



Varuwan Vadivelan Institute of Technology

Dharmapuri – 636 703

LAB MANUAL

Regulation : 2013
Branch : *B.E. - Civil Engineering*
Year & Semester : I Year / II Semester

**CE6261-COMPUTER AIDED DRAFTING AND
MODELLING LABORATORY**



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1		Study of Capabilities of Software for Drafting and Modeling – Coordinate Systems		
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ANNA UNIVERSITY-CHENNAI

Regulation 2013

GE6261 COMPUTER AIDED DRAFTING AND MODELING LABORATORY

OBJECTIVES:

- To develop skill to use software to create 2D and 3D models.

LIST OF EXERCISES USING SOFTWARE CAPABLE OF DRAFTING AND MODELING

1. Study of capabilities of software for Drafting and Modeling – Coordinate systems (absolute, relative, polar, etc.) – Creation of simple figures like polygon and general multi-line figures.
2. Drawing of a Title Block with necessary text and projection symbol.
3. Drawing of curves like parabola, spiral, involute using Bspline or cubic spline.
4. Drawing of front view and top view of simple solids like prism, pyramid, cylinder, cone, etc, and dimensioning.
5. Drawing front view, top view and side view of objects from the given pictorial views (Ex. V-block, Base of a mixie, Simple stool, Objects with hole and curves).
6. Drawing of a plan of residential building (Two bed rooms, kitchen, hall, etc.)
7. Drawing of a simple steel truss.
8. Drawing sectional views of prism, pyramid, cylinder, cone, etc,
9. Drawing isometric projection of simple objects.
10. Creation of 3-D models of simple objects and obtaining 2-D multi-view drawings from 3-D model.

TOTAL HOURS=60Hrs

Note: Plotting of drawings must be made for each exercise and attached to the records written by students.

INTRODUCTION TO AUTOCAD

DRAWING AND MODIFYING OBJECTS

AutoCAD is drafting/modeling software used all over the world by almost all Manufacturing companies. It is variable software which can be used in all engineering divisions. It is a drafting version popularly known to everyone associated with mechanical engineering. The AutoCAD drawing enables the designer to communicate his ideas to the outside of department easily.

Conversion of AutoCAD files to other software is also using drawing exchange Formula. In the lesson, commands and procedures for drawing and modifying the Objects are explored.

START AND SAVE A DRAWING

When you start a drawing, you specify the type of units and other settings you can also choose how to save your files, including saving back up files.

The settings you select, English a metric determines default values used for many system variables of controlling text dimensions, grid, snap and default line type and hatch pattern life.

ENGLISH - Creates a new drawing based on Imperial Measurement System. The drawing is based on cad.dwt template.

METRIC

Creates a new drawing based on metric measurement. The drawing is based on Ocadiso.dwt template. Save drawing files for later use.

CONTROL THE VIEWS

You can magnify the details in your drawing for a closer view or shift the view to a different part of the drawing. If you can save the view by name, you can restore them later.

UNITS

Every object is measured in units. In AutoCAD we can determine the value of the units before we draw.

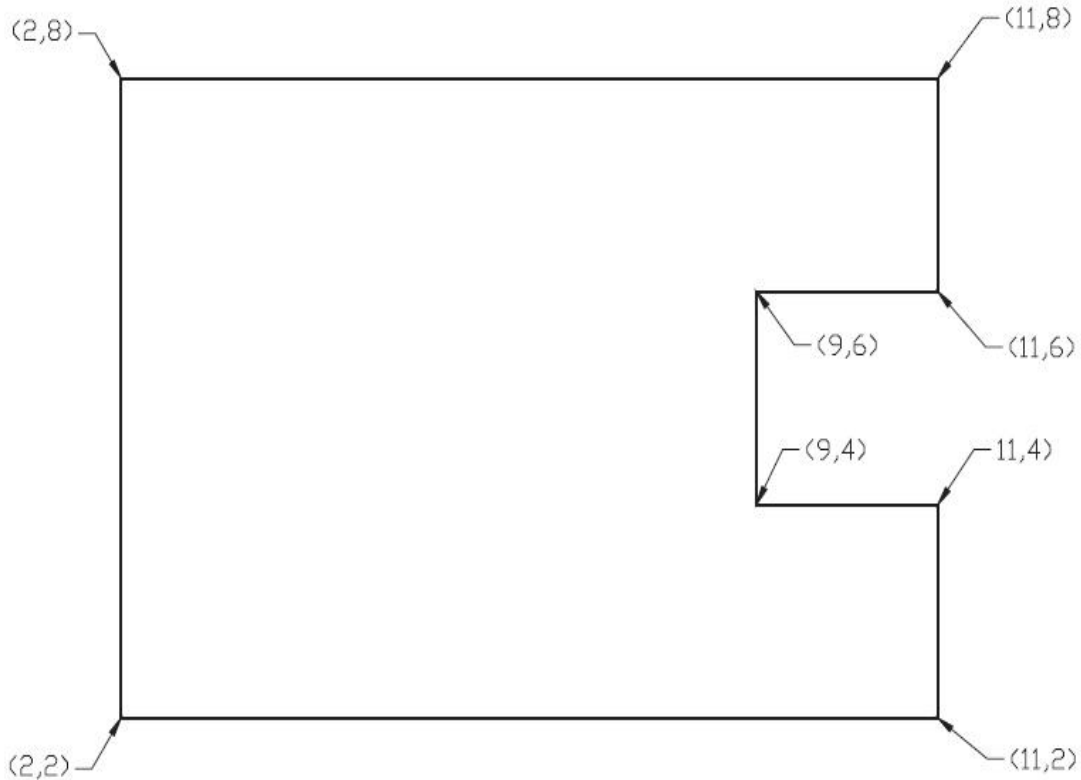
LIMITS

The drawings limits are two-dimensional points in the world coordinate that represent a lower-left limit and an upper right limit. You cannot impose limits on the Z direction.

AUTOCAD COORDINATE ENTRY METHODS

ABSOLUTE METHOD: (X,Y)

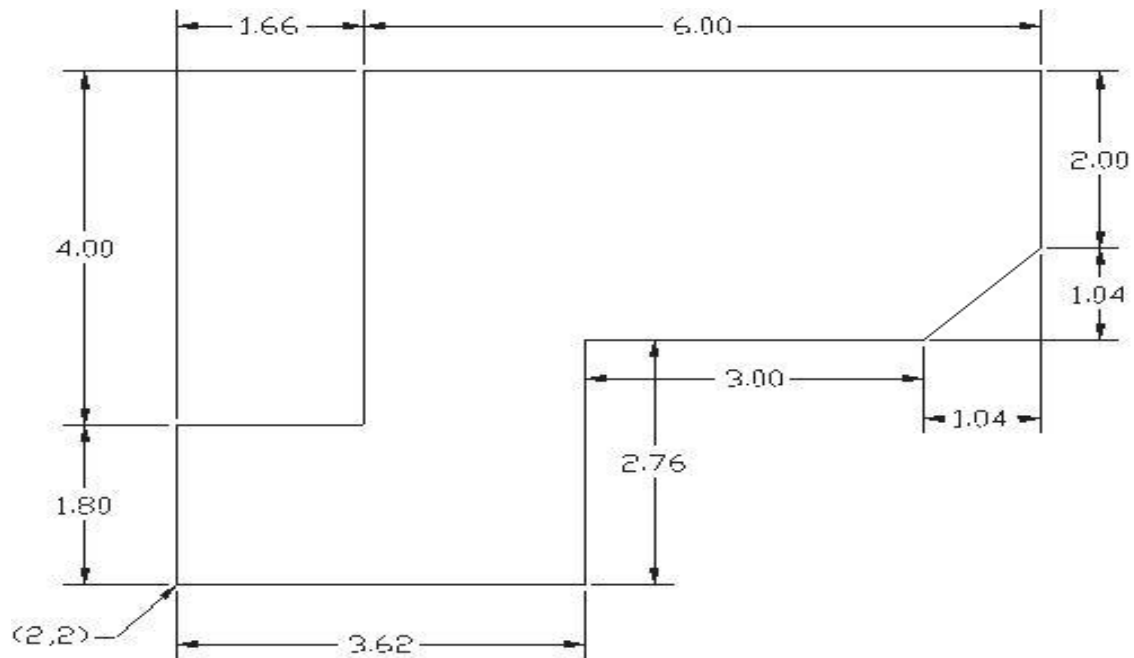
Absolute Cartesian coordinates specify a point's exact distance from the origin point of the coordinate system, which is represented as (0,0). The absolute X and Y coordinates are signed numbers.



RELATIVE METHOD: (@X,Y)

Relative Cartesian coordinates specify a point's exact distance from the last point that was entered.

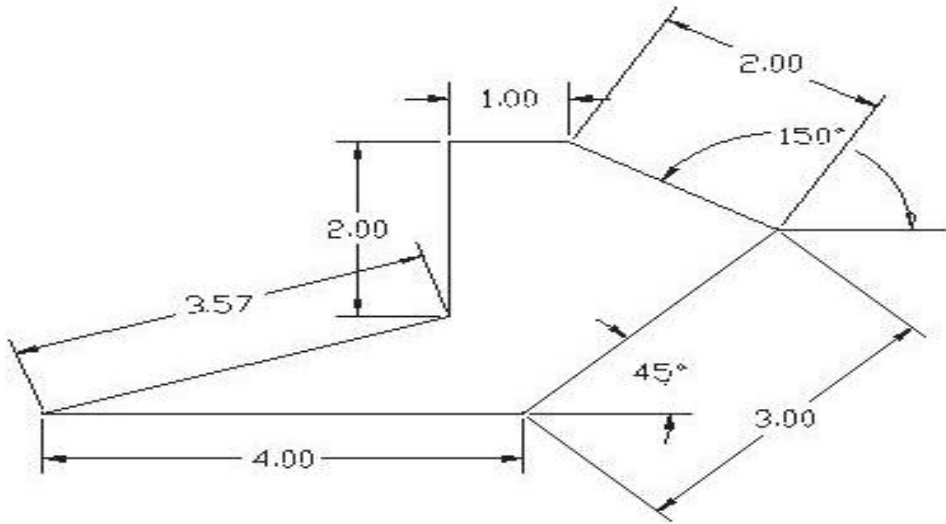
For example, typing @4,2 tells AutoCAD to locate a point that is four X units and two Y units away from the last point entered. The X and Y relative coordinates are signed numbers. Direct distance entry is a shorthand relative coordinate entry method.



POLAR METHOD: (@DISTANCE<ANGLE)

Polar coordinates specify a point's exact location by a distance and angle from the last point that was entered. The distance is always positive and the angle is measured from the positive X axis.

For example, typing @4<45 tells AutoCAD to locate a point that is four units away from the current location and at an angle of 45 degrees from the horizontal.



Polar method

AUTO CAD COMMANDS:

LINE

Line command allows you to create a line where the end points are specified by two dimensional or three dimensional coordinates.

POLYLINE

PLINE command allows you to draw line and arc segments, but from start to end, it is treated as a single object. With PLINE command, you can draw objects even with line width.

ARC

The ARC command allows you to create an arc segment. There are different methods of creating an arc. The different methods of creating an arc are:

- 3 Points
- Centre, Start, Radius
- Start, Centre, End
- Start, Centre, Angle
- Start, Centre, Length
- Start, End, Angle
- Start, End, Direction
- Start, End, Radius

CIRCLE

The CIRCLE command allows you to create a circle. There are four different methods for drawing circles. They are:

- Centre
- Radius/Centre,
- Diameter
- 3 Points (3P)
- 2 Points (2P)
- Tangent, Tangent, Radius

POLYGON

Polygon command creates a regular polygon. You must specify the number of sides of the polygon and whether it is Inscribed or Circumscribed polygon. Now, specify the centre and radius of the polygon circle.

OSNAP

Osnap allow you to *snap* onto a specific object location when you are picking a point. For example, using Osnap you can accurately pick the end point of a line or the center of a circle.

ERASE

This command is used for deleting unwanted objects. You must select the object to be erased.

COPY

The copy command is used for making copies of selected objects. The object to be copied must be selected and the base point must be specified and the copy can be dragged and placed at the required position.

MOVE

This command is used for moving selected objects. The object to be moved must be selected and its base point must be specified then we can drag it to the required location

ROTATE

This command is used for rotating selected objects. To rotate an object, first select it and specify a base point. Now enter an angle value or specify a second point to rotate the object.

Entering a positive angle value rotates the objects counterclockwise or clockwise, depending on the Direction Control setting and Drawing Units dialog box. The plane of rotation and the direction of the zero angles depend on the orientation of the user coordinate system.

MIRROR

The MIRROR command is used for creating mirror images of selected objects. You must first select the object to be mirrored and then specify the axis along which it is mirrored, to create an mirror image of the selected object. This is a very useful command for drawing symmetrical objects about a particular axis.

OFFSET

Offset an object to create a new object whose shape parallels the shape of the original object. Offsetting a circle or an arc creates a larger or smaller circle or arc, depending on which side you specify for the offset.

POLAR ARRAY

This command creates an array of the selected object around a centre point. You must specify the centre point of the array, the total number of items and the angle to fill for creating a polar array.

RECTANGULAR ARRAY

This command creates an array of the selected objects defined by the number of rows and columns and the offset between them.

EXTEND

The EXTEND command elongates the selected objects to a specified boundary.

TRIM

The TRIM command trims off an object using cutting edges defined by other objects. Here the user is required to select object(s) to define cutting edge(s) then select the object to be trimmed; the selected side of the object is removed based on the side of selection of the object to be trimmed relative to the cutting edge.

BREAK

The BREAK command is used to remove only parts of an object. You must specify the first and second points between which the object must be deleted.

FILLET

FILLET rounds of the edges of two arcs, circles, elliptical arcs, lines, polylines, rays, splines or xlines with an arc of a specified radius. Rounds along the edges cannot be created with zero radius.

CHAMFER

CHAMFER command draws a line at the corner between two selected lines. If the lines do not intersect, it extends the lines. If they intersect, the lines extending beyond the chamfer line can be trimmed or left as it is, by the trim mode.

LENGTHEN

The LENGTHEN command is used to increase/decrease the length of AutoCAD objects.

DIVIDE

This command places evenly spaced point objects or blocks along the length or perimeter of an object. You can select only one entity at a time to be divided. The entities that can be selected are: line, arc, circle, polyline and spline.

ZOOM ALL

This zoom to display the entire drawing in the current viewport. The display shows all the entities even if the drawing extends outside the drawing limits.

ZOOM WINDOW

ZOOM WINDOW commands to display an area specified by two diagonally opposite corner points of a rectangle window.

ZOOM EXTENTS

This zoom to display the drawing extents. If the drawings are small in the existing limits, then there is an enlarging effect on the screen. If the drawing occupies the complete area of the limits, then there are many not being much difference between ZOOM ALL and ZOOM EXTENTS.

DIMLINEAR

This command specifies the linear distance between two selected points.

DIMALIGNED

This command is used for specifying the linear distance (exact dimensional value) between two points which are inclined at an angle.

DIMANGULAR

This command is used for obtaining the angle two selected points. The three types are:

- 1) Arc Selection
- 2) Circle Selection
- 3) Line Selection
- 4) Three Point Selection

DIMRADIUS

This command gives the value of radius of the specified circle, arc or fillet. A radial dimension consists of a radius dimension line with an arrowhead at the arc or circle end.

DIMCENTER

This command marks the centre point of the selected arc or circle. The center mark cannot be updated, so set the required style before executing this command.

LINETYPE

LINETYPE greatly improves the readability of technical drawings. You can make important features stand out with bold line weights.

LAYER

Layer is an organizing tool. Layers are like one, where you can keep various types of information. It's always a good idea to keep notes and reference symbols about each element of the drawing as well as drawing dimensions on a new layer.

DISTANCE

This command lists the distance between the selected points and also lists out the angle in current plane and 3D angle from the current plane, based on the direction of picking the two points. It also lists the projected lengths on the three axes called the delta length.

TEXT

TEXT command creates text objects with specified height and orientation. Text objects can be created with a variety of character patterns call style. This command allows you to type a single line or multiple lines of text.

MTEXT

MTEXT creates paragraph that fit within a nonprinting text boundary. The user specified text boundary determines the width of the paragraph and the justification of the text within the paragraph. Each multiline text object is a single object, regardless of the number of lines it contains.

HATCH

Hatches are shaded pattern which generally represent cross-section of a mechanical component. Using BHATCH command the required area, type of hatch, hatch angle and scale can be specified.

PROPERTIES

The PROPERTIES command displays the Properties palette. The Properties palette is the main method for viewing and modifying the properties of AutoCAD objects.

VPORTS

Viewports are areas that display different views of your model. As you work on the Model tab, you can split the drawing area into one or more adjacent rectangular views known as *model space viewports*. In large or complex drawings, displaying different views reduces the time needed to zoom or pan in a single view. Also, errors you might miss in one view may be apparent in the others.

EXTRUDE

Solids can be created by extruding selected objects. Use the EXTRUDE command to create a solid or surface from a common profile of an object.

REVOLVE

REVOLVE command can create a solid or surface by revolving open or closed objects about an axis. The revolved objects define the profile of the solid or surface.

PYRAMID

This command can create a solid pyramid. The number of sides for a pyramid, from 3 to 32, can be defined. The axis endpoint defines the length and orientation of the pyramid.

CREATE LAYOUTS

A layout stimulates a sheet of paper and provides a predictable plotting setup for a layout, you can create and position view post objects and you can add a title block or other objects geometry.

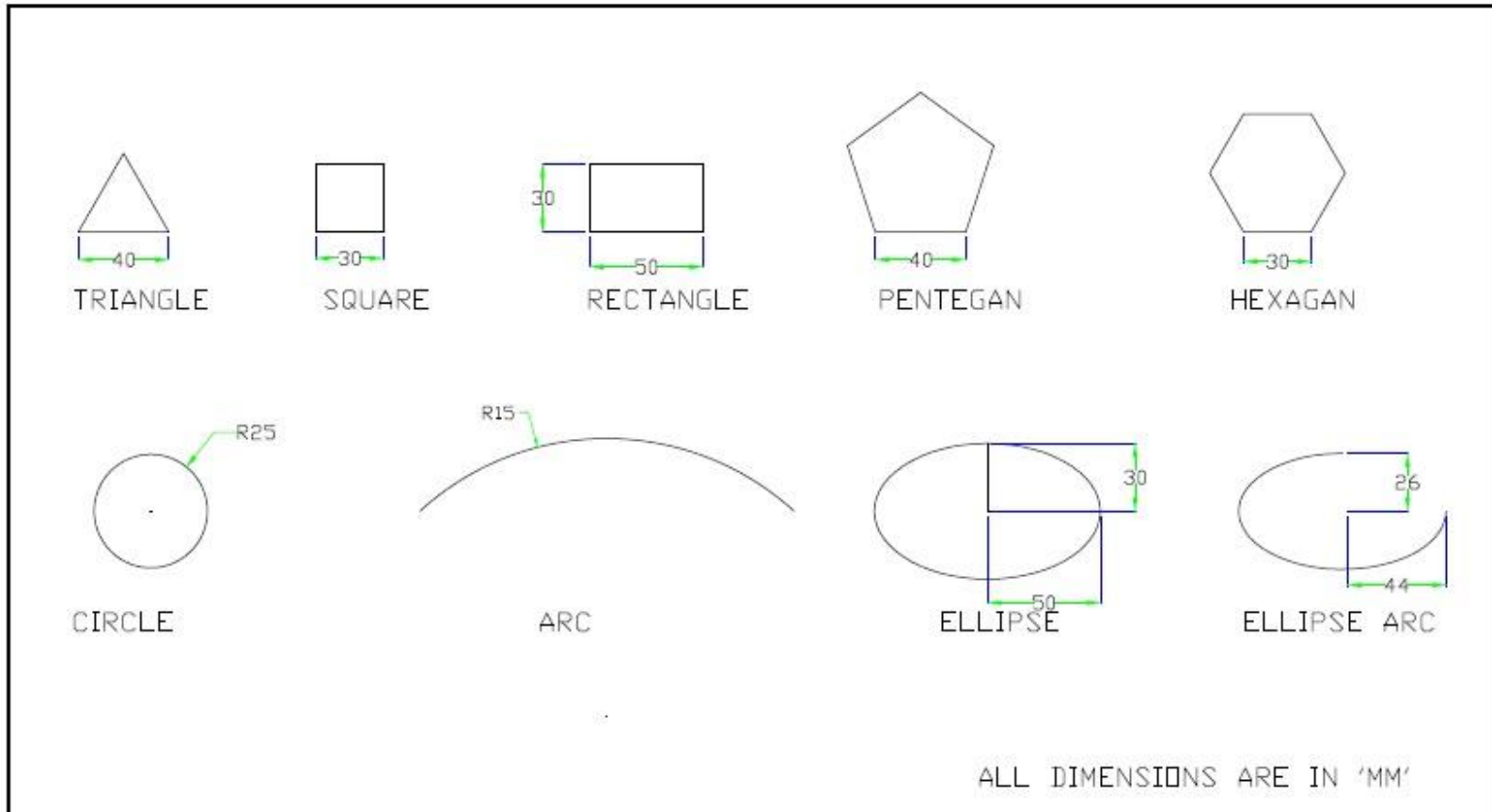
PLOT DRAWINGS

Once you have completed a drawing you can plot the drawing on paper or create a file for use with another application. In either case, you select the plot settings.

EX.NO:01

DATE:

CREATION OF SIMPLE FIGURES



EX NO: 01

DATE:

**STUDY OF CAPABILITIES OF SOFTWARE FOR DRAFTING
AND MODELING**

AIM:

To create simple objects like polygon and general multi-line figures using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007 software.

COMMANDS USED:

Limits, Zoom, Line, Arc, Circle, co-ordinate systems, Dim linear.

PROCEDURE:

- Limits are set for A4 standard drawing size. Margins are drawn using lines.
- Using Line command and appropriate co-ordinate system, the given figures are drawn and aligned
- The drawn figures are dimensioned using respective dim command.
- Finished work sheet is saved and hard copy is taken.

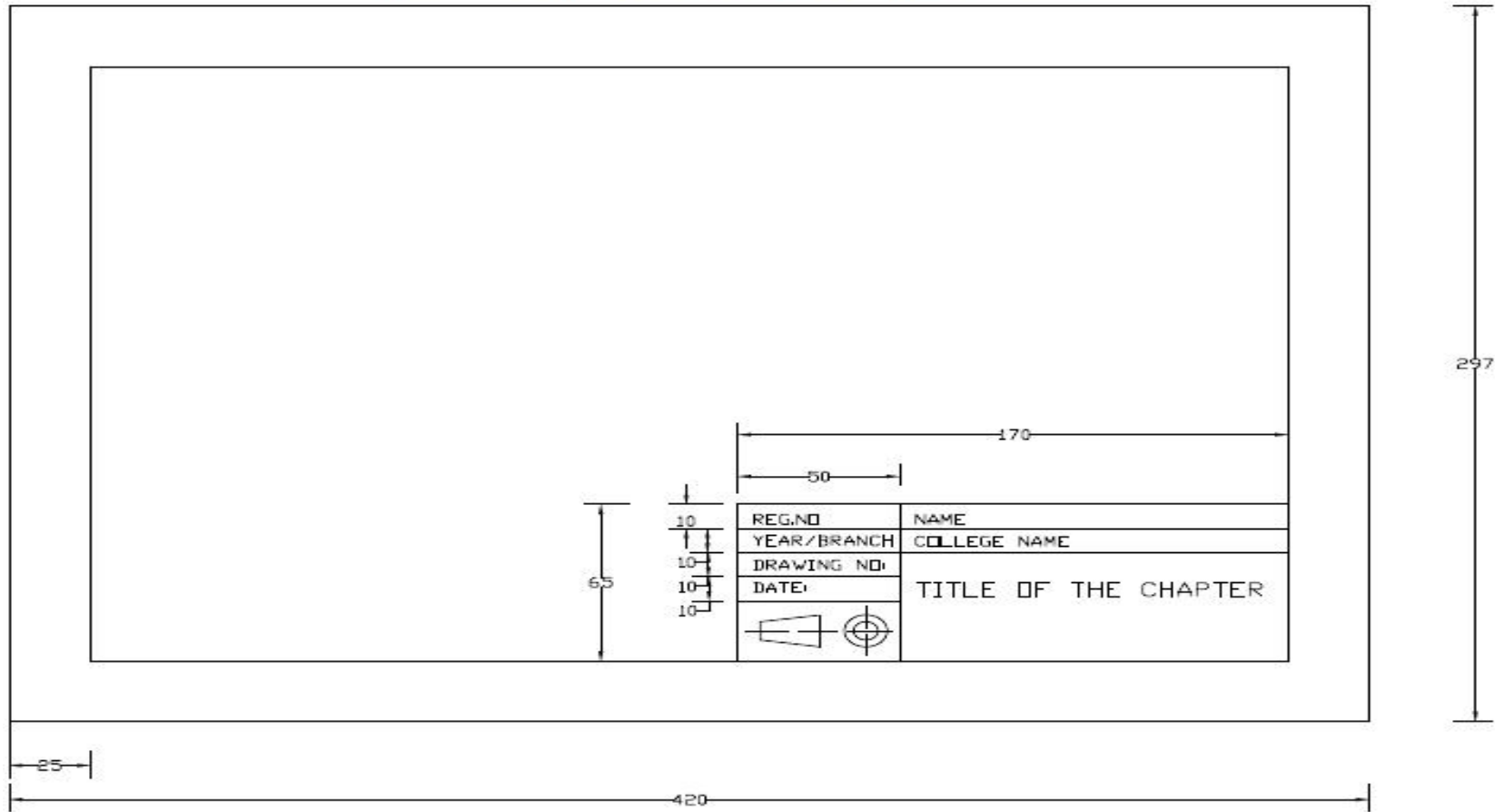
RESULT:

The given simple figures like polygon and general multi-line figures are drawn Using Auto CAD software.

Drawing of A Title Block With Necessary Text And Projection

EX NO : 2
DATE

TITLE BLOCK WITH NECESSARY TEXT AND PROJECTION SYMBOL



ALL DIMENSIONS ARE IN 'MM'

EX NO: 02

DATE:

**DRAWING OF A TITLE BLOCK WITH NECESSARY
TEXT AND PROJECTION**

AIM:

To draw a Title block for the given dimensions with necessary text and projection.
Using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

COMMANDS USED:

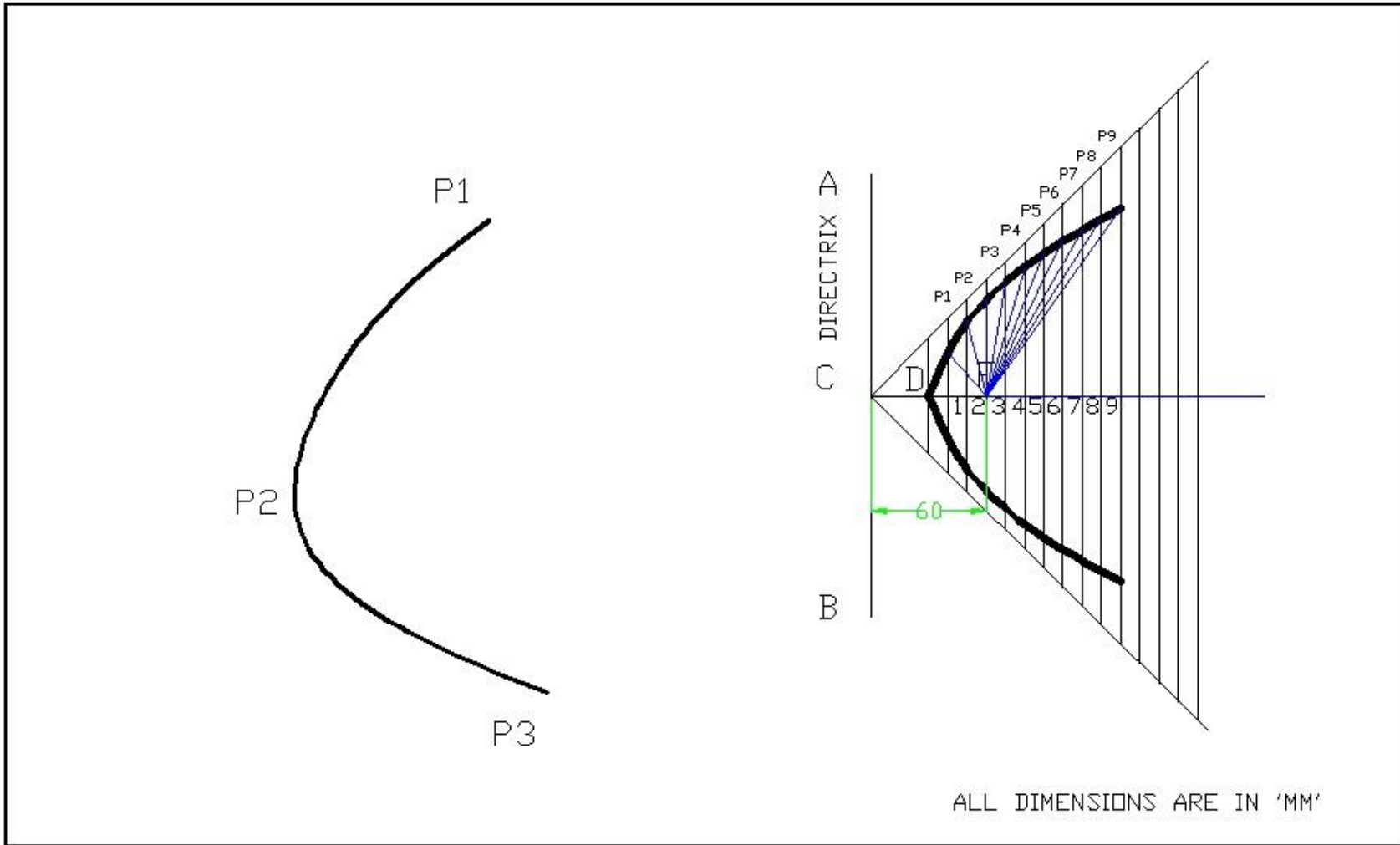
Limits, Zoom, Line, Arc, Circle, co-ordinate systems, Dim linear, Copy, Trim, Circle, Dimensions

PROCEDURE:

- Limits are set for standard drawing size. Margins are drawn using lines.
- Using Line command and appropriate co-ordinate system, the given figures are drawn and aligned
- Using Line, Circle, poly line ,trim commands, and draw a Title Block with necessary text and projection are drawn
- The drawn figures are dimensioned using respective DIM command.
- Title Block is drawn for the given dimension.
- Finished work sheet is saved and hard copy is taken.

RESULT:

Thus the Title block was drawn for the given dimensions with all necessary text and projection.



EX NO: 03

DATE:

DRAWING OF CURVES LIKE PARABOLA, SPIRAL, INVOLUTE
USING BSPLINE OR CUBIC LINE

AIM:

To create special curves like parabola, spiral, involute (square and Hexagon) using Bspline or Cubic Spline using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

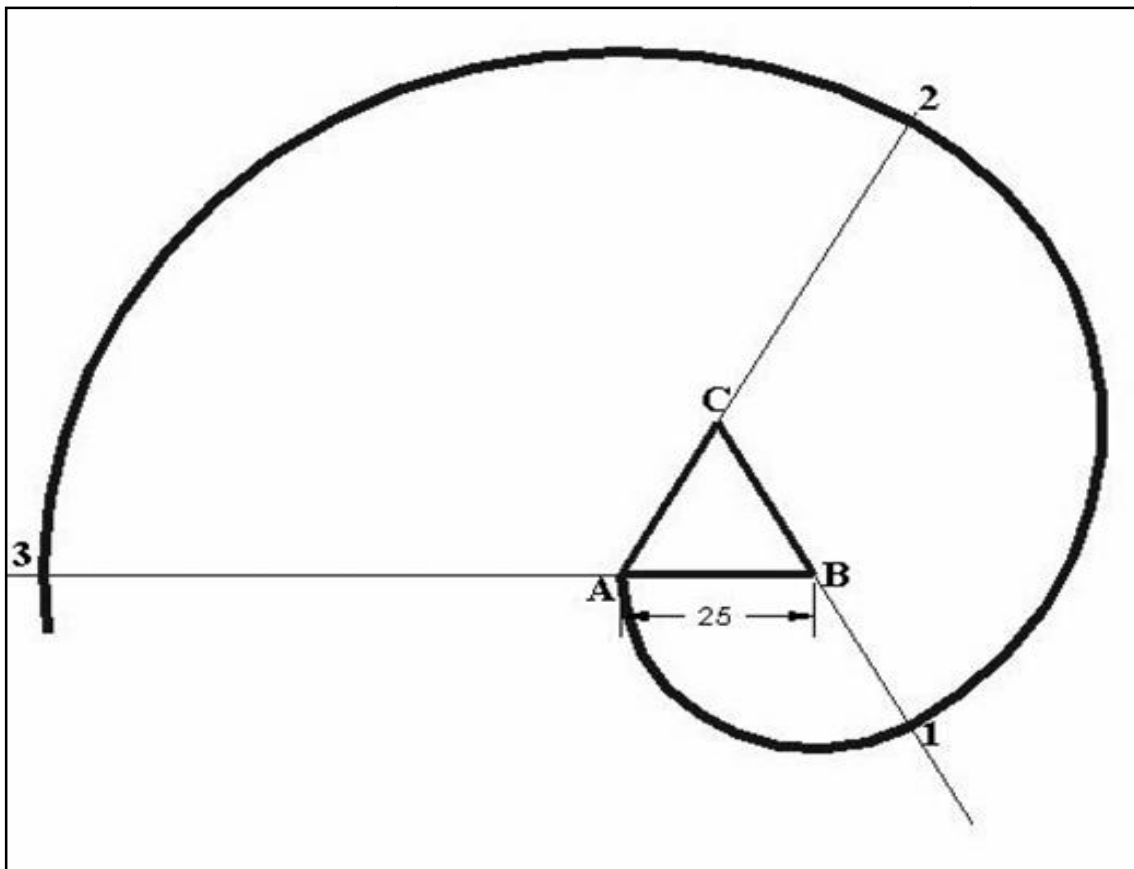
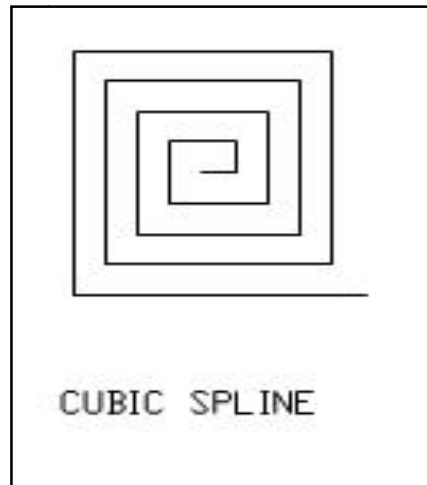
COMMANDS USED:

Limits, Zoom, Line, Polygon, Arc, Circle, Offset, Copy, Move, Trim, Pline, Pedit, DIM.

PROCEDURE:

- Limits are set for standard drawing size. Margins and title block are drawn using lines.
- From the given data, the Loci points of Parabola are found and joined using Pline and then fitted using Pedit command
- From the given data, the Loci points of square and hexagonal involutes are found and joined using arc command.
- From the given data, the Loci points of Spiral are found using concentric circles or arc method, and joined using Pline and then fitted using Pedit command.
- Drawn curves are dimensioned appropriately.
- Finished work sheet is saved and hard copy is taken.

CUBIC SPLINE AND INVOLUTE

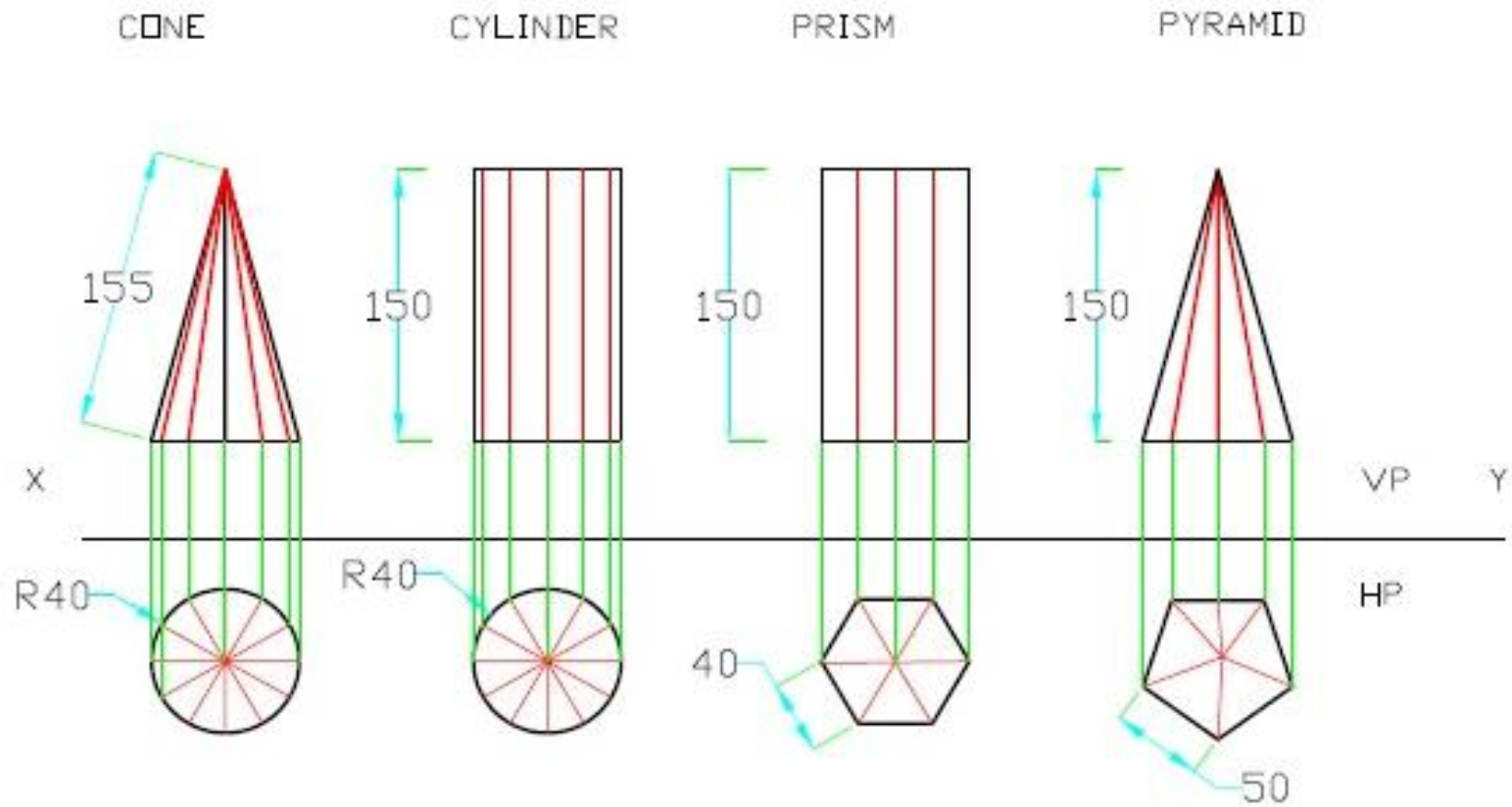


All dimensions are in 'mm'

RESULT:

The special curves like parabola, spiral, involute (square and Hexagon) using Bspline or Cubic Spline are drawn using Auto CAD spline.

FRONT VIEW & TOP VIEW OF SIMPLE SOLIDS (cone,cylinder,prism and pyramid)



EX NO: 04

DATE:

**DRAWING OF FRONT VIEW ANT TOP VIEW OF SIMPLE SOLIDS LIKE PRISM,
PYRAMID, CYLINDER, CONE, AND DIMENSIONINS**

AIM:

To draw front and top view of given simple solids (cone, cylinder, prism and pyramid) using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

COMMANDS USED:

Limits, Zoom, Line, Polygon, Arc, Circle, Offset, Copy, Move, Trim, Layer, DIM, Mtext.

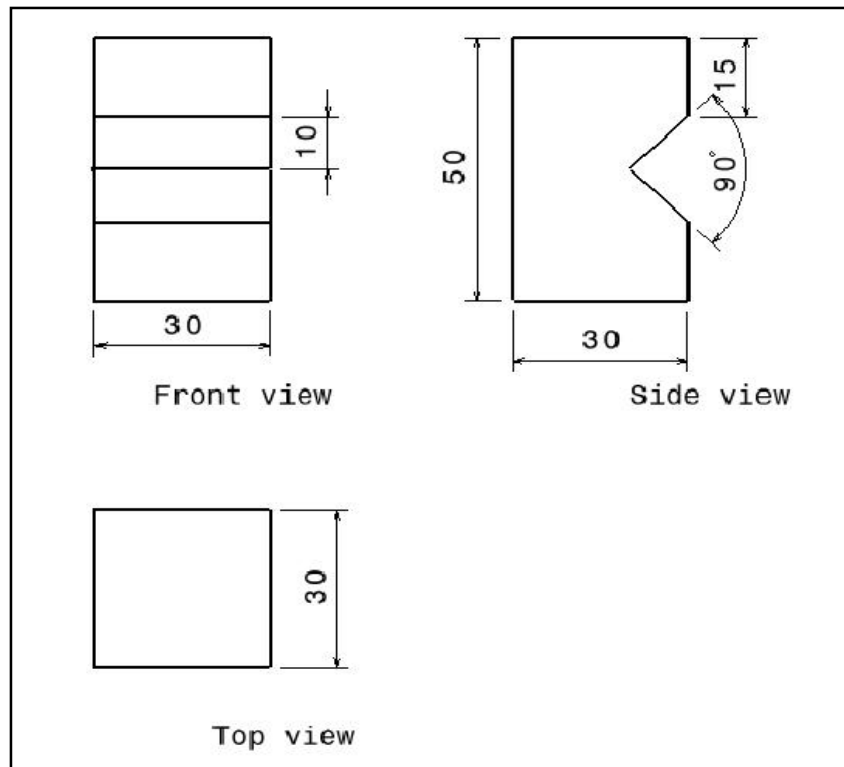
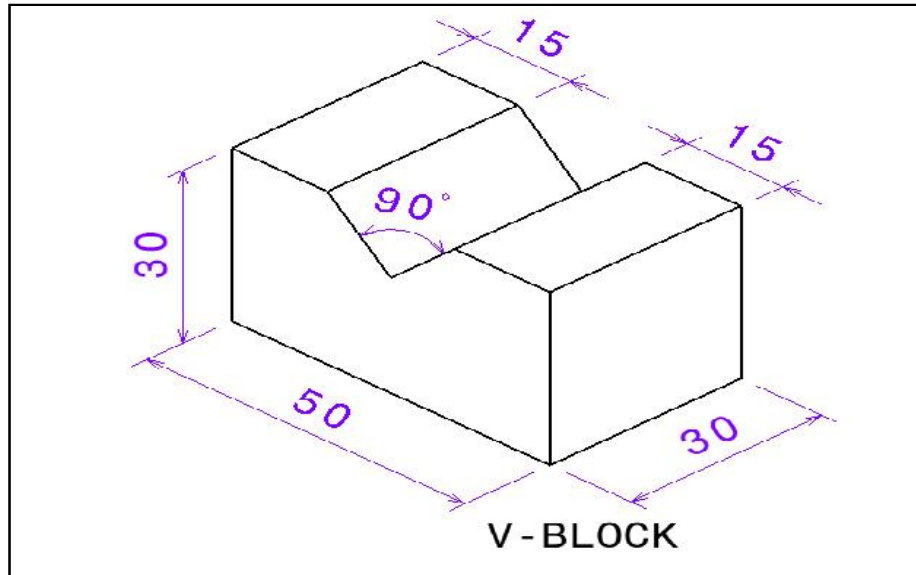
PROCEDURE:

- Limits are set for standard drawing size. Margins and title block are drawn using lines.
- Using Line, Circle, Polygon commands, front and top view of given simple Solids are drawn.
- Layer is defined (for line type, line weight and colour) separately for visible, hidden, axis and dimension lines and applied.
- Sectional views are drawn and section is created using Hatch command.
- Drawn solids are named and dimensioned accordingly.
- Finished work sheet is saved and hard copy is taken.

RESULT:

The front and top views of the given simple solids (cylinder, cone, prism and pyramid) are drawn using Auto CAD software.

Drawing front view, top view, and side view of object from the given V-BLOCK



All dimensions are in 'mm'

EX NO: 05

DATE:

DRAWING FRONT VIEW, TOP VIEW, AND SIDE VIEW OF OBJECT FROM THE

GIVEN PICTORIAL VIEWS

(V-BLOCK, BASE MIXIE AND SIMPLE STOOL)

AIM:

To draw front and top view of given simple solids (*v-block, Base mixie, simple stool*) using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

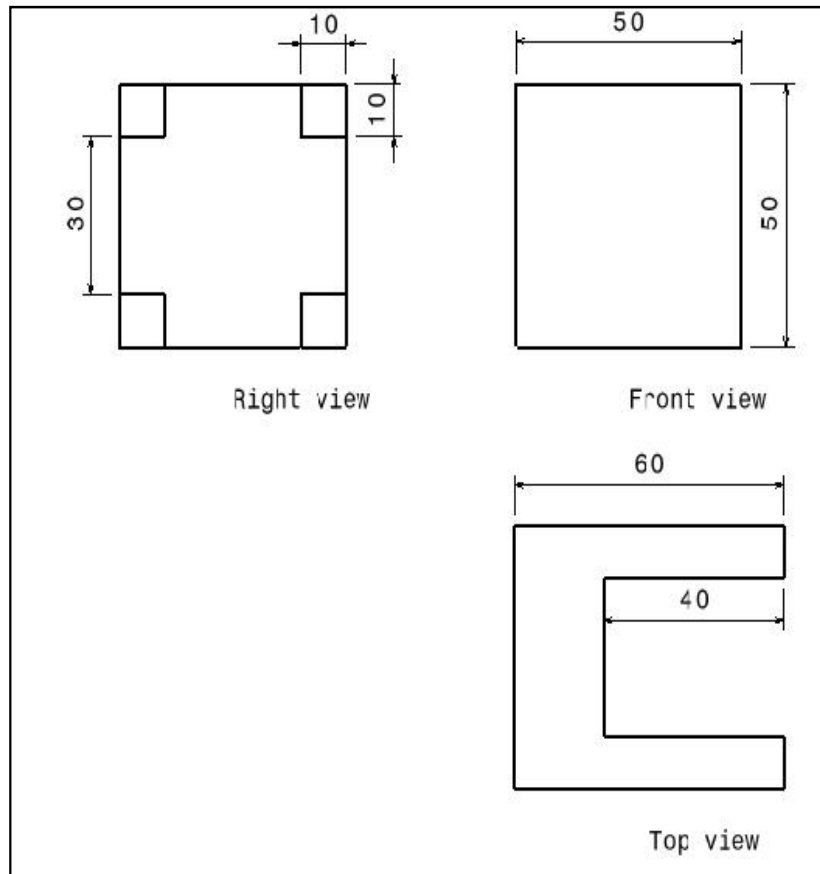
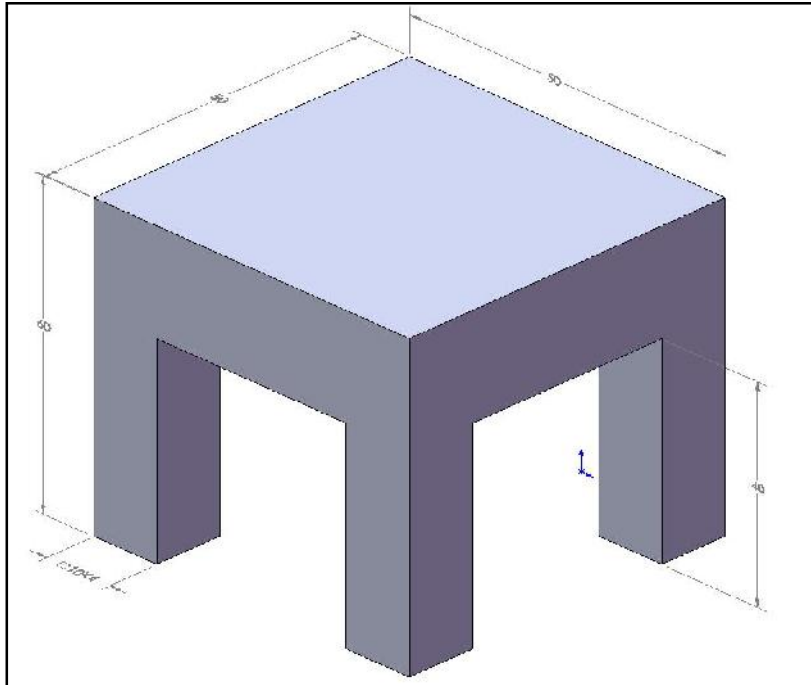
COMMANDS USED:

- Limits,
- Zoom,
- Line,
- Polygon,
- Arc,
- Circle,
- Offset,
- Copy,
- Move,
- Trim,
- Layer,
- DIM,
- Mtext.

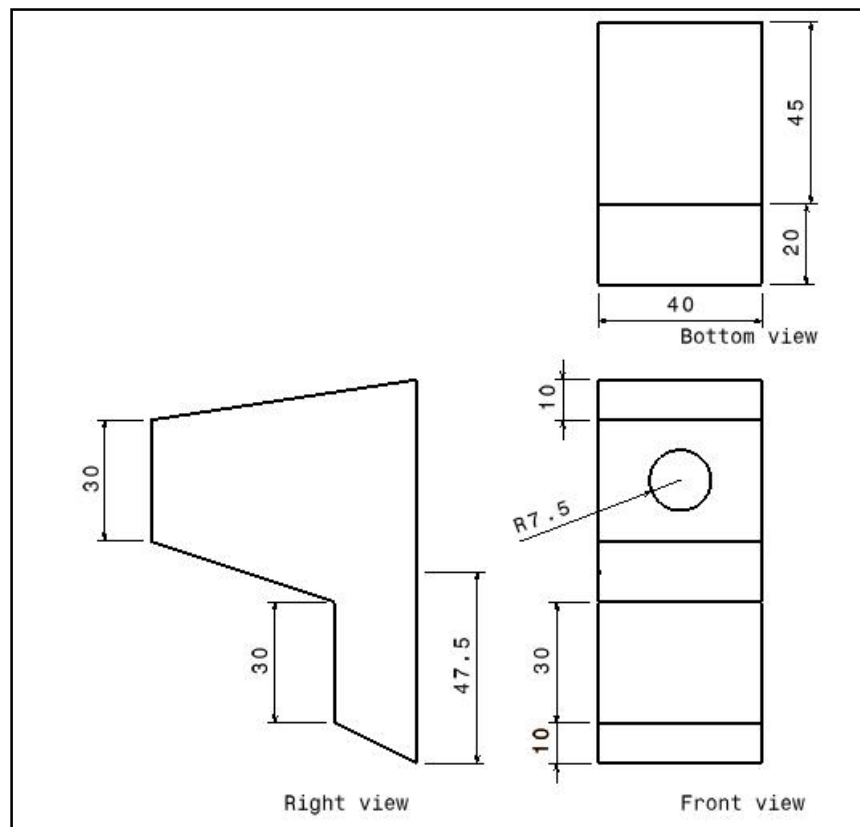
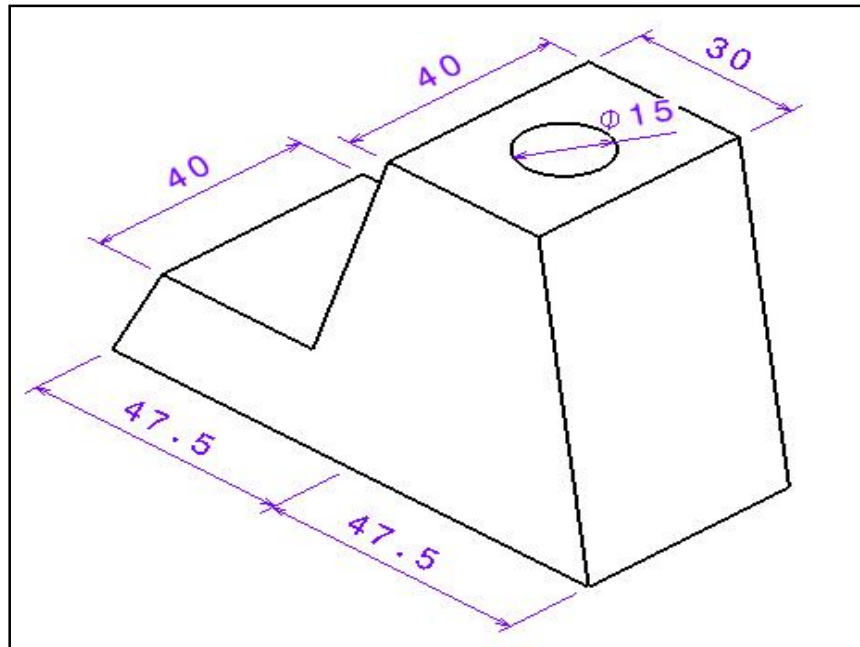
PROCEDURE:

- Limits are set for standard drawing size. Margins and title block are drawn using lines.
- Using Line, Circle, Polygon commands, front and top view of given simple solids are drawn
- Layer is defined (for line type, line weight and colour) separately for visible, hidden, axis and dimension lines and applied.
- Sectional views are drawn and section is created using Hatch command.
- Drawn solids are named and dimensioned accordingly.
- Finished work sheet is saved and hard copy is taken.

Drawing front view, top view, and side view of object from the given SIMPLE STOOL



Drawing front view, top view, and side view of object from the given MIXIE

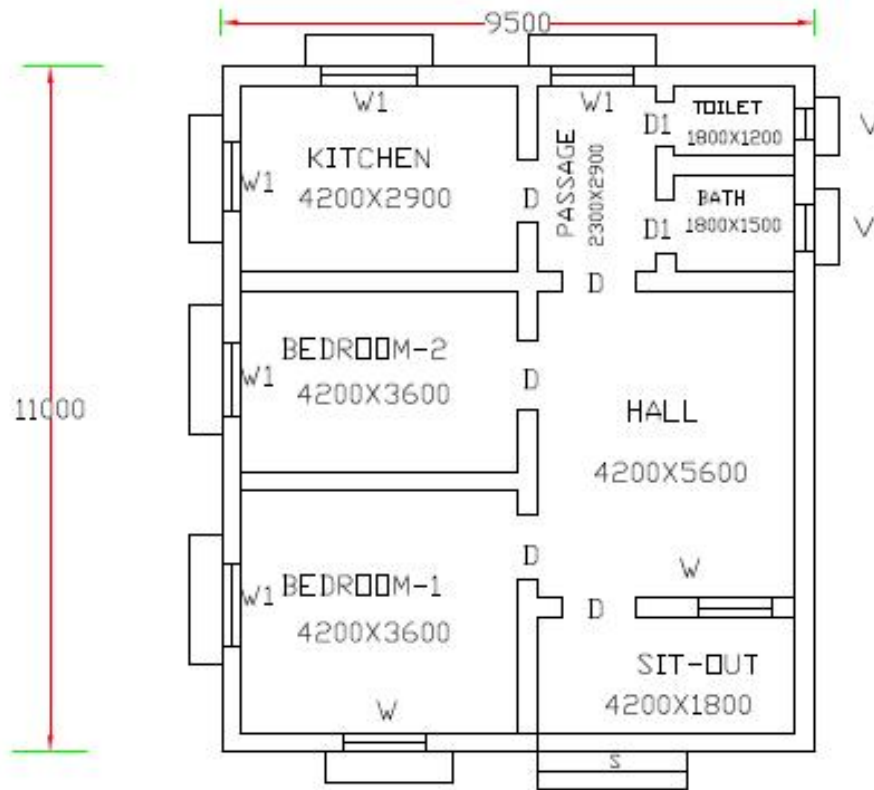


All dimensions are in 'mm'

RESULT:

The front and top views of the given simple solids (*v-block, Base mixie, and simple stool*) are drawn using Auto CAD software.

PLAN OF RESIDENTIAL BUILDING



ALL DIMENSION ARE IN MM

- S-STEPS
- D-DOOR1000X2100
- D1-DOOR1500X1200
- W-WINDOW1500X1200
- W1-WINDOW1200X1200
- V-VENTILATOR600X450
- WALL THICKNESS-300MM

EX NO: 06

DATE:

DRAWING OF A PLAN OF RESIDENTIAL BUILDING

AIM:

To create the plan of given residential building using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

COMMANDS USED:

Limits, Zoom, Line, Offset, Copy, Move, Multiline.

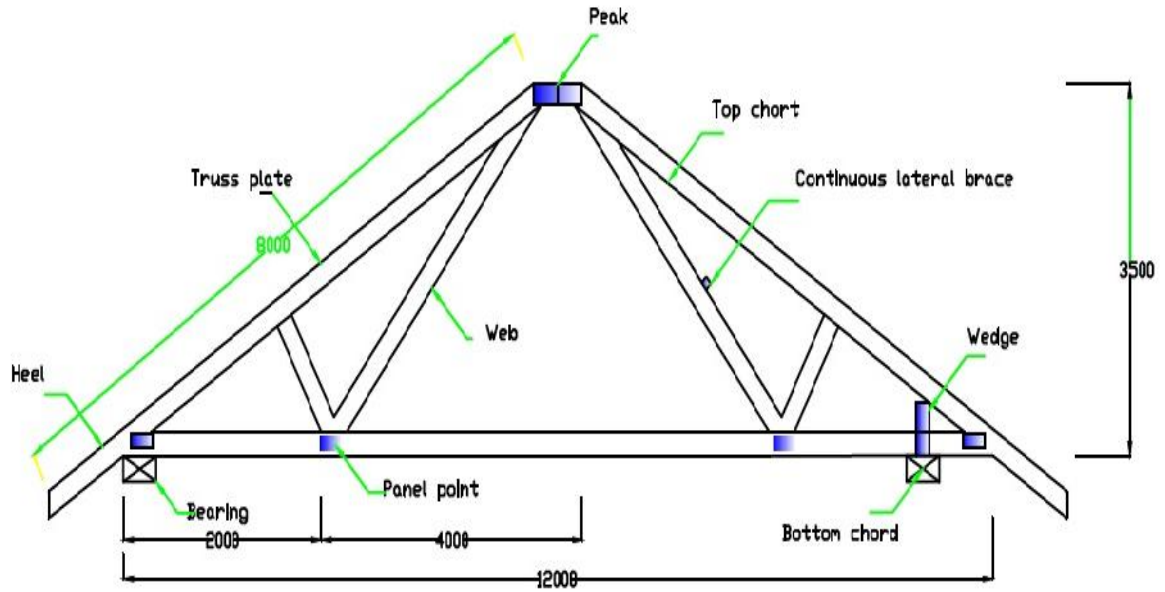
PROCEDURE:

- Limits are set for standard drawing size. Margins and title block are drawn using lines.
- Using Multiline command, plan of the given residential building is drawn.
- The plan is dimensioned and the rooms are named accordingly.
- Finished work sheet is saved and hard copy is taken.

RESULT:

The plan of the given residential building is drawn using Auto CAD software.

Simple Steel Truss



EX NO: 07

DATE:

DRAWING OF A SIMPLE STEEL TRUSS

AIM:

To draw simple trusses using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

COMMANDS USED:

Limits, Zoom, Line, Offset, Copy, Move, Trim, Extend, DIM.

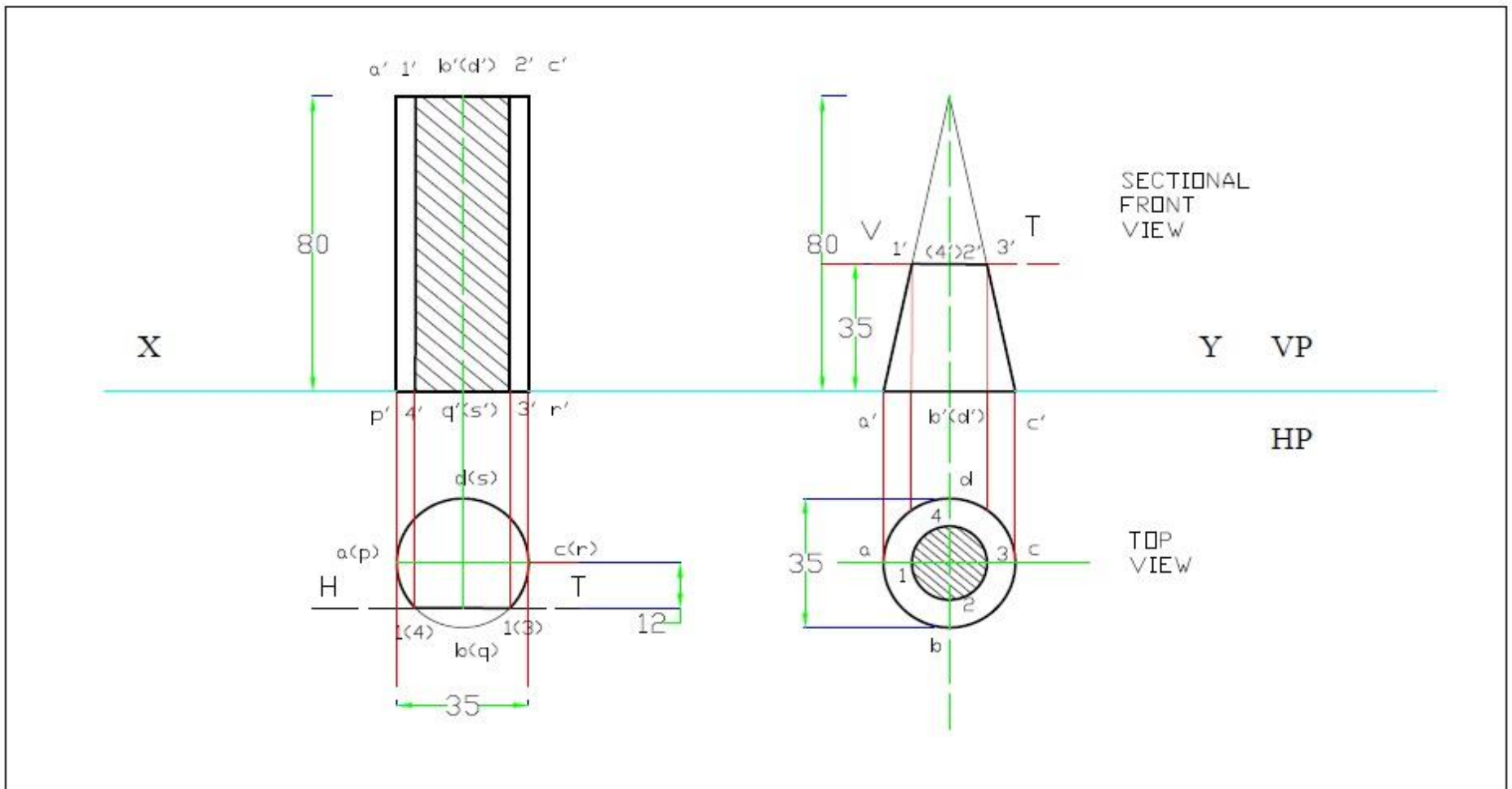
PROCEDURE:

- Limits are set for standard drawing size. Margins and title block are drawn using lines.
- Using appropriate commands, simple trusses with combinations of King and Queen posts are drawn.
- Drawn figure is dimensioned accordingly.
- Finished work sheet is saved and hard copy is taken.

RESULT:

The plan of simple trusses is drawn using Auto CAD software.

SECTIONAL VIEW OF SOLIDS(cylinder& cone)



EX NO: 08

DATE:

**DRAWING SECTIONAL VIEWS OF PRISM, PYRAMID, CYLINDER
AND CONE**

AIM:

To draw sectional view of given simple solids (cylinder, cone, prism and pyramid) using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

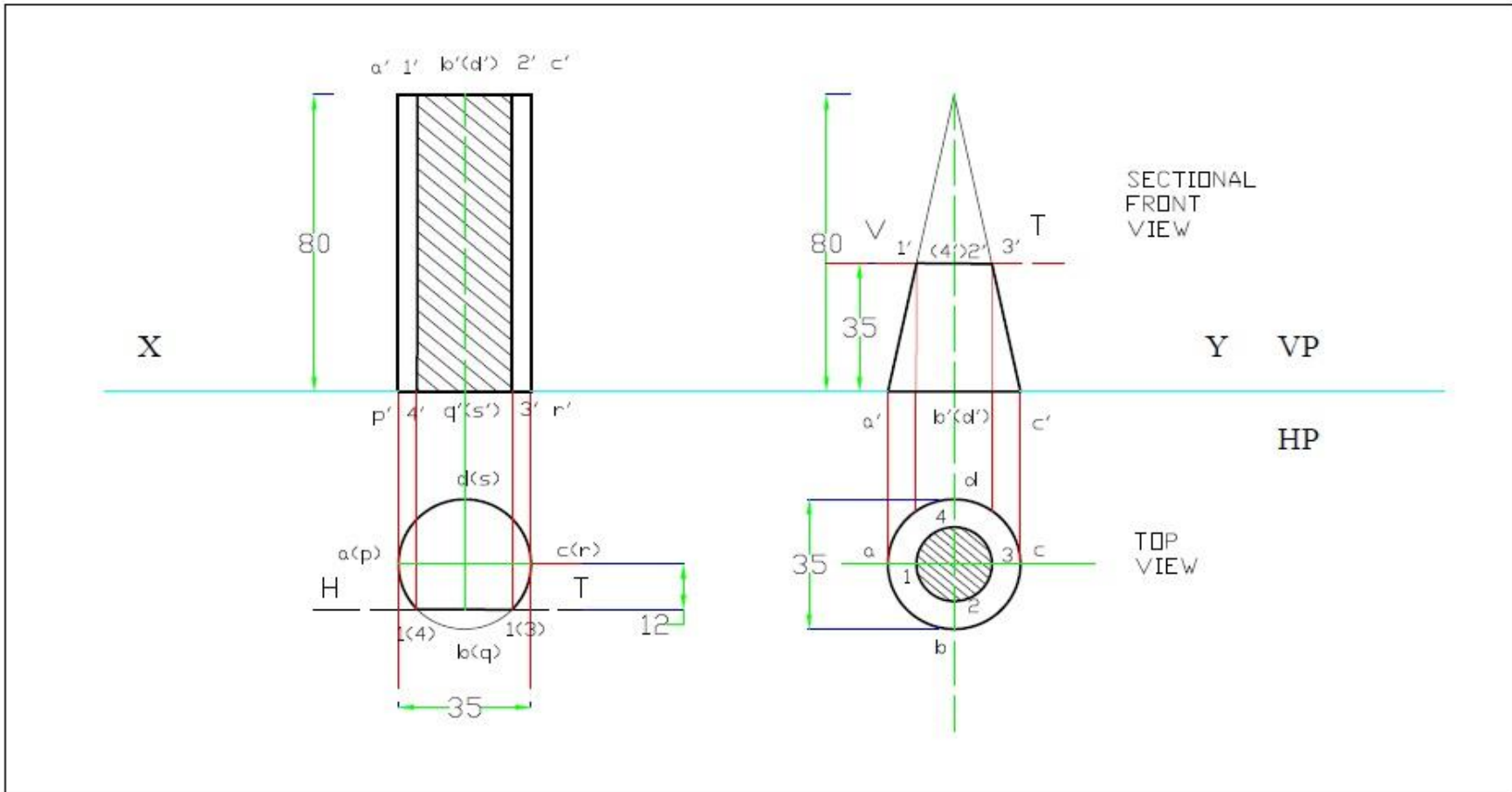
COMMANDS USED:

Limits, Zoom, Line, Polygon, Arc, Circle, Offset, Copy, Move, Trim, Layer, DIM, Mtext.

PROCEDURE:

- Limits are set for standard drawing size. Margins and title block are drawn using lines.
- Using Line, Circle, Polygon commands, front and top view of given simple solids are drawn
- Layer is defined (for line type, line weight and colour) separately for visible, hidden, axis and dimension lines and applied.
- Sectional views are drawn and section is created using Hatch command.
- Drawn solids are named and dimensioned accordingly.
- Finished work sheet is saved and hard copy is taken.

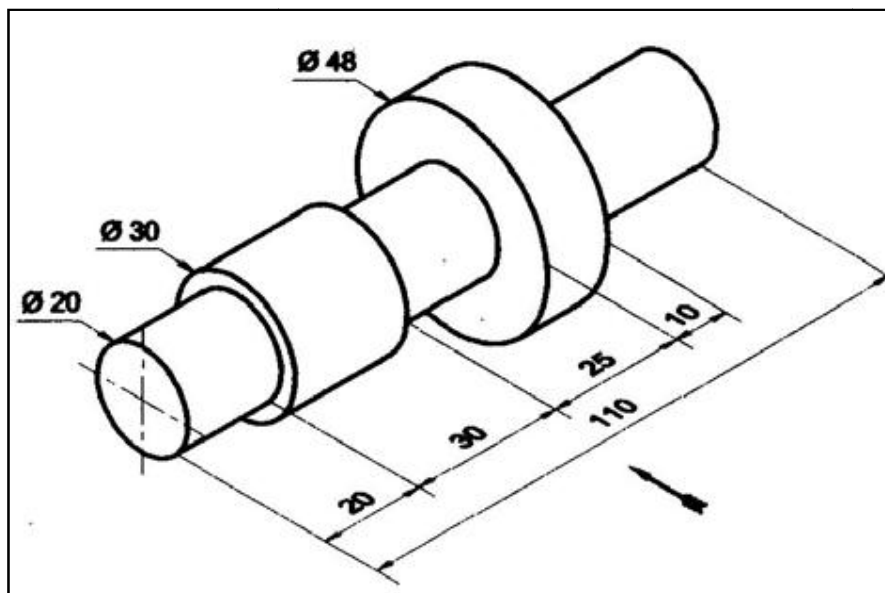
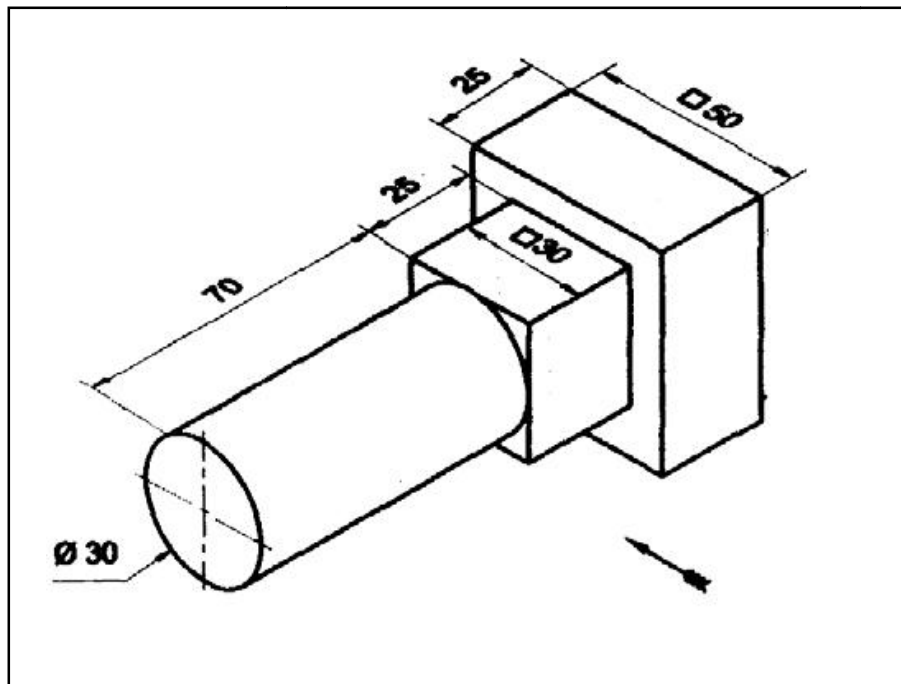
SECTIONAL VIEW OF SOLIDS(cylinder& cone)



RESULT:

The sectional view of given simple solids (cylinder, cone, prism and pyramid) are drawn using Auto CAD software.

Isometric Projection of Simple Objects



All dimensions are in 'mm'

EX NO: 09

DATE:

DRAWING ISOMETRIC PROJECTION OF SIMPLE OBJECTS

AIM:

To draw the isometric view of given simple objects using Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

COMMANDS USED:

Limits, Zoom, Line, Polygon, Arc, Circle, Offset, Copy, Move, Trim, Layer, DIM, Mtext

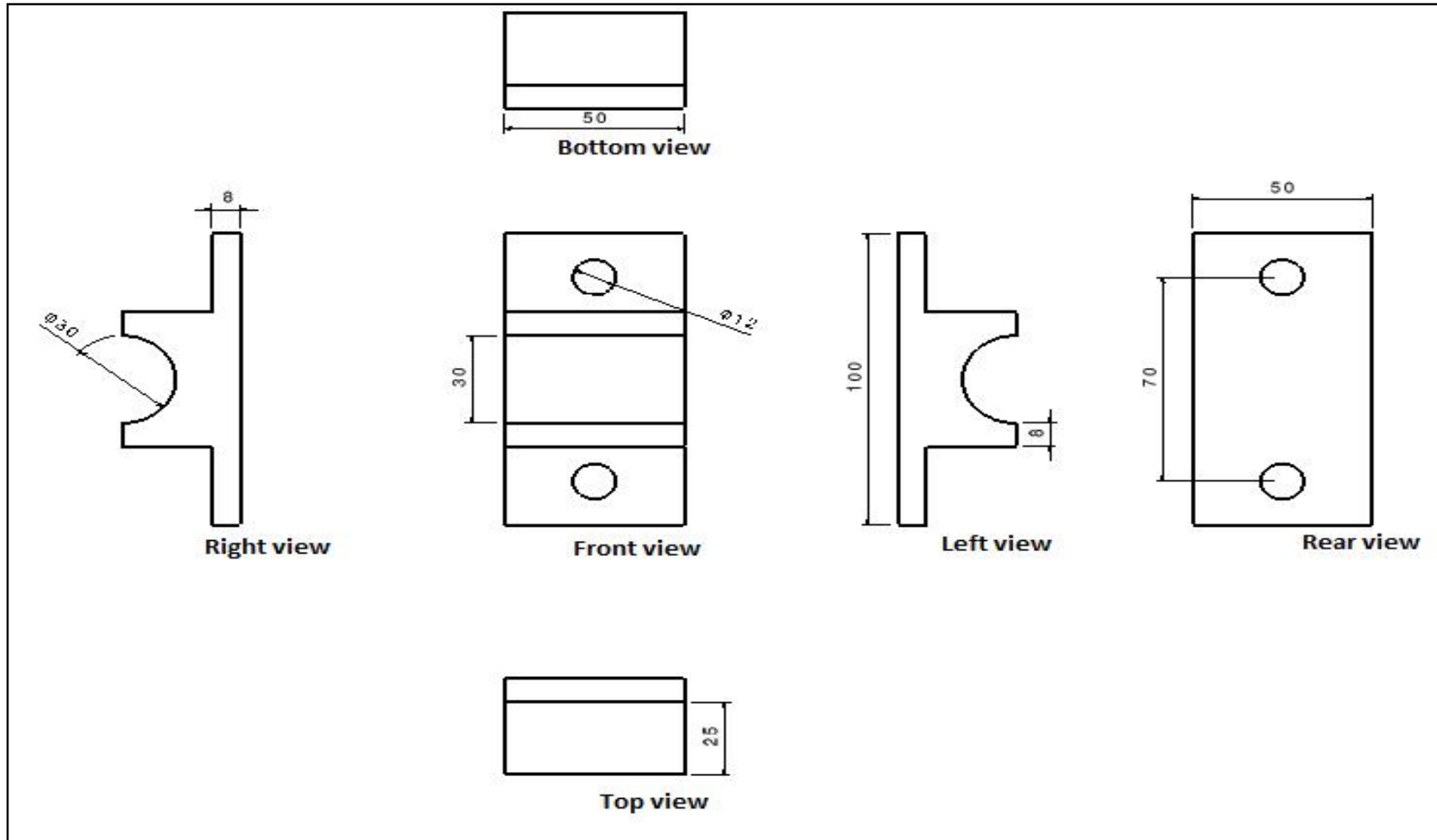
PROCEDURE:

- Limits are set for standard drawing size. Margins and title block are drawn using lines.
- Using Line, Circle, Polygon commands, front and top view of given simple solids are drawn.
- Snap is changed from Rectangular to Isometric to help draw the Isometric view. F5 key is used to invoke/toggle between the different Isoplanes (Left, Top, Right) accordingly.
- Given objects are taken and drawn in Isometric.
- Drawn solids are named and dimensioned accordingly.
- Finished work sheet is saved and hard copy is taken.

RESULT:

The Isometric views of the given simple objects are drawn using Auto CAD software.

Creation Of 3-D Models of Simple Objects And Obtaining 2-D Multi-View



EX NO: 10

DATE:

CREATION OF 3-D MODELS OF SIMPLE OBJECTS

AIM:

To create 3-D Models of given simple solids and obtain 2-D multi view drawings
Auto CAD software.

SOFTWARE USED:

- Auto CAD 2007

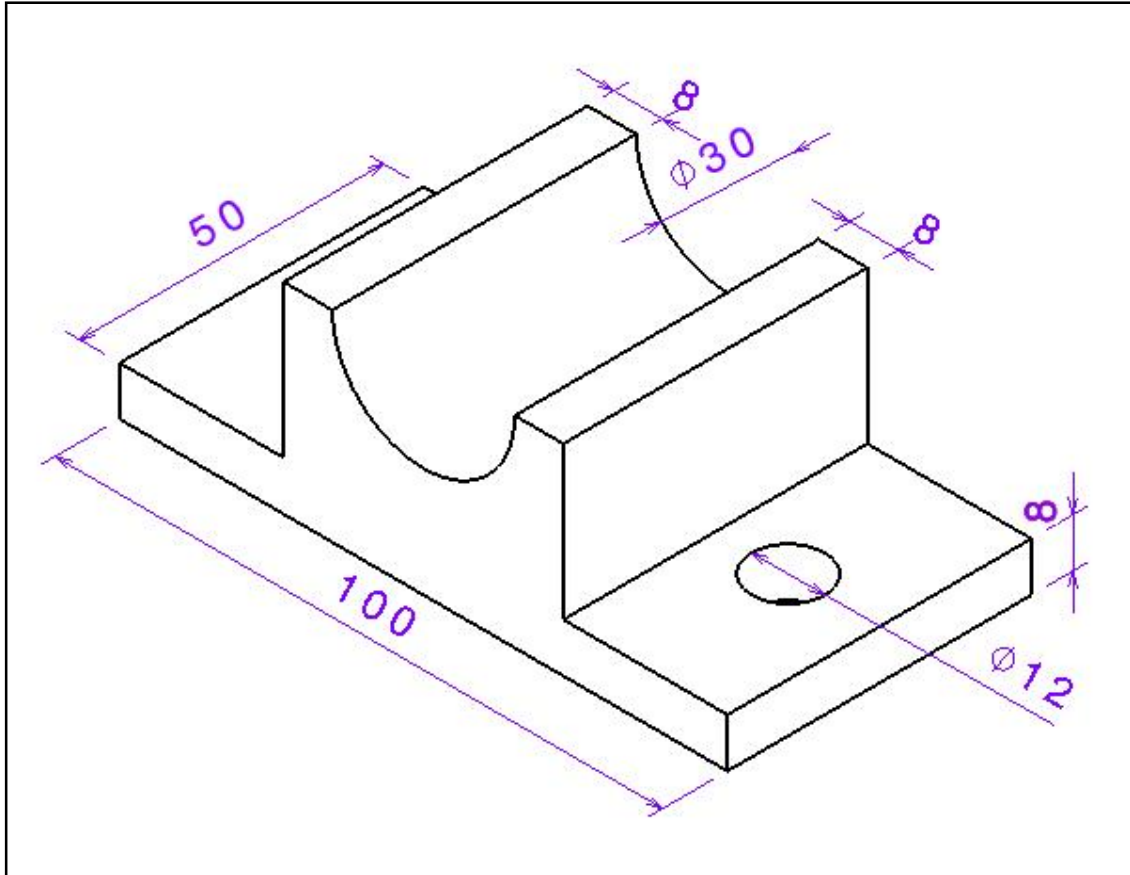
COMMANDS USED:

- Zoom,
- Line,
- Polygon,
- Arc,
- Circle,
- DIM,
- Extrude,
- Revolve.

PROCEDURE:

- Auto CAD 3-D template is opened for creating 3-D models.
- Prisms are created using Extrude command and pyramids by Pyramid command.
- Objects of revolution like cylinder and cone are created using Revolve command.
- Finished work sheet is saved and hard copy is taken.

Creation of 3-D Models of Simple Objects

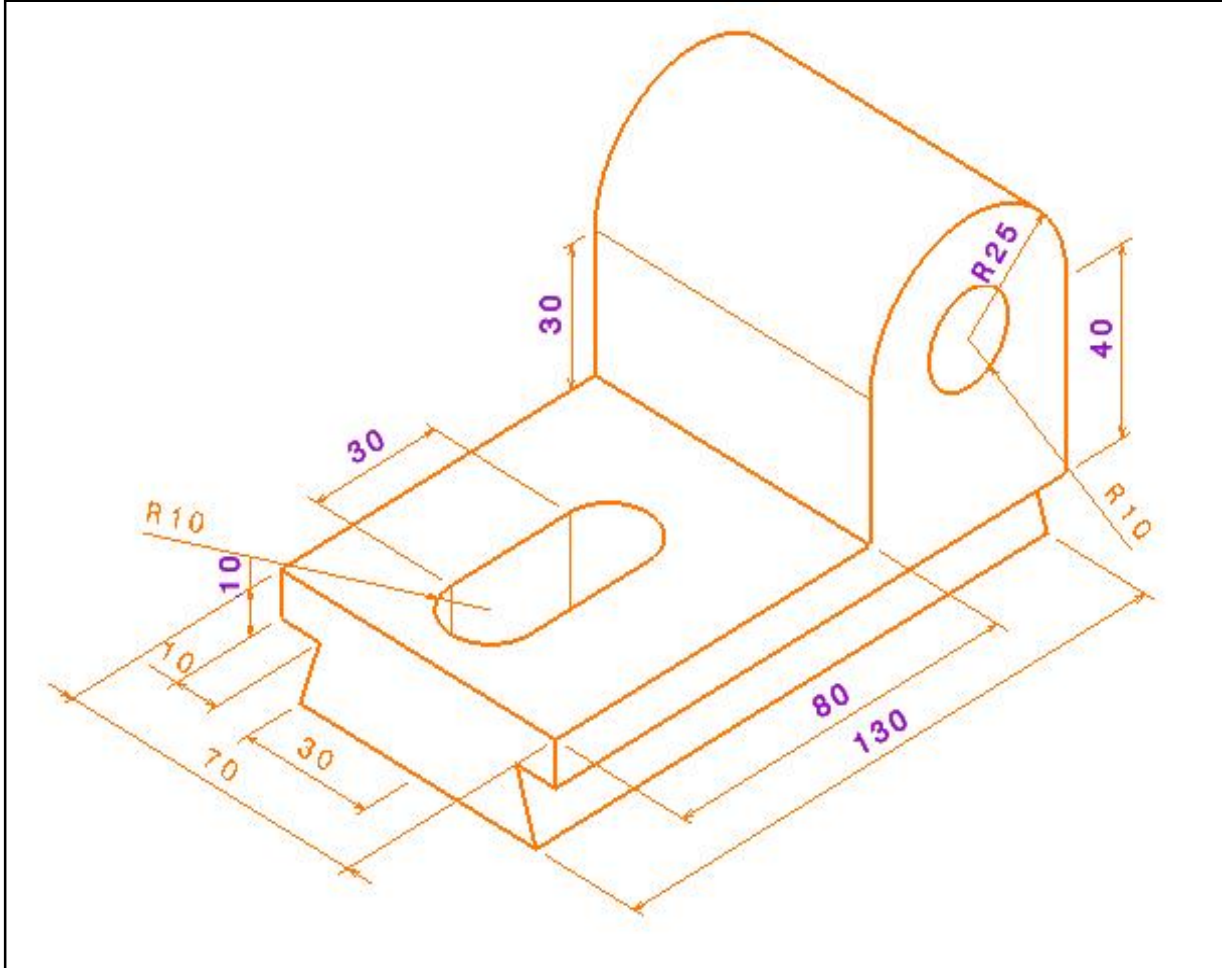


All dimensions are in 'mm'

RESULT:

3-D Models of given simple solids are created using Auto CAD software.

2D Multiple Views from 3D Model Object



All dimensions are in 'mm'

EX NO: 11

DATE:

OBTAINING 2-D MULTI-VIEW DRAWINGS FROM 3-D MODEL

AIM:

To create a2d multiple views from 3d model using Auto CAD software.

SOFTWARE USED:

❖ Auto CAD 2007

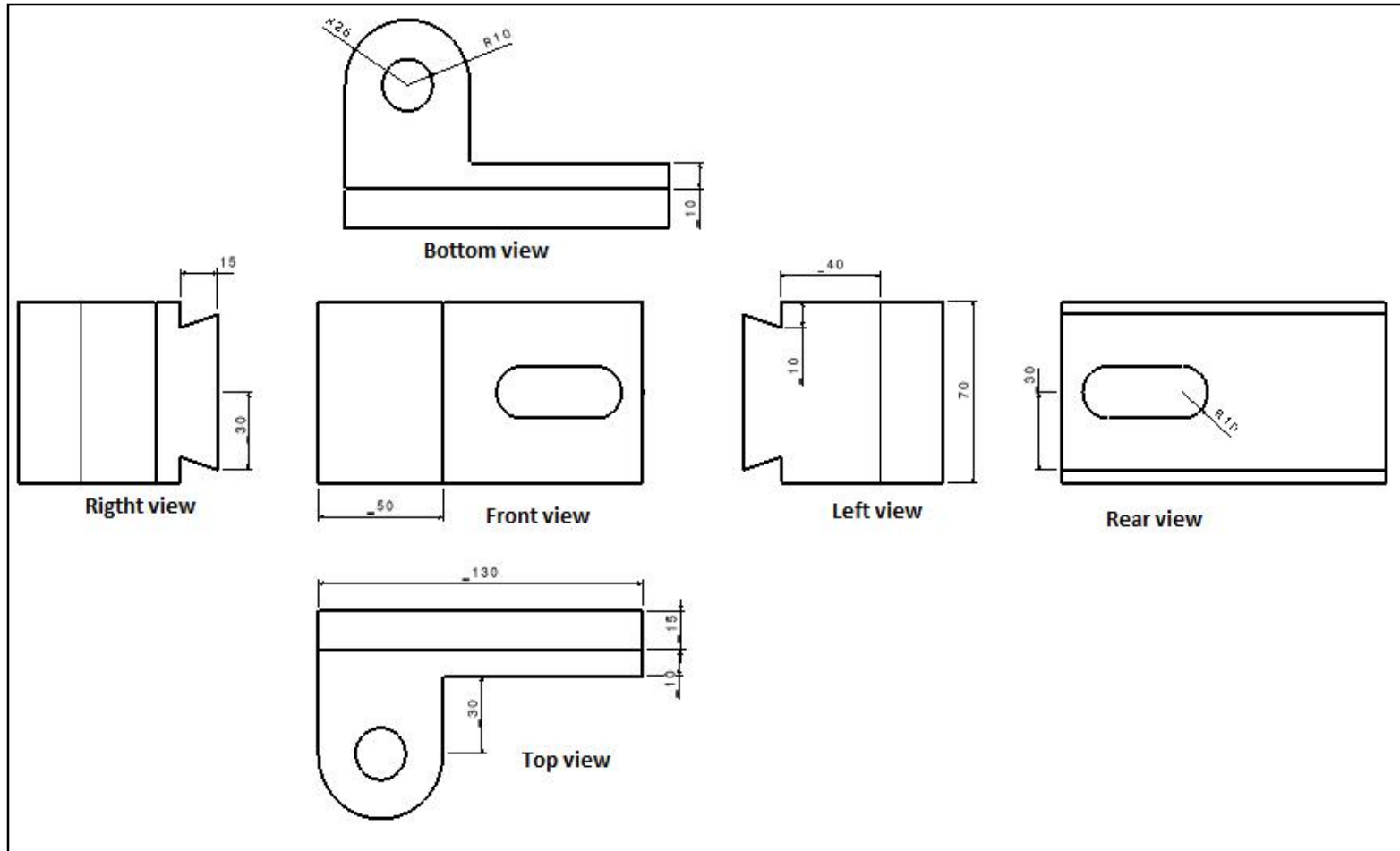
COMMANDS USED:

Limits, Zoom, Line, Circle, Layout, Text, Mtext.

PROCEDURE:

- Limits are set for standard drawing size are drawn.
- Given Solid is modeled using Extrude command.
- Layout is created with four viewports for Front, Top, Left Side and Isometric views.
- Dimensioning is done in the appropriate views.
- Finished work sheet is saved and hard copy is taken.

2D Multiple Views



All dimensions are in 'mm'

RESULT:

2D multiple views were obtained for the given solid using Auto CAD software.

SHORTCUT KEY FOR ALL THE COMMANDS

- ❖ Line -L
- ❖ Circle -c
- ❖ Move -m
- ❖ Copy -co
- ❖ Hatching -h
- ❖ Explode -x
- ❖ Trim -tr
- ❖ Mirror-mi
- ❖ Offset -o
- ❖ Dimension style -d
- ❖ Text-t
- ❖ Text edit -ed
- ❖ Rotate -ro
- ❖ Fillet -f
- ❖ Break -br
- ❖ Arc- a
- ❖ Polygon -pol
- ❖ Ellipse -el
- ❖ Multiline -mline
- ❖ Poly line-pline

FUNCTION KEYS

FUNCTION KEYS	FUNCTION DEFINED
F1	Help
F2	Command window on/off
F3	Object Osnap on/off
F4	Tablet on/off
F5	Isoplane top/right/left
F6	Co-ordinate display on/off
F7	Grid on/off
F8	Ortho on/off
F9	snap on/off
F10	Polar tracking on/off
F11	Object snap tracking on/off