates use a distance and an angle to describe the position of a point. The angle 0 is always pointing to the right on your screen (east).



EXERCISE-1

AIM: Draw basic drawing elements using AutoCAD

PROCEDURE:

DRAW COMMAND

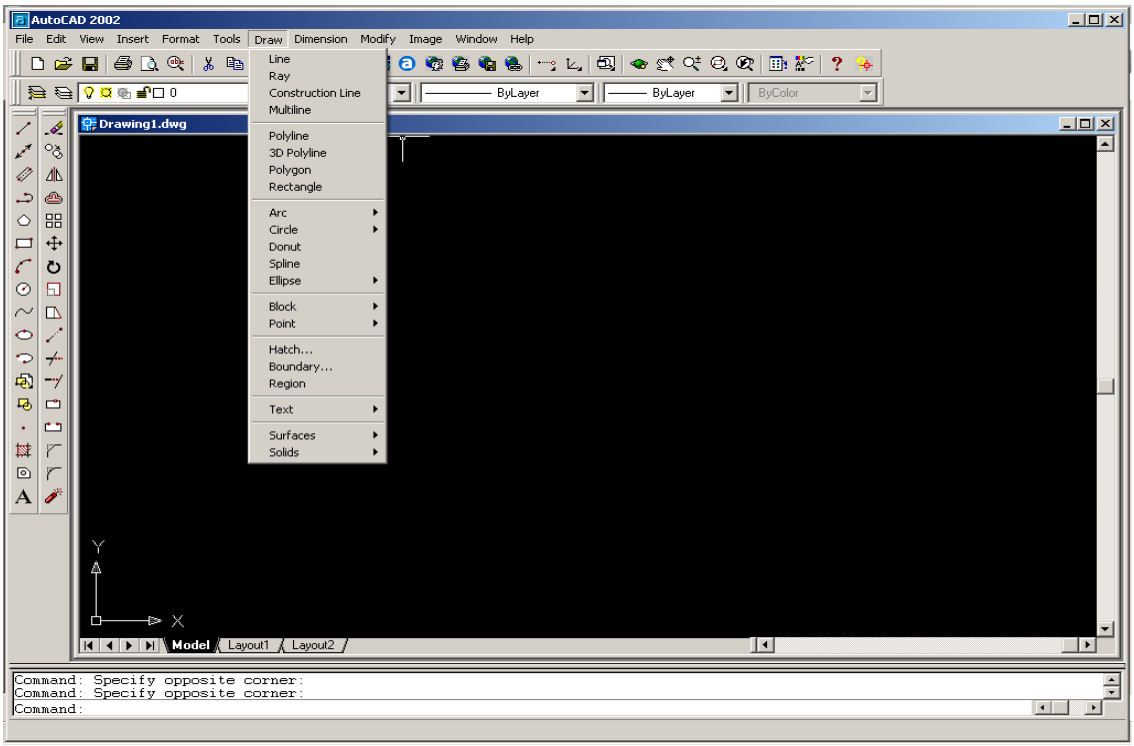


Figure 2: Screen layout for draw command function

1. 'LINE'

Lines are probably the most simple of AutoCAD objects. Using the Line command, a line can be drawn between any two points picked within the drawing area. When you *pick* the first point and move the *cross-hairs* to the location of the second point you will see a *rubber band line* which shows where the line will be drawn when the second point is picked. Line *objects* have two ends (the first point and the last point). You can continue picking points and AutoCAD will draw a straight line between each picked point and the previous point. Each line segment drawn is a separate *object* and can be moved or erased as required.

U – Line Undo – An alternative to erase. UNDO will remove the last line drawn or command that was executed, and allow continuing from the previous line.
C – Line closure – if a sequence of line will form a closed polygon, the last line can be drawn without providing endpoints or coordinates for the termination point, by typing C at the "To point:" will close the polygon precisely.

2. CIRCLE

The Circle command is used to draw circles. There are a number of ways to define the circle:

- **Center and Radius or Diameter –** The default method of drawing a circle is to enter the center point and radius or diameter.
- **Two point Circles –** Drawing a two-point circle can be useful if the diameter is known, but the center point is not.
- **Three – Point Circles –** If certain points of circle are known by entering any three points.
- **Tangent, Tangent, and Radius –** A circle may be drawn tangent to existing features by selecting two features that the circle will be tangent to, and then specifying a radius.

3. ARC

The Arc command allows you to draw an arc of a circle. There are numerous ways to define an arc; the default method uses three pick points, a start point, a second point and an end point.

- '3-point'.
- 'Start, Cen, End, or 'Cen, Start, End' (Start, Center, End)
- 'Start, Cen, Angle' or 'Cen, Start, Angle' (Start, Center, Angle)
- 'Start, Cen, Length' or 'Cen, Start, Length' (Start, Center, Length)
- 'Start, End, Angle' (Start, End, Angle)
- 'Start, End, Radius' (Start, End, Radius)
- 'Start, End, Dir' (Start, End, Direction)

4. POINT

This particular command allows for creating point with different style and size. Determination of style and size are provided on pull-down menu 'Settings → Point Style….'

5. POLYLINE - PL

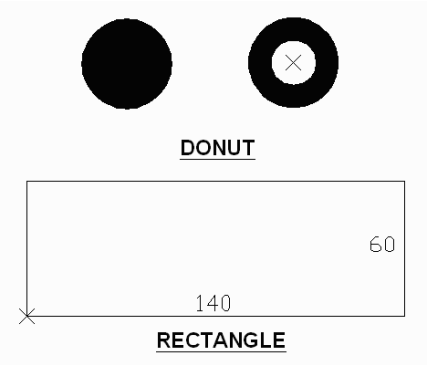
The polyline refers to a single object with multiple connected straight-line and/or arc segments.

6. DONUT

This command draws a solid donut shape. AutoCAD asks you to define the inside diameter i.e. the diameter of the hole and then the outside diameter of the donut.

7. RECTANGLE

The Rectangle command is used to draw a rectangle whose sides are vertical and horizontal. The position and size of the rectangle are defined by picking two diagonal corners.



8. 'POLYGON'

AutoCAD allows you to draw 2D polygons with POLYGON command. The number of sides can be anywhere from 3 to 1024. There are several options to draw the polygon:-

(i) **Edge Option (E)** – draws a polygon by specifying the endpoints of the first edge.

(ii) **Inscribed Option (I)** – draws the polygon of equal length for all sides inscribe inside an imaginary circle having the same diameter as the distance across apposite polygon corners (for an even number of sides).

(iii) **Circumscribed Option (C)** – draws a polygon circumscribed around the outside of an imaginary circle having the same diameter as the distance across the opposite polygon sides (for an even number of sides).

9. ELLIPSE

The Ellipse command gives you a number of different creation options:-

1) Axis, Eccentricity – The default option is to pick the two end points of an axis and then a third point to define the eccentricity of the ellipse.
2) Center, Axis, Axis – draw an ellipse by defining the center point and axis endpoints.

AutoCAD also allows drawing an ellipse by specifying a rotation angle after defining two endpoints of one of the two axes. The rotation defines the major – axis – to – minor – axis ratio of the ellipse by rotating a circle about the first axis. The greater the rotation angle value, the greater the ratio of major to minor axes. AutoCAD draws a circle if you set the rotation angle to 0 degrees.

10. HATCH

Hatch patterns are repetitive patterns of lines, dots, and other symbols used to represent a surface or specific material. Hatch patterns are used on sections and details, although to shows the plan views and elevations.

OBJECT SNAP

Object Snap (OSNAP)

OSNAP tools include CENter, ENDpoint, INSertion, INTersection, MIDpoint, NEArest, NODe, PERpendicular, QUAdrant and TANgent. These options can be used for drawing lines that connect at certain points to existing lines, circles, arcs, or ellipses. Notice that as options were listed, the first three letters of some of the options were written in capitol letters. Typing these letters at the prompt for start point or endpoint will activate the desired object snap option.

a) CENter - allows object to be drawn based on the center of a circle. A line or arc often needs to be drawn from the center of an existing circle, or several circles may need to be drawn around one center point.
b) ENDpoint - allows an arc, line, or circle to be drawn to or from endpoint of a previous line or arc by snapping to the desired endpoint.
c) INSert - snap to the insertion point of Shape, Text, Attribute, or Block.
d) INTersection - allows a line or arc to be extended to or from the intersection of any lines, arcs or circles.
e) MIDpoint - allows the selection of a line or arc at its midpoint by picking the Midpoint icon.
f) NEArest - to snap to the circle, line, or arc that is nearest to the crosshairs.
g) NODe - to snap to a point.
h) PERpendicular- allows line to be drawn perpendicular to an existing line.
i) QUAdrant - allows a line or arc to be snapped to the 0-, 90-, 180- and 270-degree positions of a circle or arc.
j) TANgent - allows a line to be snapped tangent to a circle or an arc.

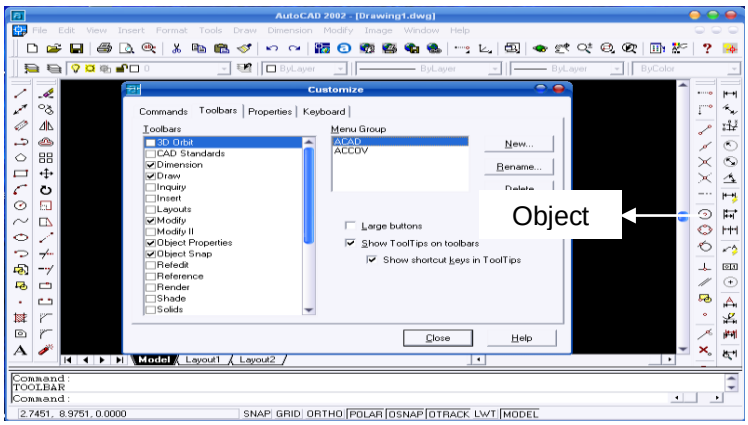


Figure 3: Screen layout for ‘object snap’ command

There are four basic methods of accessing the Osnaps:

1. The Osnaps are available from a *flyout* button on the Standard toolbar.
2. From the pull-down menu, ‘View’ -> Toolbars → ‘Object Snap’.
3. Access the Osnaps from the cursor menu. Hold the *Shift* key down on the keyboard and right-click the mouse to bring up the cursor menu. The menu appears at the current cursor position.
4. Access the Osnaps from the keyboard by typing their abbreviated name. Example: If choose ‘Center’, type CEN <Enter>.

EDITING/ MODIFY COMMAND

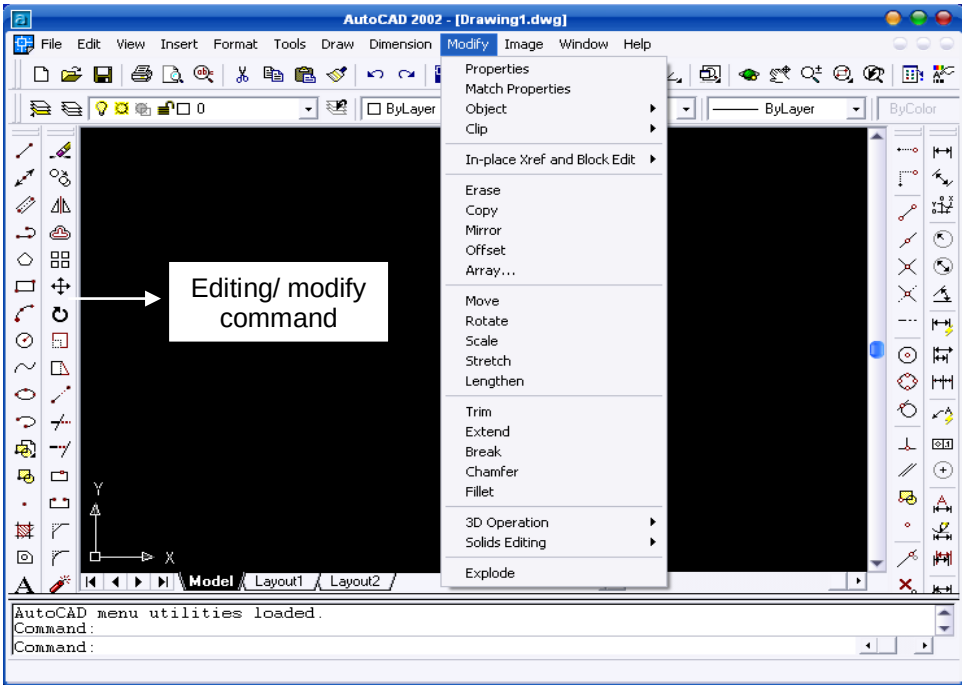


Figure 4: Screen layout for ‘editing/modify’ command

1. ‘ERASE’ (E)

The Erase command is one of the simplest AutoCAD commands and is one of the most used. The command erases (deletes) any selected object(s) from the drawing.

2. ‘UNDO’ (U)

The Undo command takes back the last drawing or modification command that was executed.

3. ‘MOVE’ (M)

The Move command works in a similar way to the Copy command except that no copy is made, the selected object(s) is simply moved from one location to another.

4. ‘COPY’ (CP)

The Copy command can be used to create one or more duplicates of any drawing object or objects which have previously created. To copy more than one object, type M for 'Multiple' after selecting the object.

5. 'MIRROR'

The mirror command will create a mirror image of any selected drawing elements along any line of symmetry specified by the user. You'll given the option of keeping or discarding the original object prior to making the MIRROR permanent. If the object is being completed by MIRROR, don't delete the old object.

6. 'ROTATE'

The rotate command rotates any selected objects about a defined point by the angle specified. By default AutoCAD will rotate objects anticlockwise when an angle is entered.

7. 'OFFSET'

Offset will make a copy of a line or series of selected lines by a specified distance in the direction specified.

8. 'EXTEND'

The extend command is similar to the trim command in how it functions, except it extends a selected line to a point of intersection of another selected object.

9. 'TRIM'

The TRIM command is an extremely useful tool which will erase all parts of an object beyond or within its intersection with another object. The command allows you to edit Lines, Arcs, Circles, or Polylines using a cutting edge.

10. 'BREAK'

The break command is identical to the above break at point command, except the break line isn't as neat as when using the above command i.e the break point leaves a significant gap between what is now two separate objects:

11 'FILLET'

The fillet command is very similar to the chamfer command above, except instead of creating a straight line chamfer, AutoCAD creates a radius between the two points.

12. 'CHAMFER'

The chamfer command will chamfer the intersection of two lines to a specified distance.

13 'STRETCH'

The STRETCH command allows you to elongate or shrink drawing entities. Thos can be especially helpful when editing the shape and size of a floor plan to meet the changing needs. Drawings made with Lines, Arcs, Traces, Solids, and Polylines can all be stretched.

14. 'ARRAY'

The Array command will allow for multiple copies of an object or group of objects in rectangular or circular (polar) patterns.

15. 'SCALE'

The SCALE command will allow changing the size of an existing object or an entire drawing base on scale factor.

SELECT OBJECTS

In addition to selecting objects to edit using the pick box, AutoCAD provides the options of Window, Crossing, CPolygon, WPolygon, Fence, All, Last, Add, Remove, Previous and Multiple to select an object for editing. Each of the following selection methods can be entered at the Select objects: prompt for editing command.

1) Pickbox - only one object is selected at a time.
2) Window (W) – rather than selecting objects to edit one at a time, a group of entities may be selected by surrounding them completely in a window.
3) Crossing (C) – The command process will be the same with as the Window option. Any objects that lie entirely in the window will be edited. Any objects that are partially inside the window also will be edited.
4) Previous (P) – All objects in a previous set.
5) Remove (R) – allow for the objects to be removed from the selection set.
6) Add (A) – allow for additional objects to be added to the selection set.
7) Last (L) – allows the most recently created object to be selected for editing.
8) Fence (F) – allows the long rows or columns to be edited.
9) Window Polygon (WP) – allows for selecting points to form a polygon around objects to be edited.
10) Crossing Polygon (CP) – edit all objects that are in or cross the polygon.
11) All – select all objects in drawing

DISPLAY COMMAND

'Zoom' (Z)

Zooming the drawing area and its contents to different sizes is a very important part of the construction of drawings. This ability provides a close-up view for better accuracy and detail or a distant view to get the whole picture.

a. 'Zoom' → All (A) – zoom the drawing so that it is at its full extent, with its edges touching the edges of the drawing area.
b. 'Zoom' → Center (C) – select a new view by specifying its center point and the magnification value or height of the view in current units.
c. 'Zoom' → Dynamic (D) – provides a quick and easy method to move to another view of drawing. With Zoom Dynamic, you can see the entire drawing and then simply select the location and size of the next view by means of cursor manipulations.
d. 'Zoom' → Extend (E) – lets you see the entire drawing on screen. Unlike the All option, the Extents Option uses only the drawing extents and not the drawing limits.
e. 'Zoom' → Left (L) – display drawing on the bottom left of graphic screen and the object zooming size is similar to 'Zoom' -→ Center.
f. 'Zoom' → Previous (P) – displays the last displayed view.
g. 'Zoom' → Window (W) – specify a smaller area of the part of the drawing being currently displayed and have that portion fill the drawing area.
h. 'Zoom' → Scale (X/XP) – display drawing on graphic layout base on scale or magnification factor.

'Pan' (Z)

AutoCAD allows viewing a different portion of the drawing in the current view without changing the magnification. Unlike the Move command, which moves the drawing objects to different locations on the coordinate system, the pan command does not change the location of the objects on the coordinate system.

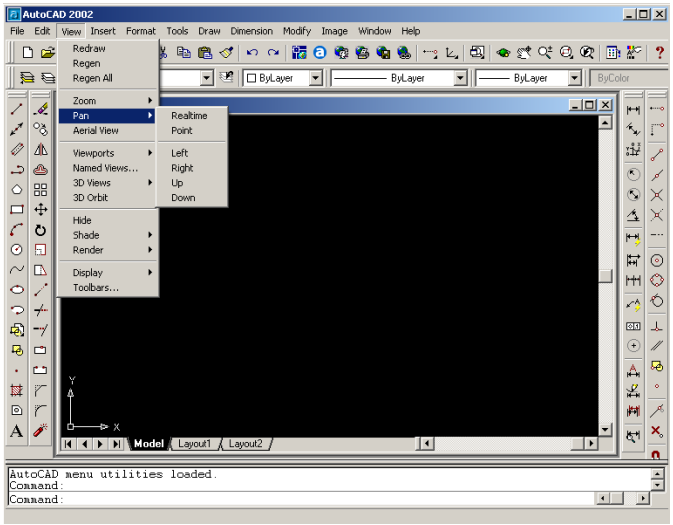


Figure 6: Screen layout for 'pan' command

'Redraw' (R)

The REDRAW command removes the current screen display momentarily, and then replaces the drawing. As the drawing is replaced, blip marks are removed from the drawing editor. Lines that may have been partially erased while editing other entities also will be restored.

'Viewres' (V)

The resolution of arcs and circles is controlled by the VIEWRES command. Resolution refers to the amount of detail that is represented when arcs and circles are drawn. The higher the resolution, the more lines that are used and the smoother the arc or circle will appear.

LAYER (LA)

Layers are a way of managing, tidying and also controlling the visual layout of a drawing. A whole section of a drawing can be turned on or off, or simply one aspect can be controlled - text for example.

Layers are controlled by the layer properties manager dialog box. Enter the LAYER command:-

1. From 'Pull – down Menu' → Format -→ Layer
   ('Layer Properties Manager' dialog box appears on screen)

Or

2. At the command Prompt,
   Command: Layer <Enter>
   ('Layer Properties Manager' dialog box appears on screen)

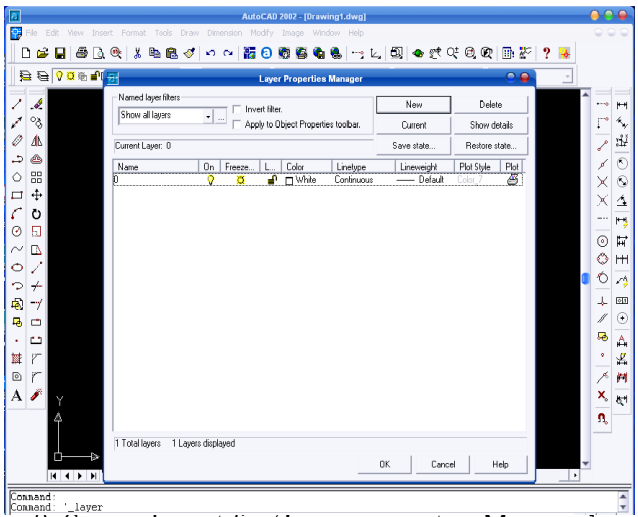


Figure 8: Screen layout for 'Layer properties Manager'

Creating a New Layer

Once in the ‘Layer Properties Manager’ choose the ‘New’ button from the upper right hand corner. The new layer now appears in ‘Layer Properties Manager’ dialog box.

After a layer has been created, colors and linetype can be assigned by layers. To alter the color of a layer, pick the layer so that the layer name is highlighted. The color of each layer can be altered by picking the color icon from the color list or by choosing Color from the detail box. The linetype can be altered using methods similar to altering layer color. Highlighting the name of the layer to be altered, and then selecting the linetype name will display the Select Linetype dialog box. The dialog box can be used to load and select line type.

Setting Layers On/Off

The entire new layers that have been created are set in the ON option. By setting a layer to OFF, objects drawn on that layer will be invisible.

Thawing and Freezing Layers

Freeze – Open the Layer and Linetype Properties dialog box and select the layer that would like to freeze, and pick the freeze button. The sun icon (Thaw) will be change to and icicle (Freeze). Picking the OK button will activate the selection, restore the drawing screen, and remove the objects on the selected layer from the display.

Thaw – The Thaw option of the LAYER command will allow for specified frozen layers to be thawed. To thaw a layer that is frozen, open the Layer and Linetype Properties dialog box and select the icicle icon of the layer to be thawed. As the icicle is picked, it will change to the sun icon, indicating that the layer has been thawed. Picking the OK button will activate the selection, restore the drawing screen and display the material on the layer that had been frozen.

‘PROPERTIES’ – Modify/Change

MODIFY – The command can be used to control the properties of existing objects. The command can be started by picking the Properties icon from the far right side of the Object Properties toolbar, by selecting ‘Properties’ from the Modify pull-down menu, or by typing DDMODIFY at the command prompt. Once the entity is selected, the Modify Line box is displayed. The box allows for altering the Color, Layer, Linetype, Handle, Thickness, and Ltscale.

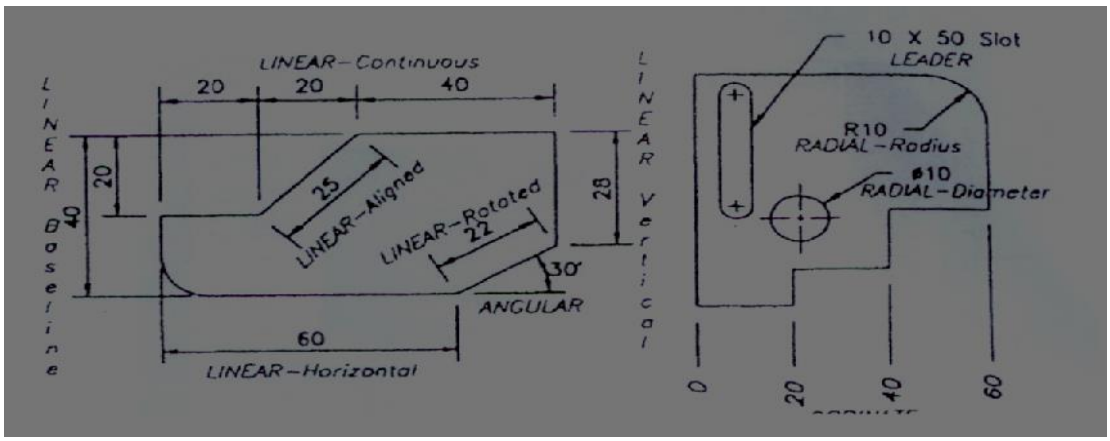
CHANGE – The final command to be considered when setting linetype, color, or layers. Each of the attributes related to layers can be altered within the LAYER command. When a color of layer is changed from red to blue, all lines on the specified layer are become blue. The CHANGE command will allow for only selected entities to be altered.

TEXT

Text is used to label the various components of the drawing and to create the necessary shop or field notes needed for fabrication and construction of the design. AutoCAD includes a large number of text fonts. Text can be stretched, compressed, obliqued, mirrored, or drawn in a vertical column by applying a style. Each text string can be sized, rotated, and justified to meet drawing needs. The Justify option allows placing text in one of the 14 available justification points. AutoCAD prompts

The Style option of the TEXT commands are allows determining how text characters and symbols appear, other than adjusting the usual height, slant, and angle of rotation. To specify a text style from the Style option of the TEXT commands, it must have been defined by using the STYLE command. In other words, the style command creates a new style or modifies an existing style. AutoCAD displays the Text Style dialog box.

DIMENSION



AutoCAD provides a full range of a dimensioning commands and utilities. The dimension types available are Linear, Angular, Diameter, Radius and Ordinate.

1). Linear Dimension
   - Horizontal
   - Vertical
   - Aligned
   - Continuous
   - Baseline
   - Rotated

2). Radial Dimension
   - Radius
   - Diameter

3). Angular Dimension

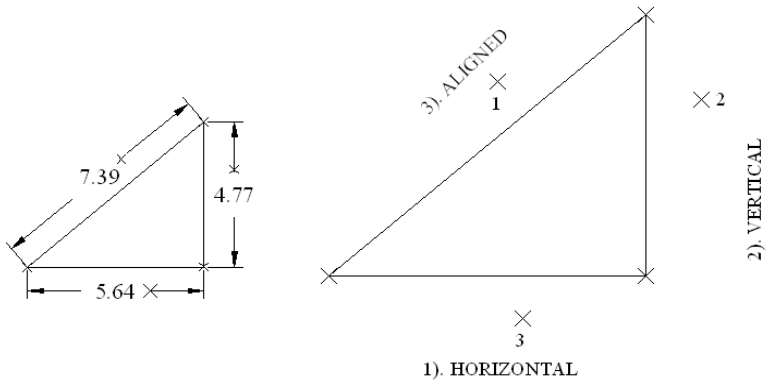
4). Leader Dimension

5). Ordinate Dimension

There are also other general utility, editing, and style-related commands and subcommands that assist in drawing the correct dimensions quickly and with accuracy. AutoCAD provides a comprehensive set of subdialog boxes accessible through the Dimension Style Manager dialog box for creating new Dimension Style and managing existing ones. AutoCAD displays the Dimension Style Manager dialog box.

COMMAND: DIMENSION

- Linear (Horizontal, Vertical & Aligned)



RESULT: The basic drawing elements were drawn using AutoCAD

EXERCISE 2

AIM: To draw the projections of a 75mm long straight line, in the following positions:

(i) Parallel to both the H.P. and the V.P. and 25mm from each.
(ii) Parallel to and 30mm above the H.P. and in the V.P.
(iii) Parallel to and 40mm in front of the V.P. and in the H.P

PROCEDURE:

Case (i):

1. Draw a horizontal line XY.
2. Draw a line of length 75mm parallel to XY at a distance of 25mm above it. Mark the ends as a', b'. This is the FV.
3. Draw a line of length 75mm parallel to XY at a distance of 25mm below it. Mark the ends as a, b. This is the TV.

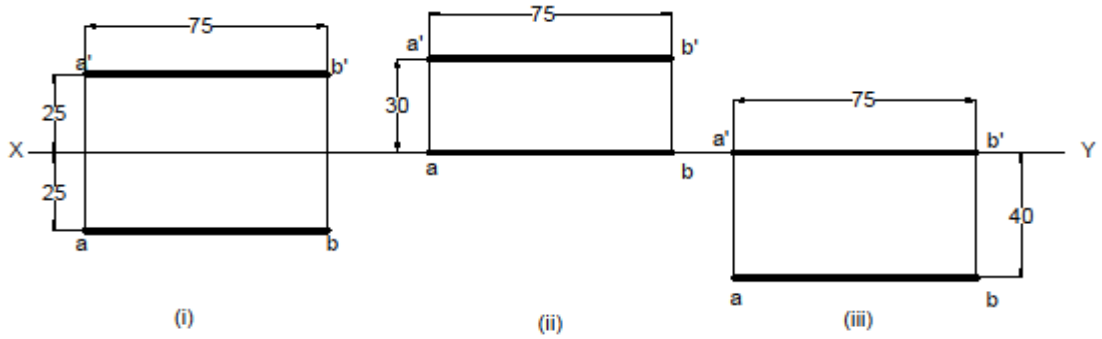
Case (ii):

1. Draw a horizontal line XY.
2. Draw a line of length 75mm parallel to XY at a distance of 30mm above it. Mark the ends as a', b'. This is the FV.
3. Project the above line on XY. Mark the ends as a, b. This is the TV.

Case (iii):

1. Draw a horizontal line XY.
2. Draw a line of length 75mm parallel to XY at a distance of 40mm below it. Mark the ends as a, b. This is the TV.
3. Project the above line on XY. Mark the ends as a', b'. This is the FV.

RESULT: The required projections are obtained in AutoCAD.


75
a'
b'
25
25
X
a
b
(i)
75
a'
b'
30
a
b
(ii)
75
a'
b'
Y
40
a
b
(iii)

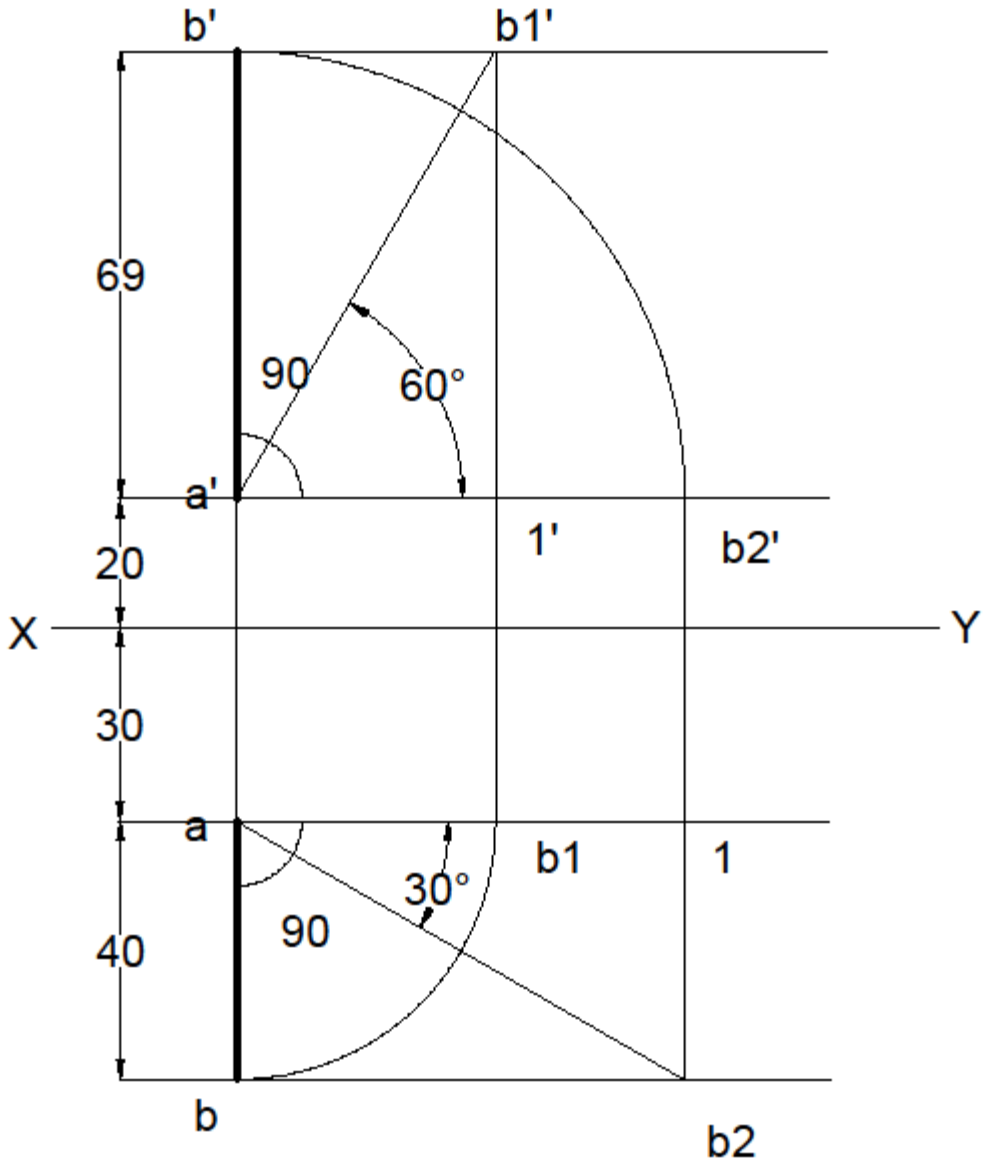
EXERCISE 3

AIM: To draw projections of a line AB of length 80mm when it is makes an angle of 60^0 with H.P. and 30^0 with V.P and one end of the line is 20mm above H.P. and 30mm in front of V.P.

PROCEDURE:

1. Draw a horizontal line XY.
2. Create a point **a'** at a distance of 20mm above XY and project it 30mm below XY and mark its TV as point **a**.
3. Draw an 80mm long line at an angle of 60^0 with XY from point **a'** above XY and at an angle of 30^0 from point **a** below XY. Mark the ends of the lines as points **b'** and **b** respectively.
4. Draw horizontal lines through the ends to represent locus of the ends of the line.
5. Project the line a'b' to the locus of a and rotate to meet the locus of b below XY. Mark the point of intersection as b. Join ab. This is TV of the line AB.
6. Project the line ab to the locus of a' and rotate to meet the locus of b'. Mark the point of intersection as b'. Join a'b'. This is FV of the line AB.

RESULT: Thus, the required projections are obtained using AutoCAD.


b'
b1'
69
90
60°
a'
1'
b2'
20
X
Y
30
a
b1
1
30°
90
40
b
b2

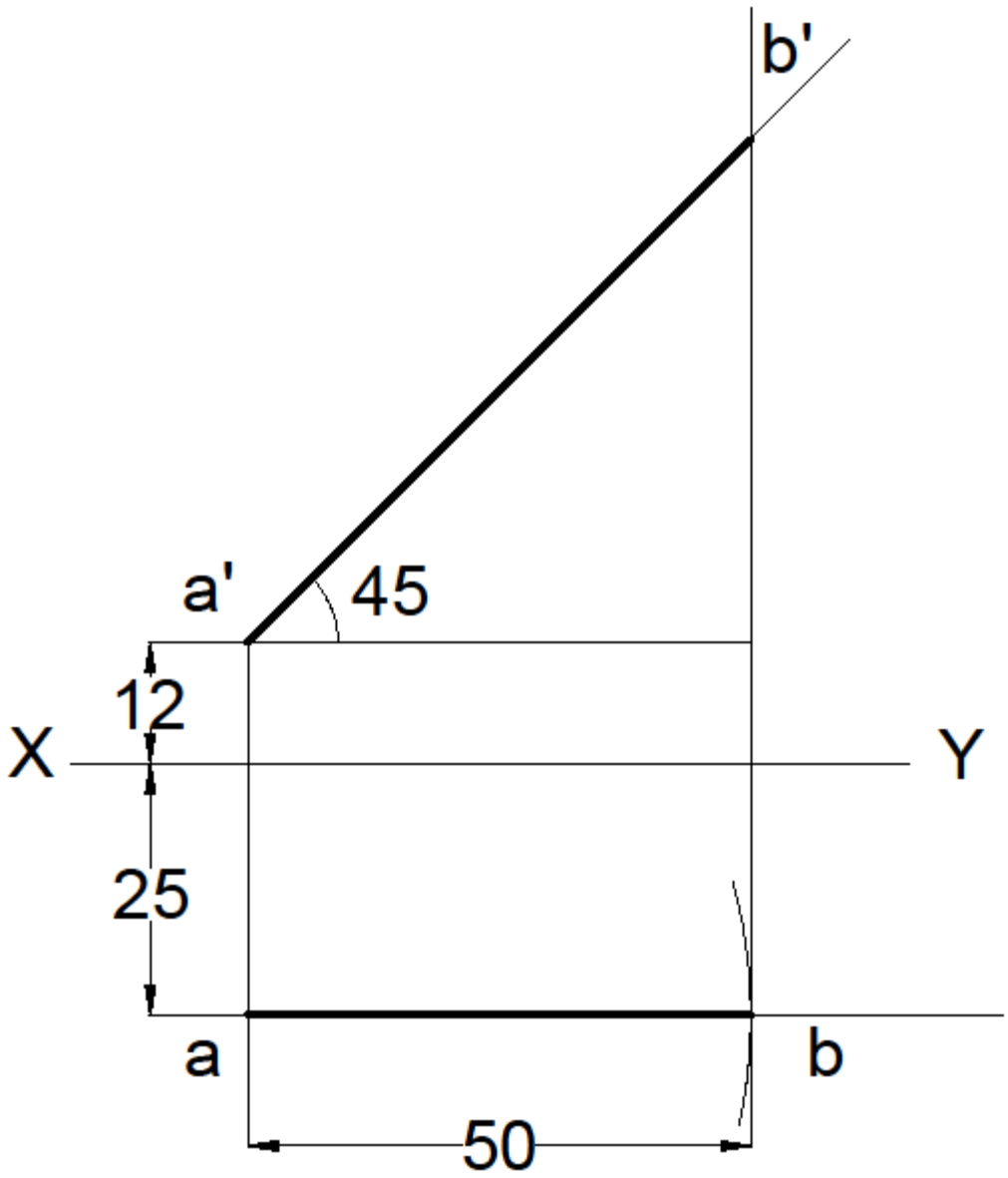
EXERCISE 4

AIM: To draw projections and determine true length of the line whose top view is 50mm long; parallel to V.P. and inclined at 45^0 to the H.P. An end of the line is 12mm above the H.P. and 25mm in front of the V.P.

PROCEDURE:

1. Draw a horizontal line XY.
2. Create a point **a'** at a distance of 12mm above XY and project it 25mm below XY and mark its TV as point **a**.
3. Draw a line at an angle of 45^0 with XY from point **a'** above XY.
4. Draw a horizontal line through a and mark a length of 50 mm on the line. This is the TV of the line.
5. Project the TV above XY to meet the line drawn in step 3 at b'. a'b' is the FV of the line.

RESULT: Thus, the required projections are obtained using AutoCAD. True length of the line is found to be 70.71mm.


b'
a'
45
12
X
Y
25
a
b
50

EXERCISE 5

AIM: To draw projections of regular pentagon of 25mm side which has a side on the ground and its plane is inclined at 45^0 to the H.P. and perpendicular to the V.P.

PROCEDURE:

1. Draw a horizontal line XY.

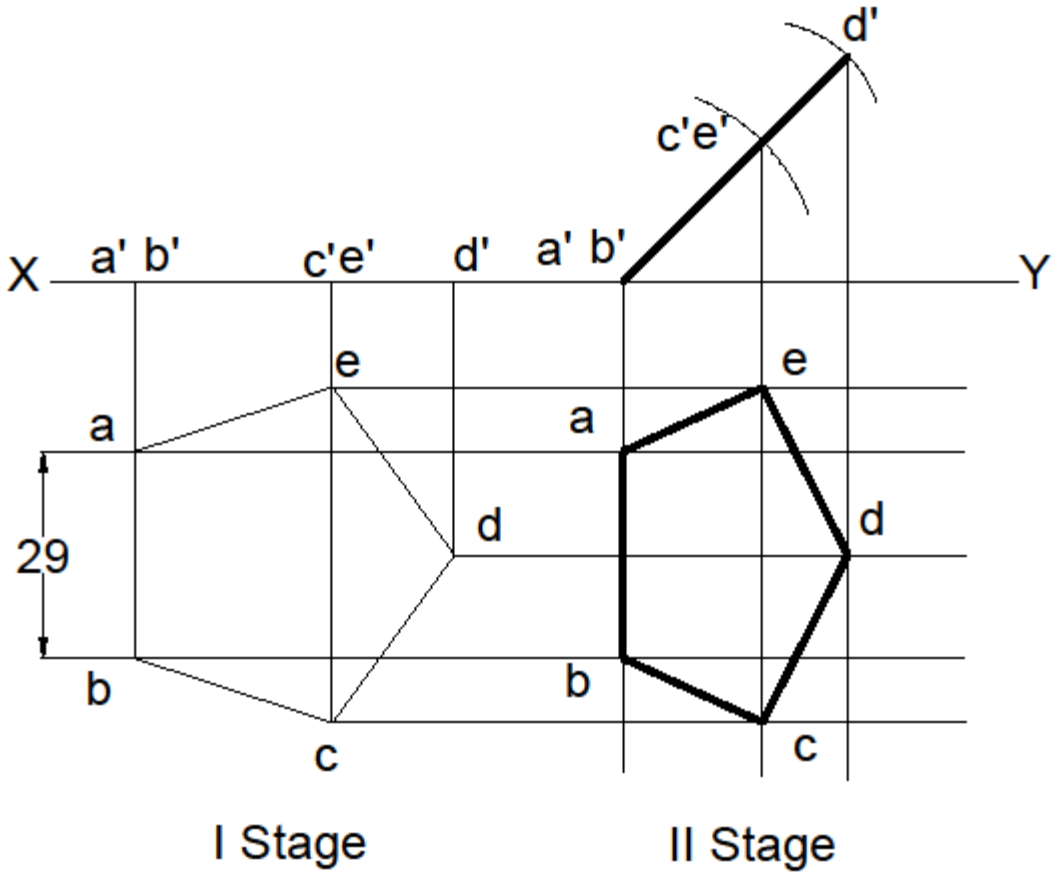
Stage I: Pentagonal plane is placed on HP with one edge perpendicular to XY.

2. Draw a regular pentagon abcde of 25mm side below XY such that an edge is perpendicular to XY. This is TV of the first stage.
3. Project the vertices of the pentagon on XY and mark the points as a',b',c',d',e'. This is FV of the first stage.

Stage II: Pentagonal plane has one side on HP and inclined at an angle of 45^0 to HP.

4. Reproduce the FV of first stage keeping a' on XY and a'd' inclined at 45^0 to XY. This is the required FV.
5. Draw projectors from the a',b',c',d',e' to the TV to intersect the horizontal locus lines from TV of the first stage.
6. Mark the points of intersection as abcde. This is the required TV.

RESULT: Thus, the required projections are obtained using AutoCAD.


d'
c'e'
X
a' b'
c'e'
d'
a' b'
Y
e
e
a
a
d
d
29
b
b
c
c
I Stage
II Stage

EXERCISE 6

AIM: To draw the projections of the following solids situated in their respective positions, taking a side of base 40mm long or the diameter of the base 50mm long and the axis 65mm long.

(i) A hexagonal pyramid, base on the H.P. and a side of the base parallel to and 25mm in front of the V.P.

(ii) A cone, apex in the H.P. axis vertical and 40mm in front of the V.P.

PROCEDURE:

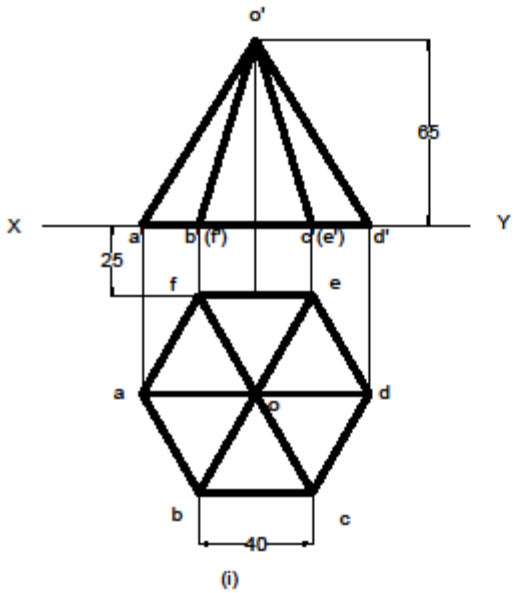
Case I: Hexagonal Pyramid

1. Draw a horizontal line XY.
2. Draw a hexagon in TV such that side closer to XY is parallel to XY and is 25mm below XY.
3. Locate the centre of hexagon and join it with the vertices.
4. Project the vertices of the hexagon on XY. Project the centre on XY and extend it in FV for a length of 65mm (axis length) to locate the apex of the pyramid.
5. Join the vertices projected on XY to the apex. This forms the FV of the pyramid.

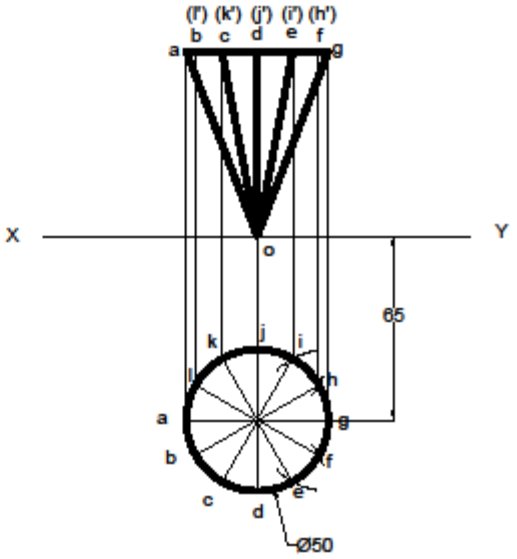
Case II: Cone

1. Draw a horizontal line XY.
2. Draw a circle of radius 25mm in TV with its centre at 65mm below XY.
3. Divide the circle in 12 parts.
4. Project the centre of the circle on XY and mark it as the apex of the cone. Extend this point to a height of 65mm and draw a horizontal line through it.
5. Project the points on circle to that line. Join these points to the apex.

RESULT: Thus, the required projections are obtained using AutoCAD.


o'
65
X
Y
a
b (f')
c (e')
d'
25
f
e
a
o
d
b
c
40

(i)


(f') (k') (j') (i') (h')
b c d e f
a
g
X
Y
o
65
k
j
i
l
h
a
g
b
f
e
c
d
Ø50

(ii)

EXERCISE 7

AIM: To draw the projections of a pentagonal prism, base 25mm side and axis50mm long, resting on one of its rectangular faces on the H.P. with the axis inclined at 45^0 to the V.P.

PROCEDURE:

1. Draw a horizontal line XY.

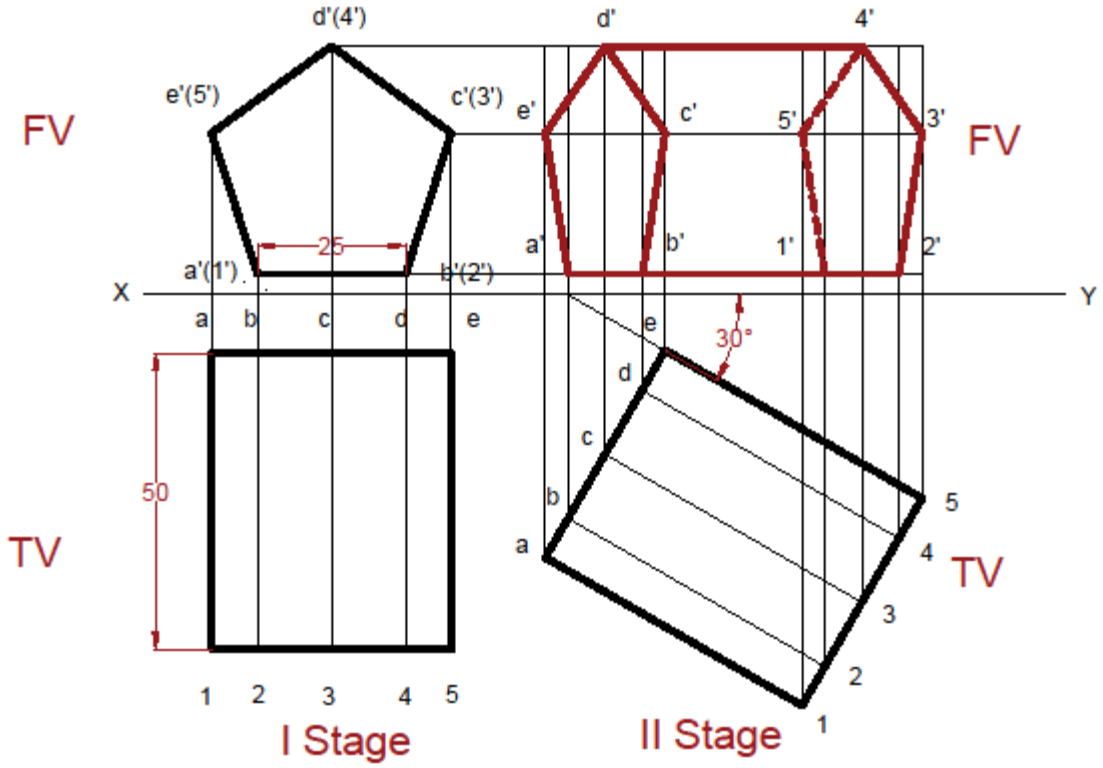
Stage I

2. Draw a pentagon of side 25mm on XY and project the points to a convenient distance below XY.
3. Draw a rectangle of length 50mm along the projection lines and width same as the width of the pentagon in FV. This is the TV of prism placed with its rectangular face on HP and axis perpendicular to VP.

Stage II

4. Reproduce the TV along a line inclined at an angle of 45^0 to XY.
5. Join the points of intersection of horizontal lines drawn from the FV in stage I and projection lines drawn from the TV in stage II to draw the FV of stage II.

RESULT: Thus, the required projections are obtained using AutoCAD.


FV
TV
d'(4')
e'(5')
c'(3')
a'(1')
b'(2')
25
X
Y
a
b
c
d
e
50
1
2
3
4
5
I Stage
d'
4'
e'
c'
5'
3'
a'
b'
1'
2'
FV
30°
e
d
c
b
a
5
4
3
2
1
TV
II Stage

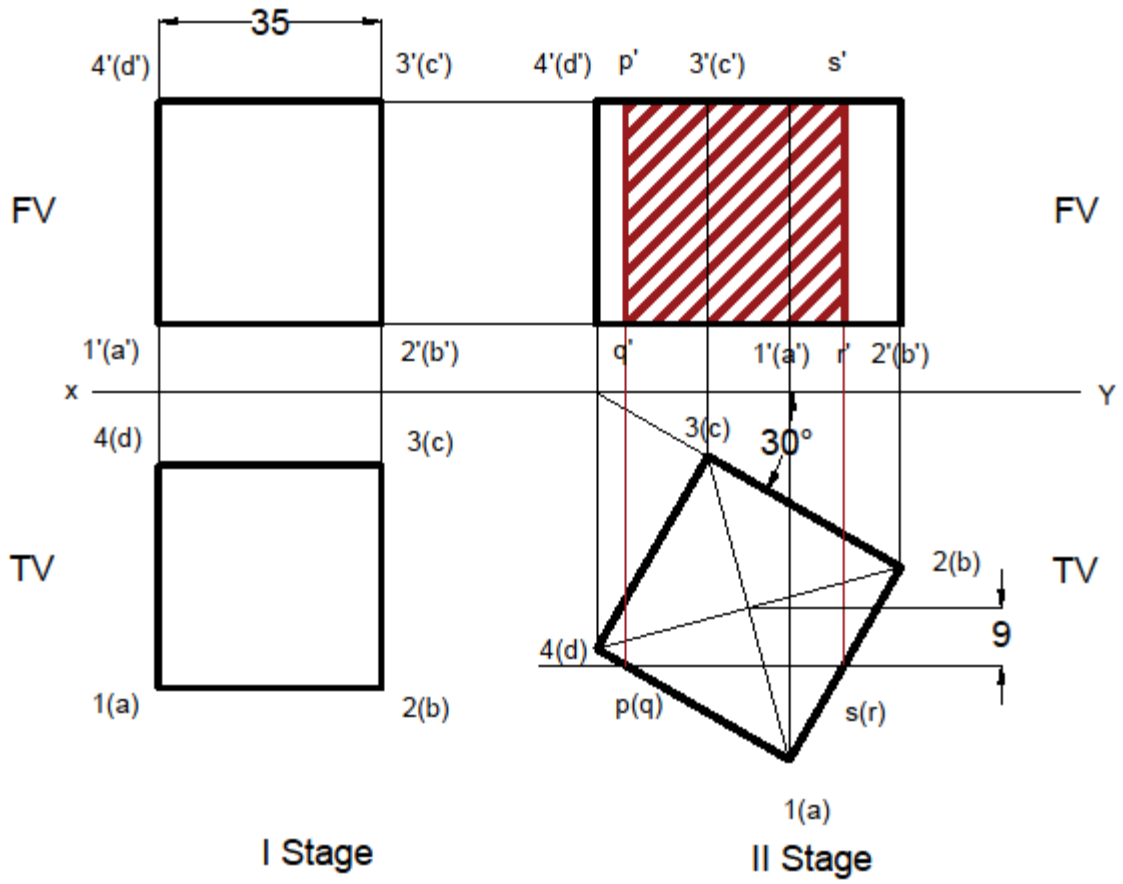
EXERCISE 8

AIM: To draw FV and TV of a cube of 35mm long edges is resting on the H.P. on one of its faces with the vertical face inclined at 30^0 to the V.P. It is cut by a section plane parallel to the V.P. and 9mm away from the axis and further away from the V.P.

PROCEDURE:

1. Draw a horizontal line XY.
2. Draw the projections of a cube placed on HP with a face parallel to VP in Stage I
3. Reproduce the TV of the cube in Stage I at an angle of 30^0 in Stage II.
4. Draw the FV of Stage II by joining the points of intersection of horizontal lines from FV of Stage I and projection lines from Stage II.
5. Draw a line 9mm away from the axis in TV of Stage II such that is further away from XY.
6. Mark the points of intersection and project them in the FV of Stage II
7. Join the points thus obtained appropriately and hatch the section.

RESULT: Thus, the required section is obtained using AutoCAD.


35
4'(d')
3'(c')
4'(d')
p'
3'(c')
s'
FV
FV
1'(a')
2'(b')
q'
1'(a')
r'
2'(b')
x
Y
4(d)
3(c)
3(c)
30°
TV
TV
2(b)
9
4(d)
1(a)
2(b)
p(q)
s(r)
1(a)
I Stage
II Stage

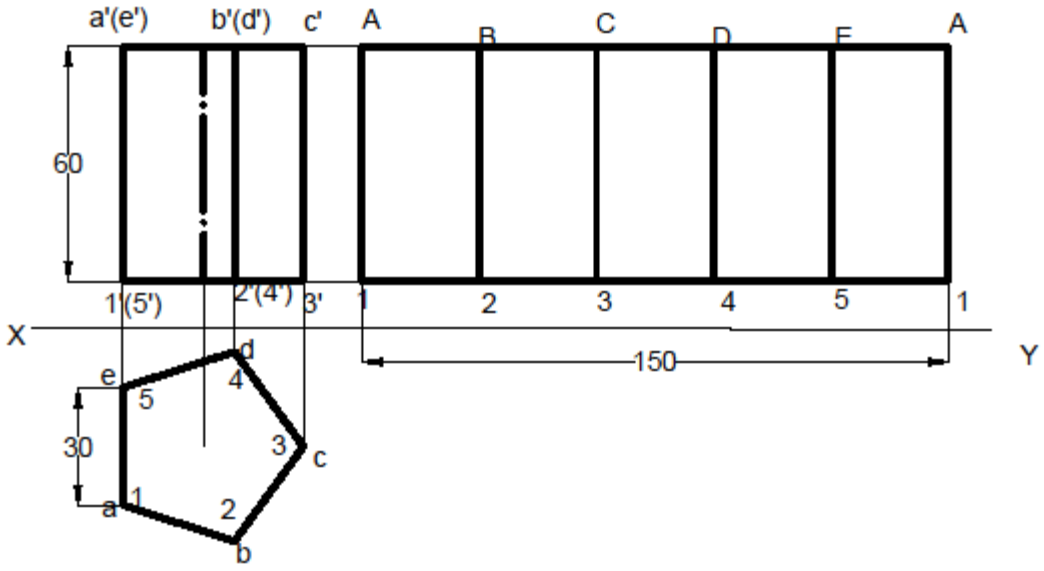
EXERCISE 9

AIM: To draw the development of lateral surface of the pentagonal prism having the base 30mm and height 60mm.and one edge of the base perpendicular to the V.P.

PROCEDURE:

1. Draw a horizontal line XY.
2. Draw projections of a pentagonal prism resting on HP such that an edge of the base is perpendicular to XY.
3. Draw a horizontal line along XY of the same length as the perimeter of the base of the prism.
4. Divide the line in 5 parts and project the points of division to a height of 60mm. Join them to form the top of prism. The figure thus obtained is the development of the pentagonal prism.

RESULT: Thus, the required development is obtained using AutoCAD.


a'(e')
b'(d')
c'
A
B
C
D
E
A
60
1'(5')
2'(4')
3'
1
2
3
4
5
1
X
Y
150
d
e
4
5
3
c
30
1
2
a
b

EXERCISE 10

AIM: To draw the development of lateral surface of

i. A cylinder having the base 50mm diameter and height 60mm.
ii. A cone having the base 40mm diameter and 60mm long axis is cut by a section plane passing through the midpoint of the axis making an angle of at 30^0 with base.

PROCEDURE:

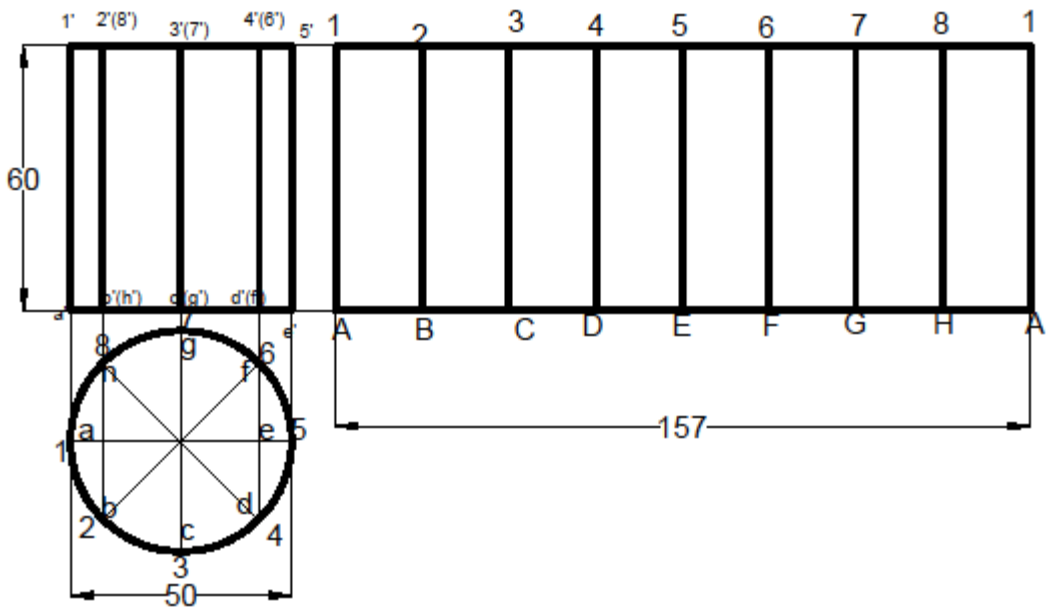
Case I

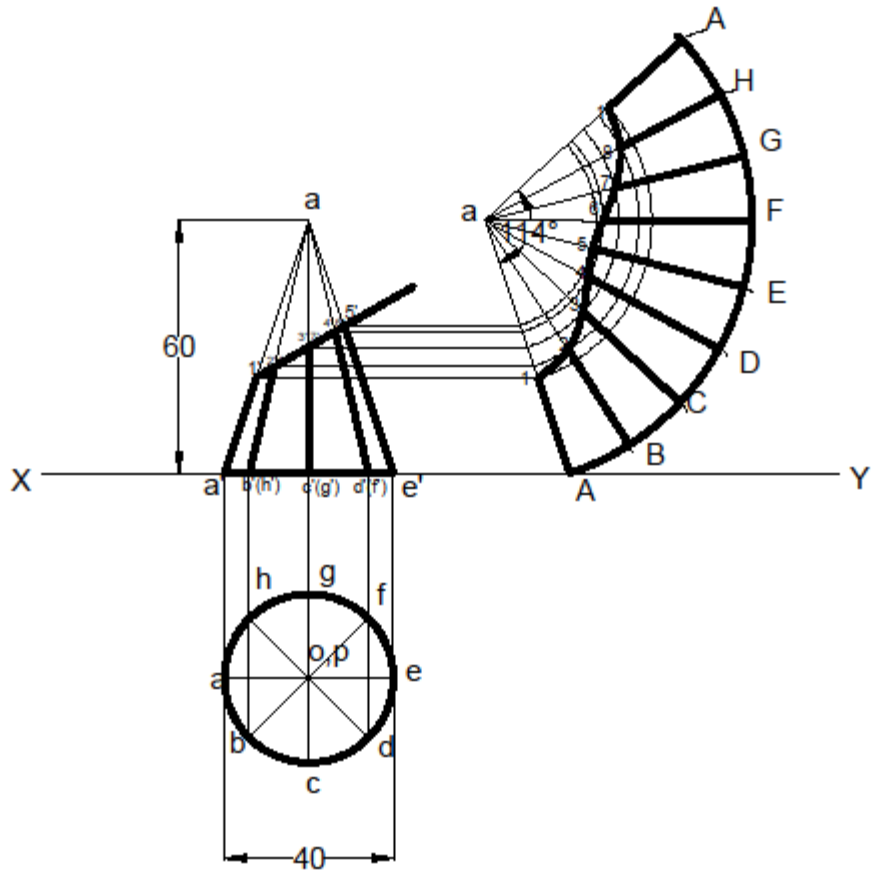
1. Draw a horizontal line XY.
2. Draw a circle of 50mm diameter in TV. Divide the circle into 8 parts and project the points above XY to draw the FV of the cylinder.
3. Draw a line on XY of length same as the circumference of the base of the cylinder and divide it into 8 parts.
4. Erect perpendiculars to a height of 60 mm from these points of division to draw the top of the cylinder. The figure thus obtained is the development of the cylinder.

Case II

1. Draw a horizontal line XY.
2. Draw a circle of 40mm diameter in TV. Divide the circle into 8 parts and project the points to XY.
3. Mark the apex of the cone on the axis at a distance of 60mm from XY. Join the points on XY to the apex to obtain the FV of the cone.
4. Measure $\Theta = \frac{r}{l}(2\pi)$; r= radius of the circle, l= Slant height of the cone
5. Draw a line equal to slant edge. With one end of the line as centre, draw an arc subtending Θ at the centre.
6. Divide the arc into 8 parts and join the points of division with the centre.
7. Draw a line through the midpoint of the axis in FV obtained in step 3, making 30^0 with the base. Mark the points of intersection.
8. Mark these points on the line drawn in step 5 and draw arcs through them.
9. Join the points suitably to obtain the development of the cut section.

RESULT: Thus, the required development is obtained using AutoCAD.


1'
2'(8')
3'(7')
4'(6')
5'
1
2
3
4
5
6
7
8
1
60
A
B
C
D
E
F
G
H
A
157
50


A
H
G
F
E
D
C
B
A
a
114°
a
60
X
Y
a
b'(h')
c'(g')
d'(f')
e'
h
g
f
o
p
a
e
b
d
c
40

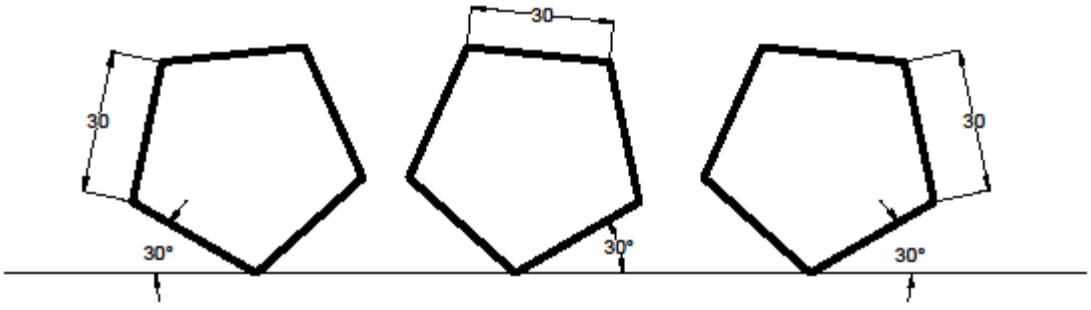
EXERCISE 11

AIM: Construct isometric view of regular pentagon of side 30mm.

PROCEDURE:

1. Draw a horizontal line for reference.
2. Switch to isometric snap.
3. Draw a pentagon with edge length 30mm in left plane. Take the base along the axis making 150^0 with the reference line.
4. Switch to right plane to draw the pentagon. Take the base along the axis making 30^0 with the reference line.
5. Switch to top plane to draw the pentagon. The two axes are inclined at 30^0 and 150^0 in top plane. Thus the pentagon drawn appears like a pentagon drawn in left or right plane.

RESULT: Thus, the isometric views are obtained using AutoCAD.



Isometric View of Pentagon

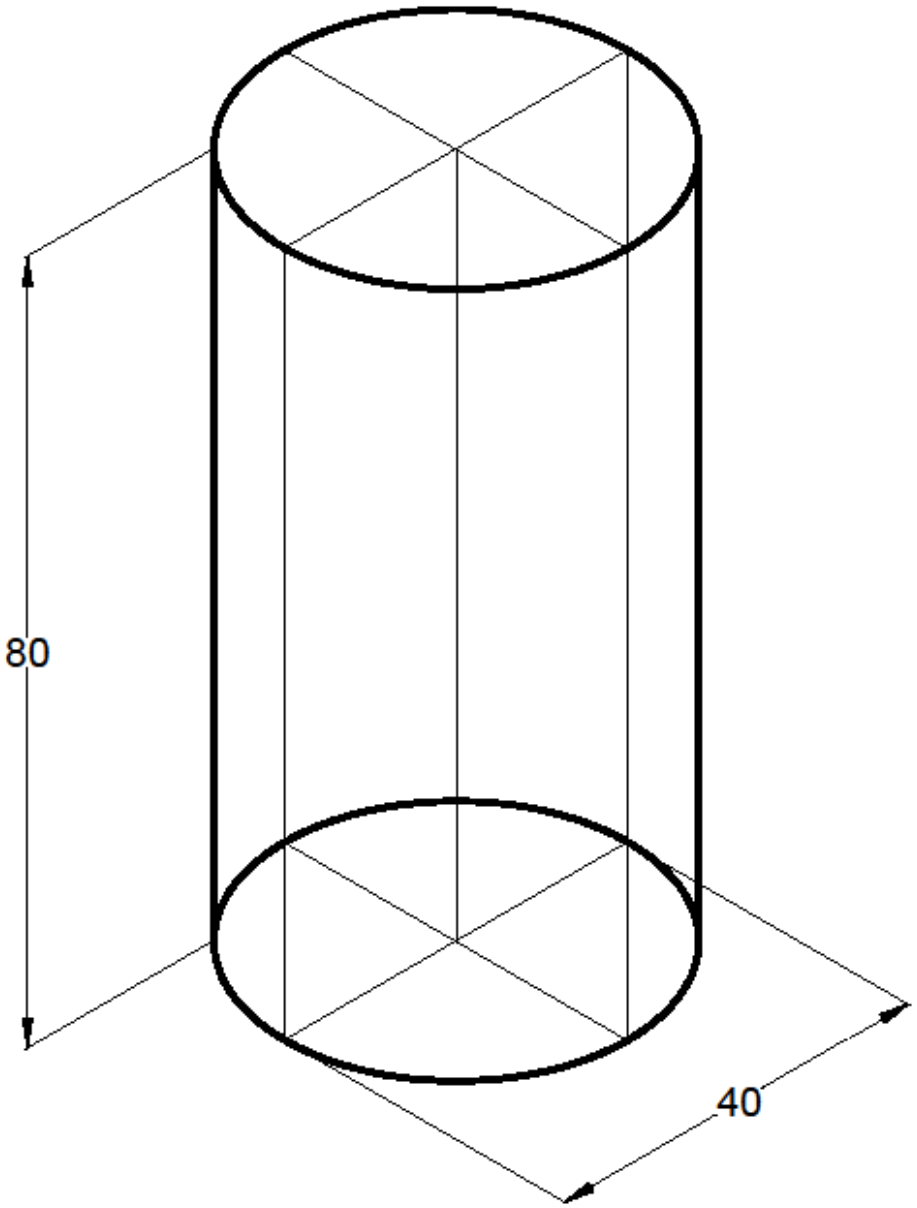
EXERCISE 12

AIM: Draw the cylinder of base 40mm diameter and height 80mm in isometric view.

PROCEDURE:

1. Switch to isometric snap.
2. Draw an isocircle of diameter 40mm in isometric view. Use Ellipse>Isocircle
3. Mark the centre of the circle. Erect the axis of length 80mm through the centre.
4. Draw another isocircle at the end of the axis.
5. Extend lines through diametric ends of the base isocircle to the top isocircle.
6. Draw lines parallel to the axis at the extremes of the isocircles drawn to represent the end generators of the cylinder.

RESULT: Thus, the isometric view of the cylinder is obtained using AutoCAD.



Isometric View of Cylinder

EXERCISE 13

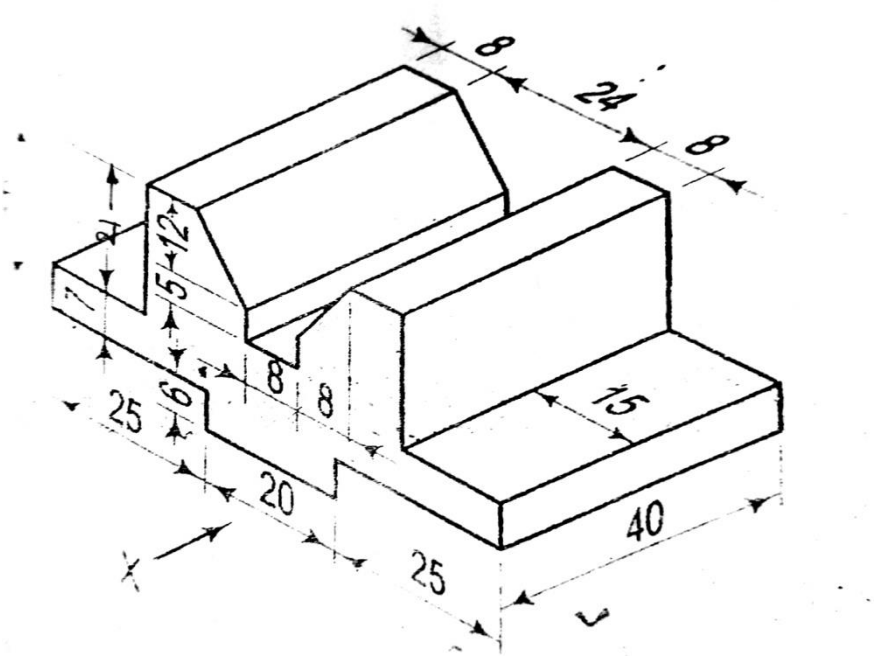
AIM: Draw Orthographic projections for a given Isometric drawing.

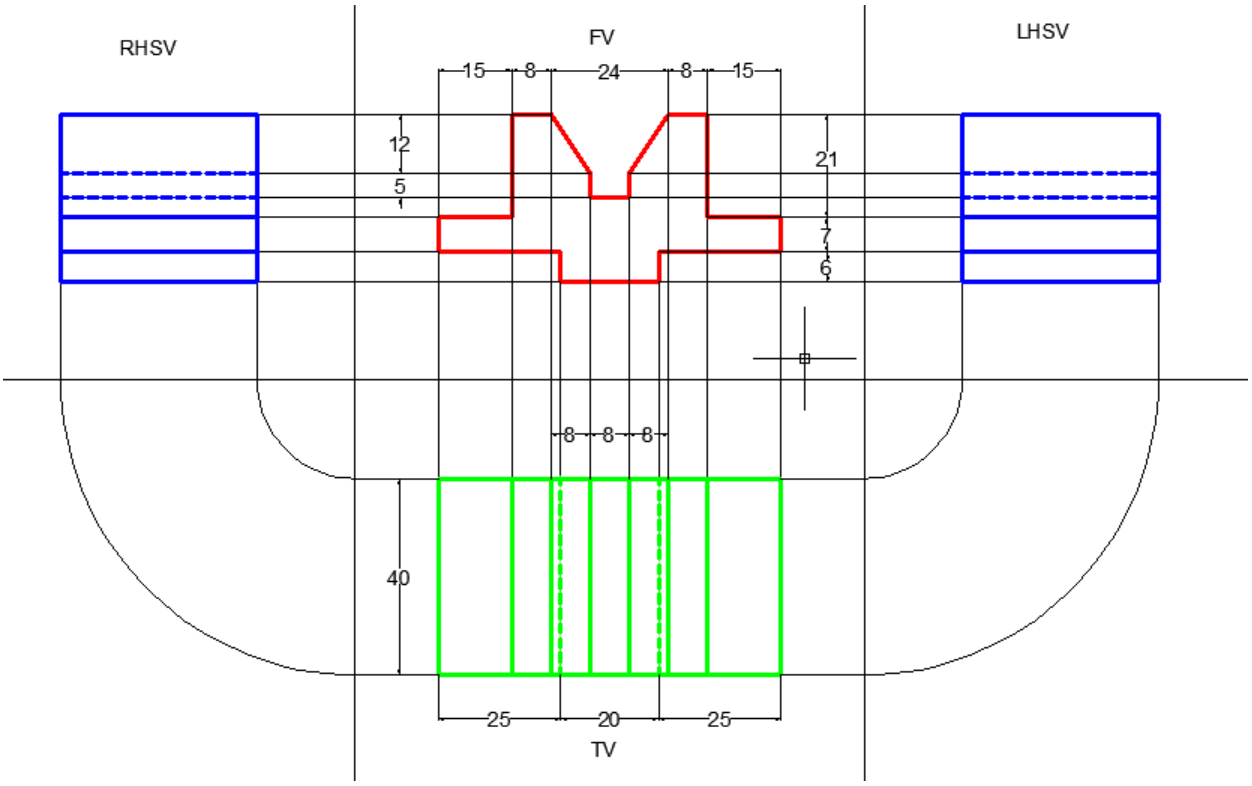
PROCEDURE:

1. Type limits in command menu and set them to 297,210 for A4.
2. Change the units to millimeters from inches and also precision to 0 by clicking Format > Units >Ok.
3. To view the page type zoom > enter and type a > enter in command bar.
4. Draw a horizontal line for reference
5. Use toolbar options to draw the various views. Draw FV. Project to draw TV. Use projections from FV, TV to draw LHSV, RHSV.
6. Edit the drawings by using edit commands.
7. Save the drawings.
8. Plot the completed views of drawing

RESULT:

The required Orthographic projections are drawn from given isometric view by using AutoCAD software.


8
24
8
21
12
5
7
6
8
8
25
20
15
40
25
X
Y


RHSV
FV
LHSV
15
8
24
8
15
12
5
21
7
6
8
8
8
40
25
20
25
TV

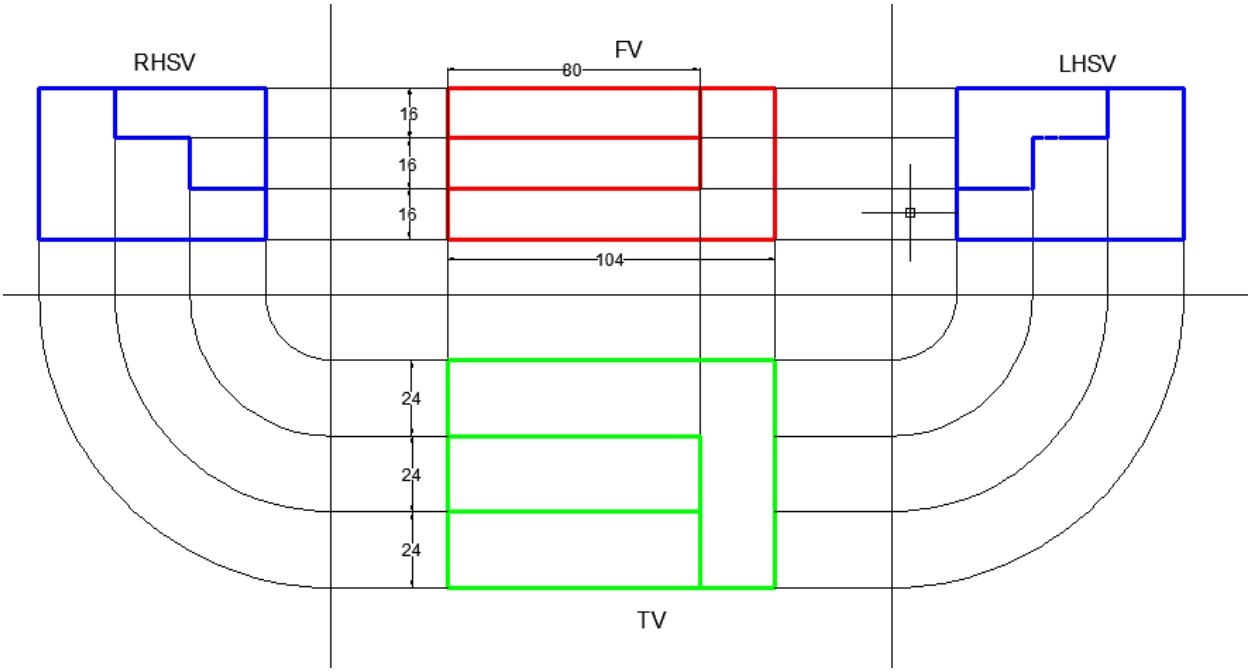
EXERCISE 14

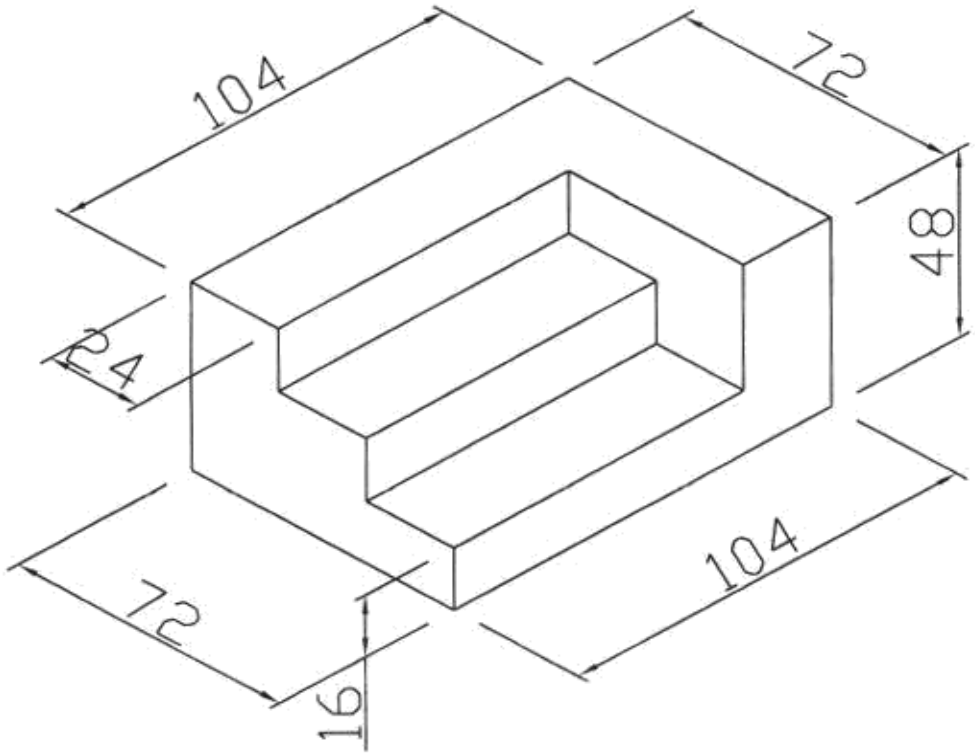
AIM: To create a 2D isometric view of the given diagram using Auto CAD.

PROCEDURE:

1. Type limits in command menu and set them to 297,210 for A4.
2. Change the units to millimeters from inches and also precision to 0 by clicking Format > Units >Ok.
3. To view the page type zoom > enter and type a > enter in command bar.
4. Go to drafting settings and turn on isometric snap.
5. Use the F5 key to change between the views of isometric planes.
6. Start from the front view and draw a line of length of line 104mm using the F8 key and continue with the 48 length line.
7. Change to top plane and draw the 72mm line.
8. Continue in the same fashion to complete the whole figure.
9. Give the dimensions from the dimension tool bar as in diagram.

RESULT: Hence the required 2D isometric diagram is created using Auto CAD.


RHSV
FV
LHSV
80
16
16
16
104
24
24
24
TV


104
72
48
24
72
104
16