SRI SHANMUGHA COLLEGE OF ENGINEERING AND TECHNOLOGY

(Approved by AICTE, Affiliated to Anna University and Accredited by NAAC & NBA (ECE)

Pullipalayam, Morur (P.O), Sankari (T.k), Salem (D.T) – 637 304

<u>DEPARTMENT OF MECHANICAL ENGINEERING</u>

ME8381 - Computer Aided Machine Drawing

Vision and Mission of the institute

VISION

To be an Institute of repute in the field of Engineering and Technology by implementing the best educational practices akin to global standards for fostering domain knowledge and developing research attitude among students to make them globally competent

MISSION

M1: Achieving excellence in Teaching Learning process using state of the art resources.

M2: Extending opportunity to upgrade faculty knowledge and skills.

M3: Implementing best student training practices for requirements of Industrial scenario of the State.

M4: Motivating faculty and students in research activity for real-time application.

Vision and Mission of the Department

VISION

To prepare competent mechanical engineers capable of working in an interdisciplinary environment contributing to society through innovation, leadership and entrepreneurship

MISSION

M1: To offer quality education which enables them in professional practice and career

M2: To provide learning opportunities in the state-of-the-art research facilities to create, interpret, apply and disseminate knowledge in their profession

M3: To prepare the students as professional engineers in the society with an awareness of environmental and ethical values

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PROGRAMOUTCOMES (POs):

PO1 Engineering knowledge: Apply the knowledge of mathematics, science, engineering fundamentals and an engineering specialization to the solution of complex engineering problems.

PO2 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.

PO3 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, safety, cultural, societal and environmental considerations.

PO4 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.

PO5 Modern tool usage: Create, select, apply appropriate techniques, resources, modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

PO6 The engineer and society: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal, cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO7 Environment and sustainability: Understand the impact of the professional engineering solutions in societal, environmental contexts, demonstrate the knowledge and need for sustainable development.

PO8 Ethics: Apply ethical principles, commit to professional ethics, responsibilities and norms of the engineering practice.

PO9 Individual and team work: Function effectively as an individual, as a member or leader in diverse teams and in multidisciplinary settings.

PO10 Communication: Communicate effectively on complex engineering activities with the engineering community with society at large being able to comprehend, write effective reports, design documentation, make effective presentations and receive clear instructions.

PO11 Project management and finance: Demonstrate knowledge, 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.

PO12 Life-long learning: Recognize the need, ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

PSO1 Manufacturing: Modelling, Simulation and Analysis in the field of Manufacturing.

PSO2 Design: Develop and implement new ideas on product design with help of modern CAD tools.

PROGRAM EDUCATIONAL OBJECTIVES (PEOs)

PEO1: To prepare students to take up career in Industry, Academia as well as in Public services.

PEO2: To provide core domain and interpersonal skills to design & develop mechanical systems for Interdisciplinary applications following ethical code.

PEO3: To develop qualities to progress in entrepreneurship and research activities.

COURSE OUTCOMES:

Upon the completion of this course the students will be able to

C116.1	follow the drawing standards
C116.2	understands the Fits and Tolerances
C116.3	re-create part drawings
C116.4	understands Sectional views and assembly drawings as per standards
C116.5	Create 3D modeling and Assembly

CO-PO MAPPING MATRIX:

Course Outcomes	Program Outcomes													
	PO 1	PO 2	PO 3	PO 4	PO 5	PO 6	PO 7	PO 8	PO 9	PO 10	PO 11	PO 12	PSO 1	PSO 2
C116.1	2	2	2		2			1	2	2		2	2	3
C116.2	2	2	1		3			1	2	2		2	2	3
C116.3	2	2	1		3			1	2	3		3	3	3
C116.4	3	1	2		2			2	2	2		2	2	3
C116.5	3	1	2		2			2	2	2		2	2	3
C116	2.25	1.75	1.50		2.50			1.25	2.00	2.25		2.25	2.25	3.00

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LIST OF EXPERIMENTS

Sl. No	K Level	Name of the Experiment	Relevance to COs	Page No
1.	K2	Study of capabilities of software for Drafting and Modeling – Coordinate systems relative, polar, etc.	CO1	
2.	K3	Drawing of a Title Block with necessary text and projection symbol	CO3	
3.	K3	Drawing of curves like parabola, spiral, involute using B spline or cubic spline.	CO3	
4.	K3	Drawing of front view and top view of simple solids like prism, pyramid, cylinder, cone, etc., and dimensioning.	CO3	
5.	K3	Drawing front view, top view and side view of objects from the given pictorial views block, Base of a mixer, Simple stool, Objects with hole and curves)	CO3	
6.	K3	Drawing of a plan of residential building	CO3	
7.	K3	Drawing of a simple steel truss	CO3	
8.	K3	Drawing sectional views of prism, pyramid, cylinder, cone, etc	CO4,CO5	
9.	K3	Drawing isometric projection of simple objects	CO3,CO5	
10.	K3	Creation of 3-D models of simple objects and obtaining 2-D multi-view drawings from 3D model	CO2,CO4,CO5	
Content Beyond the Syllabus				
1.	K3	Design and assembly of industrial components	CO2,CO4,CO5	

EX NO: 1   STUDY OF CAPABILITIES OF SOFTWARE FOR DRAFTING & MODELING CO- ORDINATE SYSTEM

AIM

To Study of capabilities of software for Drafting

COMPUTER AIDED DRAWING

A drawing is prepared traditionally using instruments. The modern alternative is to draw with the aid of computers. This method is known as computer aided drafting (or) Computer aided drawing (or) computer assisted drafting (or) computer argument drafting and computer automated drafting.

Modern CAD systems are based on the interactive computer graphics. Interactive computer graphics denotes a user defined system in which the computer is employed to create and display data in the terms of pictures and symbols, by the communication between the user and the system. A typical interactive computer graphics system in combination of both hardware and software.

The four fundamental elements of drawing are points, straight lines, arcs and circles. In computer aided drawing systems, the points on the screen be located by selected with the cursor, snapping to the grip, points (or) by entering the numerical values of the co-ordinates of the points. The other elements can be drawn easily by using relevant commands.

Scaling The capacity to enlarge (or) diminish the size of a displayed feature without changing its shape.

Zooming The capacity to enlarge (or) reduce a selected area of the drawing seen on the screen.

Translating The capacity to move parts of a drawing and redrawing them in the new position to a selected scale.
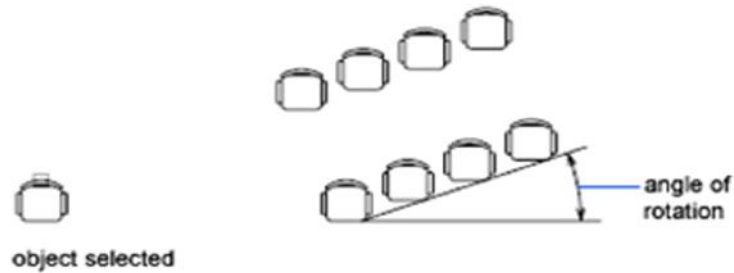
Rotation The capability of rotating the features about a selected position and redraw them in a new angular position.

Mirroring To reverse the image of a feature about the chosen line of symmetry.
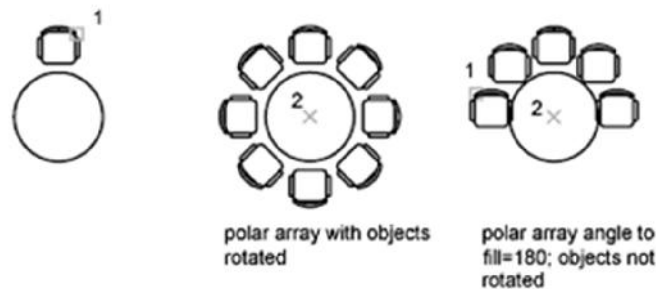
Duplicating The capability of redrawing a feature (or) component many times and displaying in orderly manner of our choice. The CAD system offers a number of automatic features such as fillet, chamfering, hatching and shading of any specified area, dimensioning and text displayed in the required format.

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

Rectangular Arrays A rectangular array is built along a baseline defined by the current snap rotation angle. This angle is zero by default, so the rows and columns of a rectangular array are orthogonal with respect to the *X* and *Y* axes. The default angle 0 direction setting can be changed in UNITS.

object selected

Polar Arrays When you create a polar array, the array is drawn counter clockwise or clockwise, depending on whether you enter a positive or a negative value for the angle to fill. The radius of the array is determined by the distance from the specified center point to a reference or base point on the last selected object. You can use the default reference point (usually an arbitrary point that coincides with a snap point), or you can specify a new base point to be used as the reference point.

polar array with objects rotated

polar array angle to fill=180; objects not rotated

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

text object not part of the boundary set

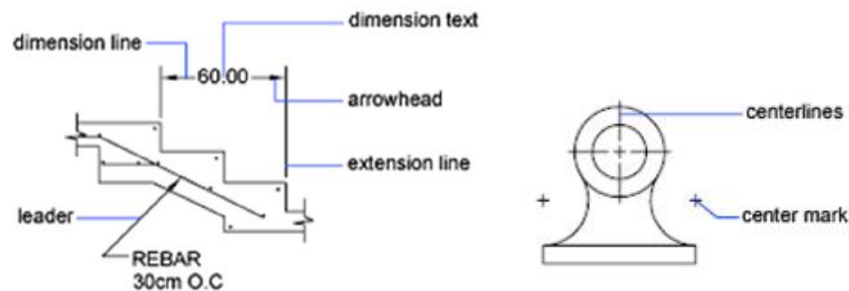text object included in the boundary set

You can choose from several methods to specify the boundaries of a hatch.
- Specify a point in an area that is enclosed by objects.
- Select objects that enclose an area.
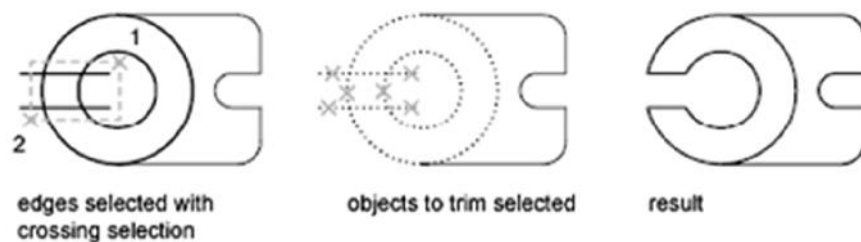- Drag a hatch pattern into an enclosed area from a tool palette or Design Center.

When you hatch a drawing, whole or partial objects that are not part of the object boundary are ignored. If a hatch line encounters an object such as text, an attribute, or a solid-fill object, and if the object is selected as part of the boundary set, HATCH hatches around the object.

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Dimensions Dimensions have several distinct elements: dimension text, dimension lines, arrowheads, and extension lines Dimension text is a text string that usually indicate the measurement value. The text can also include prefixes, suffixes, and tolerances. A dimension line indicates the direction and extent of a dimension. For angular dimensions, the dimension line is an arc. Arrowheads, also called symbols of termination, are displayed at each end of the dimension line. You can specify different sizes and shapes for arrowheads or tick marks. Extension lines, also called projection lines or witness lines, extend from the feature to the dimension line.

A center mark is a small cross that marks the center of a circle or arc. Center lines are broken lines that mark the center of a circle or arc.

Trim You can shorten or lengthen objects to meet the edges of other objects. This means you can first create an object such as a line and then later adjust it to fit exactly between other objects. Objects you select as cutting edges or boundary edges are not required to intersect the object being trimmed. You can trim or extend an object to a projected edge or to an extrapolated intersection; that is, where the objects would intersect if they were extended. If you do not specify a boundary and press ENTER at the Select Objects prompt, all displayed objects become potential boundaries. When you trim several objects, the different selection methods can help you choose the current cutting edges and objects to trim. In the following example, the cutting edges are selected using crossing selection

edges selected with crossing selection   objects to trim selected   result

Move You can move objects at a specified distance and direction from the originals. Use Coordinates, grid snap, object snaps, and other tools to move objects with precision. A practical example is moving a door in a wall. The door in the illustration is entirely within a crossing selection, while the wall lines are only partly within the crossing selection area.

Advantages of using CAD:

To increase the productivity of designer:

The designer to visualize the product and its component sub-assemblies and parts, by reading the time required in synthesis and documenting the design.

To improve quality of design:

The CAD system permits a more analysis and a number of design alternative can be investigated, designs are also through the greater accuracy provided by the system.

To improve communication:

Use of Cad system provides better engineering drawing, more standardization in the drawing, better documentation of the design, greater legibility and saving in time.

To create library:

Commonly used architectural building, electrical and mechanical components can be drawn and stored in memory and recalled when needed. Further they can be positioned anywhere on the screen and redraw to any scale angle and angle of inclination.

To create database for manufacturing:

The process of creating the documentation in the product using design much of required database to manufacture the product is also created.
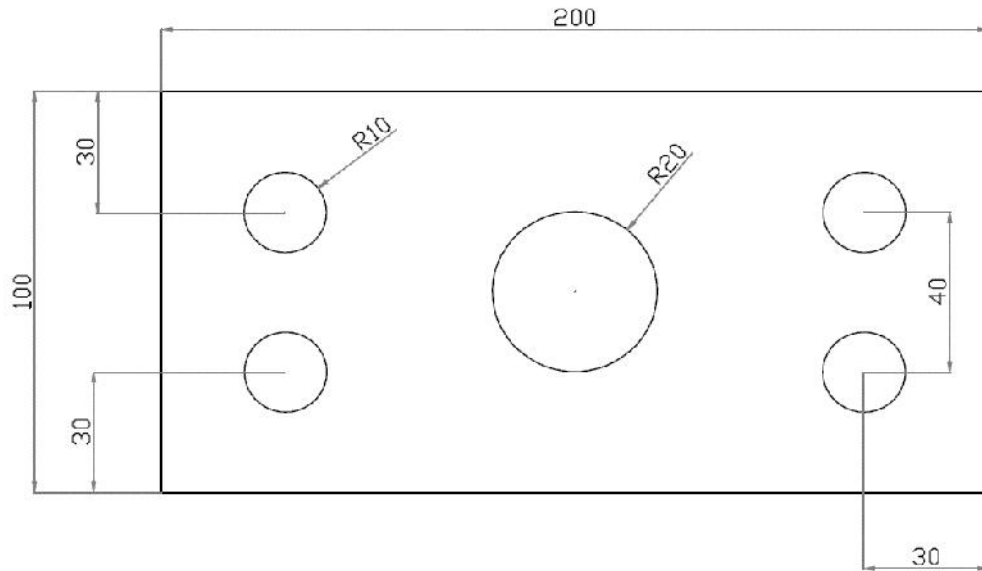
AutoCAD - Commands

Command	Description
ARC	Creates an arc
ARRAY	Creates multiple copies of objects in a pattern
BLOCK	Creates a block definition from objects you select
BREAK	Erases parts of objects or splits an object in two
CHAMFER	Bevels the edges of objects
CIRCLE	Creates a circle
COPY	Duplicates objects
DIM AND DIM1	Accesses Dimensioning mode
DIMEDIT	Edits dimensions
DIMSTYLE	Creates and modifies dimension styles
DIMTEDIT	Moves and rotates dimension text
DIST	Measures the distance and angle between two points
DIVIDE	Places evenly spaced points or blocks on the length or perimeter of an object
ELLIPSE	Creates an ellipse or an elliptical arc
ERASE	Removes objects from a drawing
EXPLODE	Breaks a compound object into its component objects
EXTEND	Extends an object to meet another object
EXTRUDE	Creates unique solid primitives by extruding existing 2D objects
FILLET	Rounds and fillets the edges of objects
GRID	Displays a dot grid in the current viewport
GROUP	Creates a named selection set of objects
HATCH	Fills a specified boundary with a pattern

HIDE	Regenerates a three-dimensional model with hidden lines Suppressed
IMPORT	Imports files in various formats into AutoCAD
INSERT	Places a named block or drawing into the current drawing
INTERSECT	Creates composite solids or regions from the intersection of two or more solids or regions and removes the areas outside of the intersection
ISOPLANE	Specifies the current isometric plane
LAYER	Manages layers and layer properties
LAYOUT	Creates a new layout and renames, copies, saves, or deletes an existing layout
LENGTHEN	Lengthens an object
LIMITS	Sets and controls the drawing boundaries and grid display
LINE	Creates straight line segments
LTSCALE	Sets the line type scale factor
LWEIGHT	Sets the current line weight, line weight display options, and Line weight units
MATLIB	Imports and exports materials to and from a library of materials
MEASURE	Places point objects or blocks at measured intervals on an object
MIRROR	Creates a mirror image copy of objects
MOVE	Displaces objects a specified distance in a specified direction
MTEXT	Creates multiline text
MULTIPLE	Repeats the next command until canceled
OFFSET	Creates concentric circles, parallel lines, and parallel curves
OPTIONS	Customizes the AutoCAD settings
ORTHO	Constrains cursor movement
OSNAP	Sets object snap modes
PAN	Moves the drawing display in the current viewport
PEDIT	Edits polylines and three-dimensional polygon meshes
PLAN	Displays the plan view of a user coordinate system
PLINE	Creates two-dimensional polylines
PLOT	Plots a drawing to a plotting device or file
PLOTSTYLE	Sets the current plot style for new objects, or the assigned plot style for objects
POINT	Creates a point object
POLYGON	Creates an equilateral closed polyline
QDIM	Quickly creates a dimension
RECTANG	Draws a rectangular polyline
REDO	Reverses the effects of the previous
REGION	Creates a region object from a selection set of existing objects RENAME Changes the names of objects
RENDER	Creates a photorealistic or realistically shaded image of a three – Dimensional wire frame or solid model

REVOLVE	Creates solids by revolving two-dimensional objects about an axis
ROTATE	Moves objects about a base point
SCALE	Enlarges or reduces selected objects equally in the X, Y, and Z directions
SKETCH	Creates a series of freehand line segments
SLICE	Slices a set of solids with a plane
SNAP	Restricts cursor movement to specified intervals
SOLDRAW	Generates profiles and sections in viewports created with SOLVIEW
SOLID	Creates solid-filled polygons
SPHERE	Creates a three-dimensional solid sphere
SPLINE	Creates a quadratic or cubic spline (NURBS) curve
SUBTRACT	Creates a composite region or solid by subtraction
TRIM	Trims objects at a cutting edge defined by other objects
U	Reverses the most recent operation
UNION	Creates a composite region or solid by addition
VPOINT	Sets the viewing direction for a three-dimensional visualization of the drawing
VPORTS	Divides the drawing area into multiple tiled or floating viewports
WBLOCK	Writes objects or a block to a new drawing file
XATTACH	Attaches an external reference to the current drawing
XPLODE	Breaks a compound object into its component objects
3D	Creates three-dimensional polygon mesh objects

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1. Draw the following Fig. By using Absolute co-ordinate system.

Command: Line
Specify first point: 0, 0
Specify next point or (undo):200, 0
Specify next point or (undo):200,100
Specify next point or (close / undo):0,100
Specify next point or (close / undo): C
Command: Circle
Circle specify center point for circle or (3P/2P/Ttr (tan tan radius):170, 30
Specify radius of circle or (Diameter) <10.0000>:10
Command: Circle
Circle specify center point for circle or (3P/2P/Ttr (tan tan radius):170, 70
Specify radius of circle or (Diameter) <10.0000>:10
Command: Circle
Circle specify center point for circle or (3P/2P/Ttr (tan tan radius):30, 70
Specify radius of circle or (Diameter) <10.0000>:10
Command: Circle
Circle specify center point for circle or (3P/2P/Ttr (tan tan radius):100, 50
Specify radius of circle or (Diameter) <10.0000>:20

2. Draw the following Fig.by using Relative co-ordinate system.

Command:Line
Specify first point: 0, 0
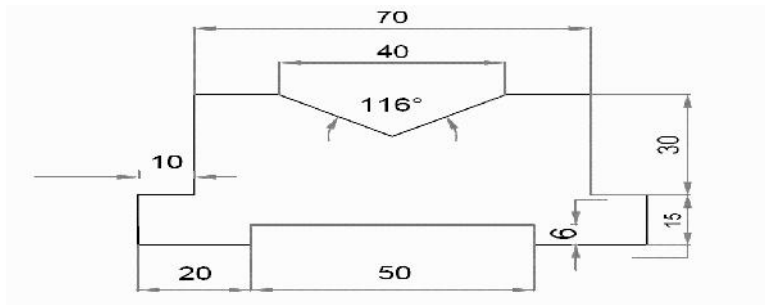Specify next point or (Undo):@20, 0
Specify next point or (Undo): @0, 5
Specify next point or (Close / Undo):@30,0
Specify next point or (Close / Undo):@0,4
Specify next point or (Close / Undo):@20,0
Specify next point or (Close / Undo):@0,3
Specify next point or (Close / Undo):@10,0
Specify next point or (Close / Undo):@-10,0
Specify next point or (Close / Undo):@0,3
Specify next point or (Close / Undo):@-20,0
Specify next point or (Close / Undo):@0,4
Specify next point or (Close / Undo):@-30,0
Specify next point or (Close / Undo):@0,5
Specify next point or (Close / Undo):@-20,0
Specify next point or (Close / Undo): C

3. Draw the following Fig.by using polar Co-ordinate system.

Command:Line

Specify first point: 0, 0
Specify next point or (Undo):@20 < 0
Specify next point or (Close / Undo):@6<90
Specify next point or (Close / Undo):@50<90
Specify next point or (Close / Undo):@6<270
Specify next point or (Close / Undo):@20<0
Specify next point or (Close / Undo):@15<90
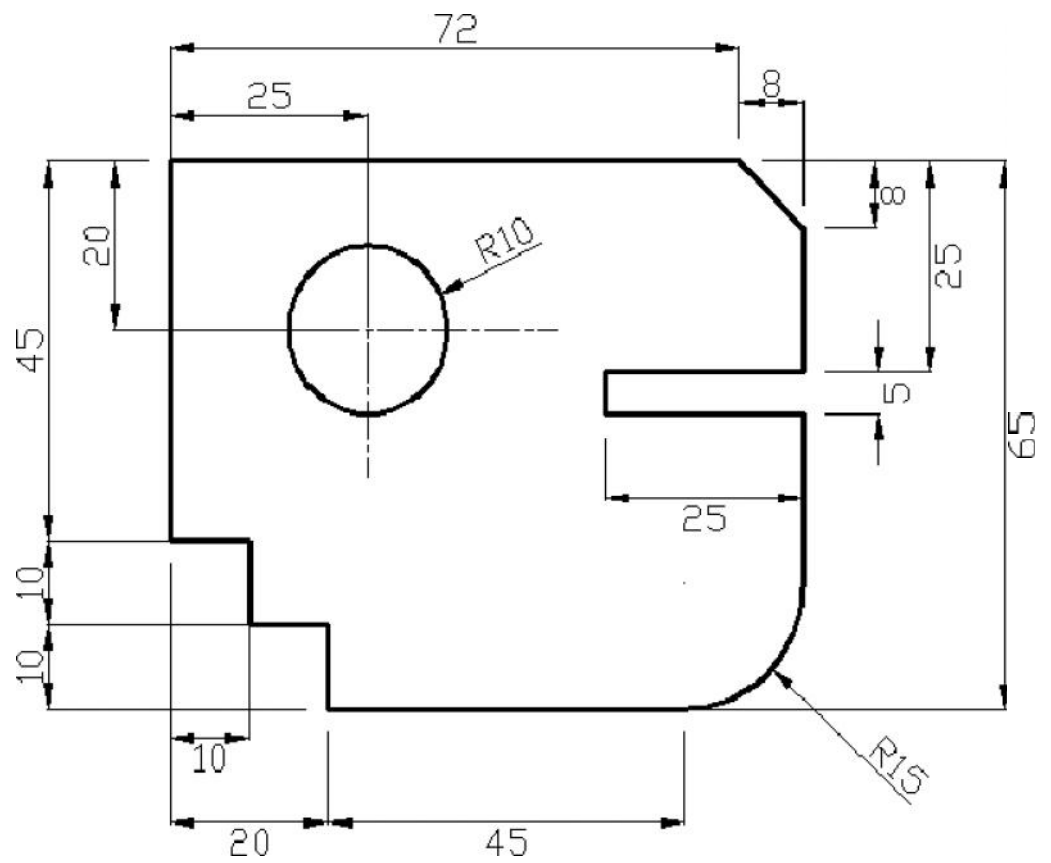Specify next point or (Close / Undo):@10<180
Specify next point or (Close / Undo):@30<90
Specify next point or (Close / Undo):@15<180
Specify next point or (Close / Undo):@23.6<212
Specify next point or (Close / Undo):@23.6<148
Specify next point or (Close / Undo):@15<180
Specify next point or (Close / Undo):@30<270

Specify next point or (Close / Undo):@10<180
Specify next point or (Close / Undo): c

RESULT

Thus the study of CAD and the commands of AutoCAD was done.

All Dimensions are in 'mm'

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Creation of Simple figures Like Polygon and General Multiline Figures

AIM:

To create the following drawing using AutoCAD commands like circle, line, fillet and trim.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

PROCEDURE:
- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.
- Using commands like line, circle, fillet and trim to complete the drawing.

Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000, 0.0000>: 0, 0

Specify upper right corner <12.0000, 9.0000>: 297,420

Command: line

Specify first point: As per limits

Specify next point or [Undo] : <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

Command: circle

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

Specify radius of circle or [Diameter]:

Command: erase

Select all the drawn circles.

Select objects: all found
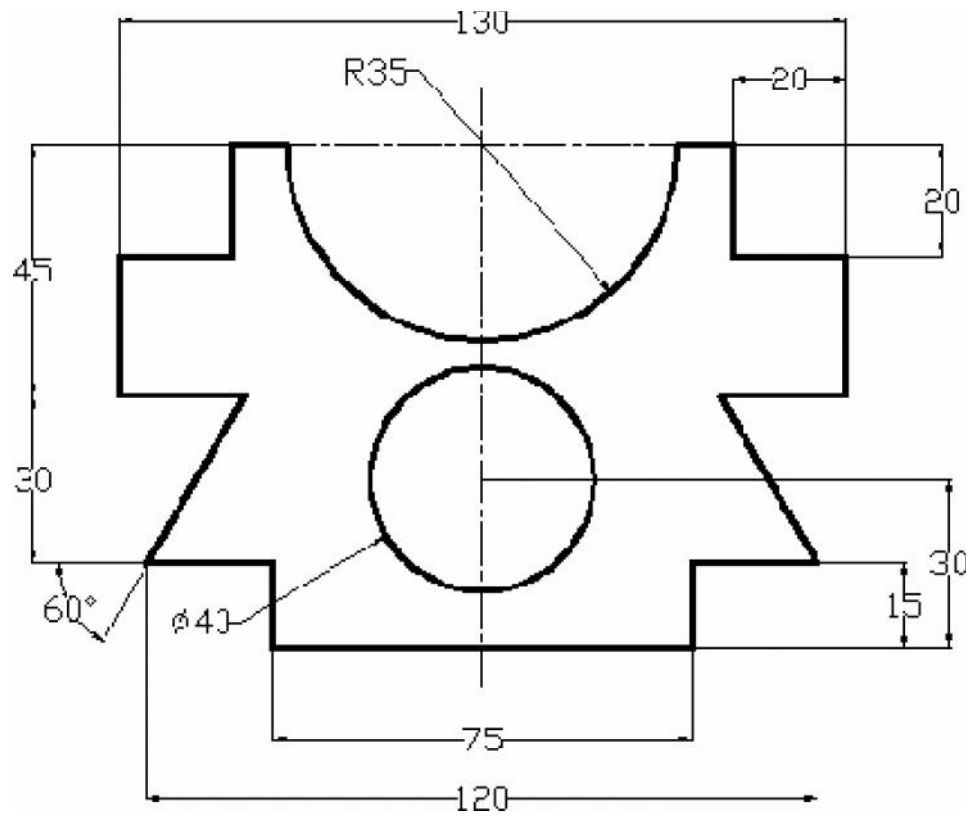
Select objects: Select objects: enter

Command: fillet

Current settings: Mode = trim, Radius =

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: R

Specify fillet radius <0.0000>:

Select first object or [Undo/Polyline/Radius/Trim/Multiple]:

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All Dimensions are in 'mm'

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Select second object or shift-select to apply corner:

Command: trim

Current settings: Projection=UCS, Edge=None

Select cutting edges:

Select objects or <select all>:

Select object to trim or shift-select to extend or

[Fence/Crossing/Project/Edge/erase/Undo]:

Command: dim linear

Dissociative dimension priority = endpoint

Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius)

Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of text <0>: Specify first corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200, 10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: all

Regenerating model.

- Detail the drawing as per dimensions.

RESULT:

The drawing is created using AutoCAD commands like circle, line, fillet, trim and detailed as per the given dimensions.

EX NO: 2 Drawing of a Title Block with Necessary Text and Projection Symbol

AIM:

To create the Title Block with Necessary Text and Projection Symbol using AutoCAD commands.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

PROCEDURE:

- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.
- Using commands like line, circle, fillet and trim to complete the drawing.

Command: line

Specify first point:

Specify next point or [Undo]: <Polar on> <Ortho on> Specify next point or Close/Undo]:

Specify next point or [Close/Undo]: enter

Command: offset

Current setting: erase source =no layer= source offset gap type=0

Specify offset distance or (through/erase/layer)<through>:5

Select object to offset of (exit/undo) <exit>: pick drawing

Specify point on side to offset or (exit/multiple/undo) <exit>: pick inside the rectangle

Specify object to offset or (exit/undo) <exit>:

Command: circle

Current setting: erase source =no layer set gap setting = 0

Specify radius of circle or (diameter): Specify diameter of circle:

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of text <0>: Specify first corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200,10

Type the word of "ALL DIMENSIONS ARE IN mm"

Detail the drawing as per dimensions.

All Dimensions are in mm

RESULT:
The drawing is created using AutoCAD commands like circle, line, offset, trim, text and detailed as per given dimensions.

EX NO:3   Drawing of Curves like Parabola, Spiral, Involute Using B spline or Cubic Spline

AIM:

To draw an involute of a pentagon with base width of 20mm.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

PROCEDURE:

- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.
- Draw the pentagon with the base width of 20mm

Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: 0,0

Specify upper right corner <12.0000, 9.0000>: 297,420

Command: line

Specify first point: As per limits

Specify next point or [Undo]: <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

Command: circle

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

Specify radius of circle or [Diameter]:

Command: erase

Select all the drawn circles.

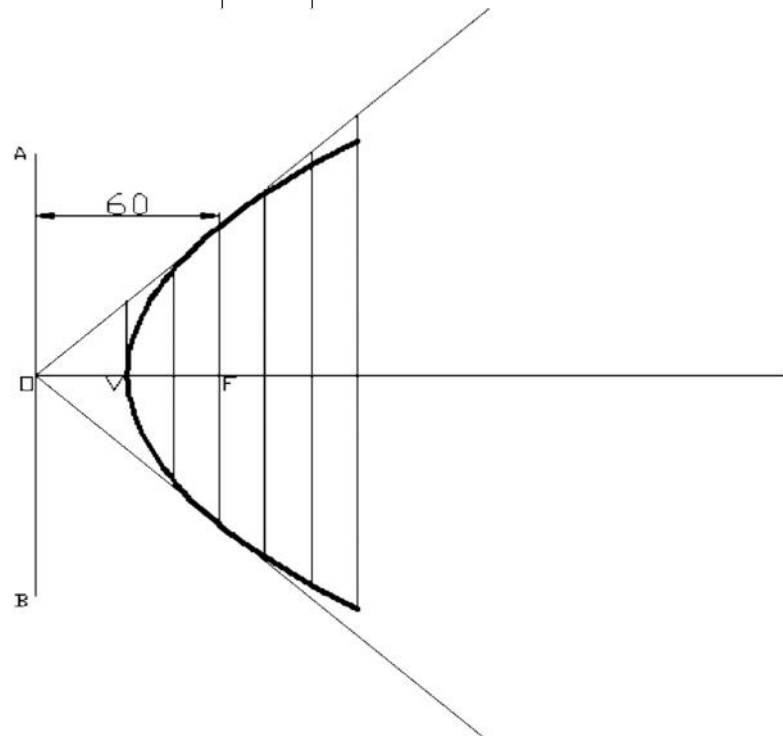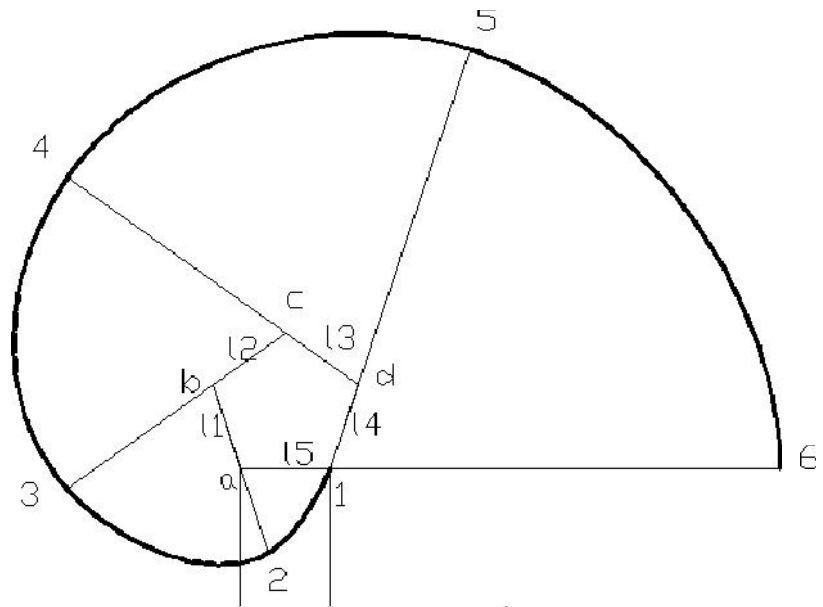Select objects: all found

Select objects: Select objects: enter

Command: trim

Current settings: Projection=UCS, Edge=None

Select cutting edges:

Select objects or <select all>:

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Select object to trim or shift-select to extend or

All Dimensions are in mm

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[Fence/Crossing/Project/Edge/erase/Undo]:

Command: dim linear

Dissociative dimension priority = endpoint

Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius) Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of text <0>: Specify first corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200,10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: all Regenerating model.

- Detail the drawing as per dimensions.

RESULT:

The involute of a pentagon with base width of 20mm is drawn and detailed as per given dimensions.

EX NO 4: Drawing of front view and op view of simple solids like prism, pyramid, cylinder, cone, etc., and dimensioning.

AIM:

To draw front view and top view of cone.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

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PROCEDURE:

- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.

Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

Specify upper right corner <12.0000, 9.0000>: 297,420

Command: line

Specify first point: As per limits

Specify next point or [Undo]: <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

Command: circle

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

Specify radius of circle or [Diameter]:

Command: trim

Current settings: Projection=UCS, Edge=None

Select cutting edges:

Select objects or <select all>:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/erase/Undo]:

Command: dim linear

Dissociative dimension priority = endpoint

Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius)

24

Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200, 10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all

Command: LTS (Line type scale)

Enter a new line type scale factor <1>:2

Command: LT (Line type)

Select load option

Select required line type

Select objects: enter

CYLINDER

Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

Specify upper right corner <12.0000, 9.0000>: 297,420

Command: line

Specify first point: As per limits

Specify next point or [Undo]: <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

Command: circle

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

Specify radius of circle or [Diameter]:

Command: trim

Current settings: Projection=UCS, Edge=None

Select cutting edges:

Select objects or <select all>:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/erase/Undo]:

Command: dim linear

25

Dissociative dimension priority = endpoint

Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius) Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200,10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all

Command: LTS (Line type scale)

Enter a new line type scale factor <1>:2

Command: LT (Line type)

Select load option

Select required line type

Select objects: enter

HEXAGONAL PRISM

Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

Specify upper right corner <12.0000, 9.0000>: 297,420

Command: line

Specify first point: As per limits

Specify next point or [Undo]: <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

Command: trim

Current settings: Projection=UCS, Edge=None

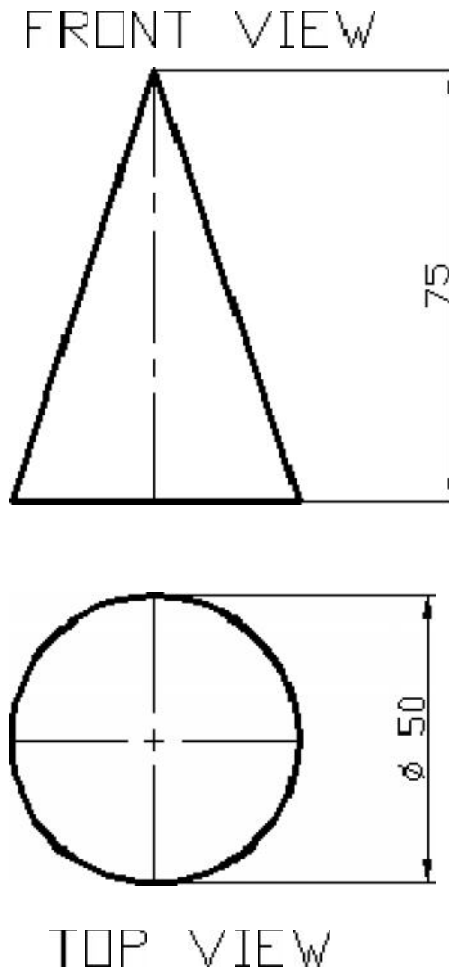Select cutting edges:

Select objects or <select all>:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/erase/Undo]:

Command: dim linear

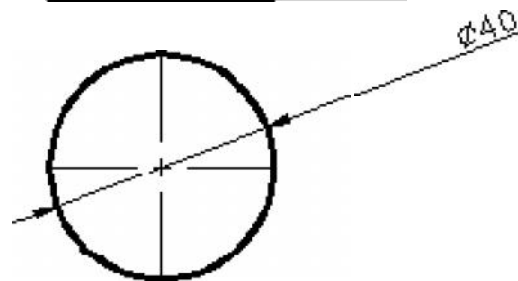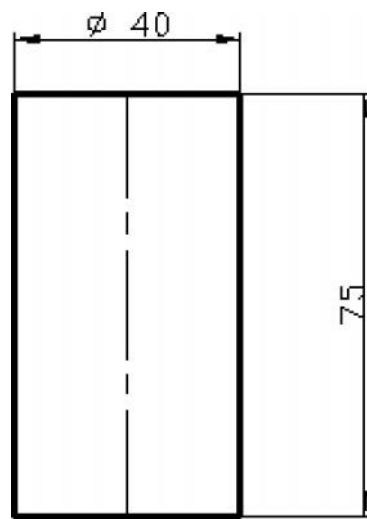Dissociative dimension priority = endpoint

Select geometry to dimension:
Specify dimension line position (continuous /stage greed/base line /ordinate/radius) Diameter dates print (edit/setting)

FRONT VIEW

75

TOP VIEW
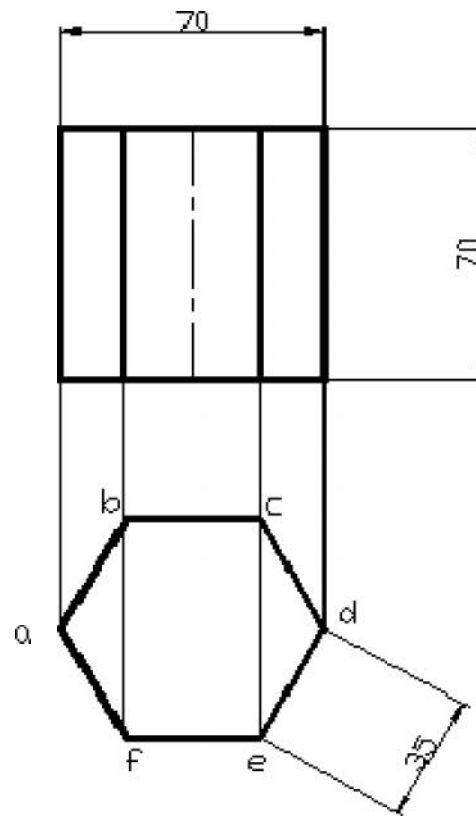
ø 50

All Dimensions are in mm

FRONT VIEW

ø 40

75

⌀40

TOP VIEW

All Dimensions are in mm

FRONT VIEW

70

70

b   c

a   d

f   e

35

TOP VIEW

RESULT:
The front view and top view of the cone is drawn and detailed as per given dimensions.

EX NO 5: Drawing front view, top view and side view of objects from the given pictorial views (eg. v-block, base of a mixie, simple tool, object with hole and curves)

AIM:
To draw 3-D view of the V-BLOCK and then draw front view, top view and side view

HARDWARE USED:
 Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:
 Windows 8, AutoCAD 2014.

PROCEDURE:
- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.
- Using commands like line, circle and trim to complete the drawing.

 Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

Specify upper right corner <12.0000, 9.0000>: 297,420

 Command: line

Specify first point: As per limits

Specify next point or [Undo] : <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

 Command: circle

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

Specify radius of circle or [Diameter]:

 Command: trim

Current settings: Projection=UCS, Edge=None

Select cutting edges:

Select objects or <select all>:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/erase/Undo]:

 Command: dim linear

Dissociative dimension priority = endpoint

30

Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius) Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200, 10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all
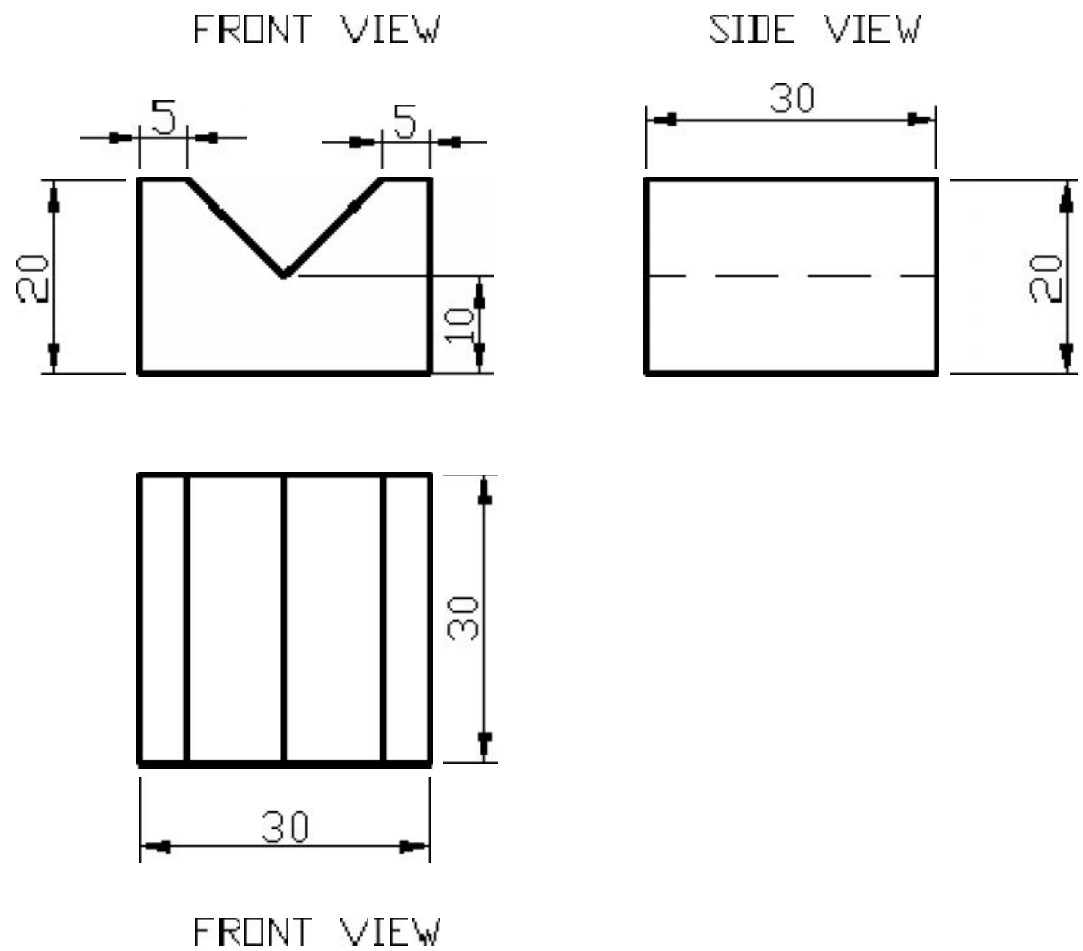
Command: LTS (Line type scale)

Enter a new line type scale factor <1>:2

Command: LT (Line type)

Select load option

Select required line type

Select objects: enter

FRONT VIEW

SIDE VIEW

5   5

20

10

30

20

30

30

FRONT VIEW

RESULT:

The 3-D view of the V-BLOCK, front view, top view and side view of it is drawn and detailed as per given dimensions.

EX NO 6: Drawing of a plan of residential building

AIM:

To draw the plan of residential building with two bedrooms kitchen and hall.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

PROCEDURE:

- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.
- Using commands like line, circle, arc and trim to complete the drawing.

Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

Specify upper right corner <12.0000, 9.0000>: 297,420

Command: line

Specify first point: As per limits

Specify next point or [Undo]: <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

Command: trim

Current settings: Projection=UCS, Edge=None

Select cutting edges:

Select objects or <select all>:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/erase/Undo]:

Command: dim linear

Dissociative dimension priority = endpoint

Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius) Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200, 10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all
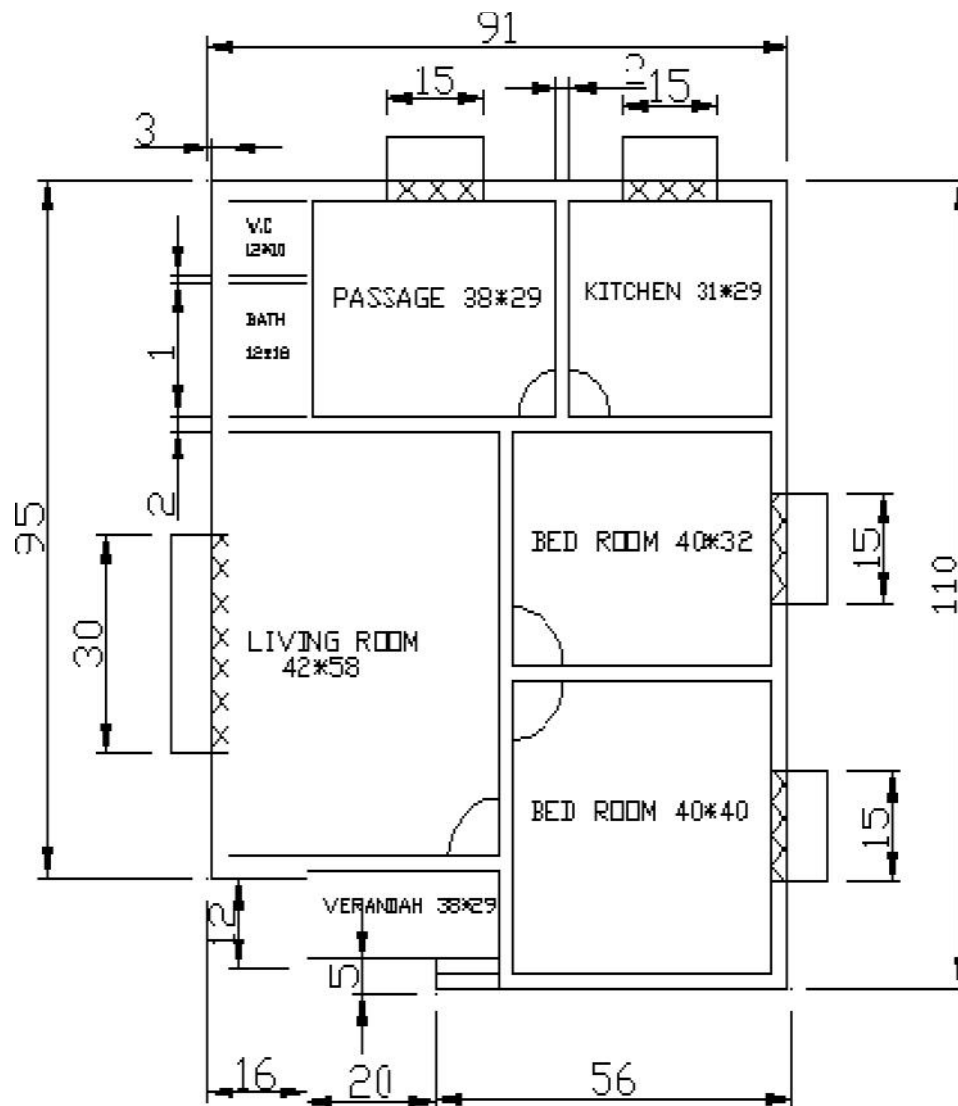
Command: LTS (Line type scale)

Enter a new line type scale factor <1>:2

Command: LT (Line type)

Select load option

Select required line type

Select objects: enter

All Dimensions are in mm

RESULT:

The plan of residential building with two bedrooms kitchen, hall is drawn and detailed as per given dimensions.

`

EX NO 7: Drawing of a simple steel truss

AIM:

To draw the following steel truss in AutoCAD.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

PROCEDURE:

- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.
- Using commands like line, circle and trim to complete the drawing.

   Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

Specify upper right corner <12.0000, 9.0000>: 297,420

   Command: line

Specify first point: As per limits

Specify next point or [Undo] : <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

   Command: trim

Current settings: Projection=UCS, Edge=None

Select cutting edges:

Select objects or <select all>:

Select object to trim or shift-select to extend or [Fence/Crossing/Project/Edge/erase/Undo]:

   Command: dim linear

Dissociative dimension priority = endpoint

Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius) Diameter dates print (edit/setting)

   Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of corner: 20, 20

`

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200, 10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all

Command: LTS (Line type scale)
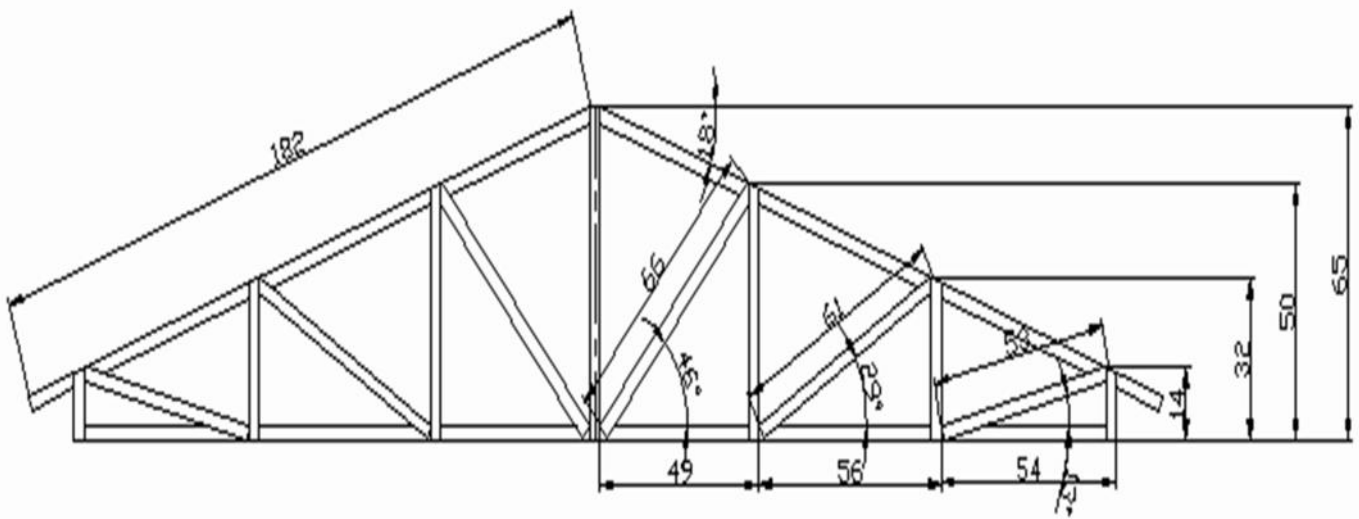
Enter a new line type scale factor <1>:2

Command: LT (Line type)

Select load option

Select required line type

Select objects: enter

All Dimensions are in mm

RESULT:

Thus the steel truss is drawn and detailed as per given dimensions.

`

EX NO 8: Drawing Sectional views of Prism, Pyramid, Cylinder, Cone and Cylinder

AIM:

To draw sectional view of cylinder.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

PROCEDURE:

- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.
- Using commands like line, circle and trim to complete the drawing.

    Command: limits

    Reset Model space limits:

    Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

    Specify upper right corner <12.0000, 9.0000>: 297,420

    Command: line

    Specify first point: As per limits

    Specify next point or [Undo] : <Polar on> <Ortho on>

    Specify next point or [Close/Undo]:

    Specify next point or [Close/Undo]:

    Specify next point or [Close/Undo]: enter

    Command: circle

    Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

    Specify radius of circle or [Diameter]:

    Command: trim

    Current settings: Projection=UCS, Edge=None

    Select cutting edges:

    Select objects or <select all>:

    Select object to trim or shift-select to extend or

    [Fence/Crossing/Project/Edge/erase/Undo]:

    Command: hatch

    Pick internal point or [Select objects/Undo/Settings]:

    Command: dim. Linear

    Dissociative dimension priority = endpoint

    Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius)

Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200, 10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all

Command: LTS (Line type scale)
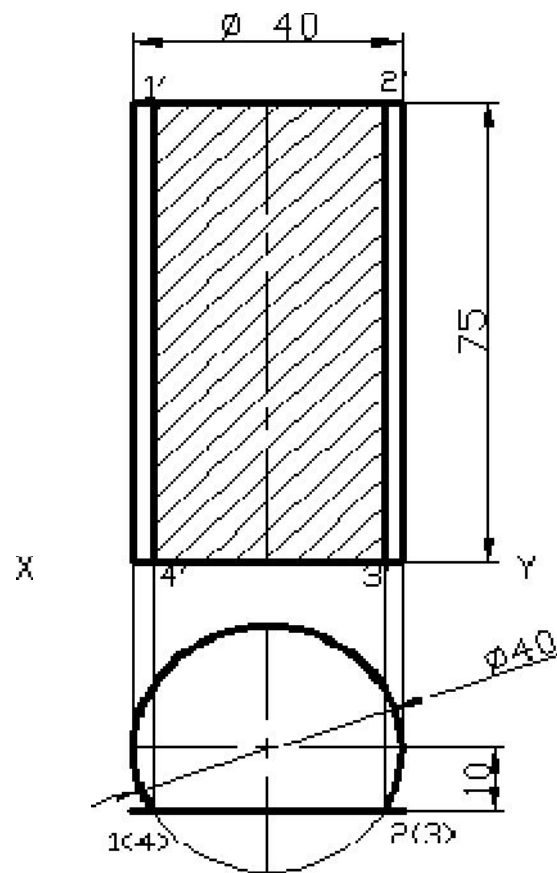
Enter a new line type scale factor <1>:2

Command: LT (Line type)

Select load option

Select required line type

Select objects: enter

CYLINDER

Ø 40

75

Ø40

10

X   Y

1'   2'

4'   3'

1(4)   2(3)

All Dimensions are in mm

Cone

Command: limits

  Reset Model space limits:

  Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

  Specify upper right corner <12.0000, 9.0000>: 297,420

Command: line

  Specify first point: As per limits

  Specify next point or [Undo] : <Polar on> <Ortho on>

  Specify next point or [Close/Undo]:

  Specify next point or [Close/Undo]:

  Specify next point or [Close/Undo]: enter

Command: circle

  Specify center point for circle or [3P/2P/Ttr (tan tan radius)]:

  Specify radius of circle or [Diameter]:

Command: trim

  Current settings: Projection=UCS, Edge=None

  Select cutting edges:

  Select objects or <select all>:

  Select object to trim or shift-select to extend or

  [Fence/Crossing/Project/Edge/erase/Undo]:

Command: hatch

  Pick internal point or [Select objects/Undo/settings]:

Command: dim linear

  Dissociative dimension priority = endpoint

  Select geometry to dimension:

  Specify dimension line position (continuous /stage greed/base line /ordinate/radius)

  Diameter dates print (edit/setting)

Command: text

  Current text style: "Standard" Text height: 0.2000 Annotative: No

  Specify start point of text or [Justify/Style]: Specify height <0.2000>:

  Specify rotation angle of corner: 20, 20

  Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200, 10

  Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

  Specify corner of window, enter a scale factor (nX or nXP), or

  [All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all
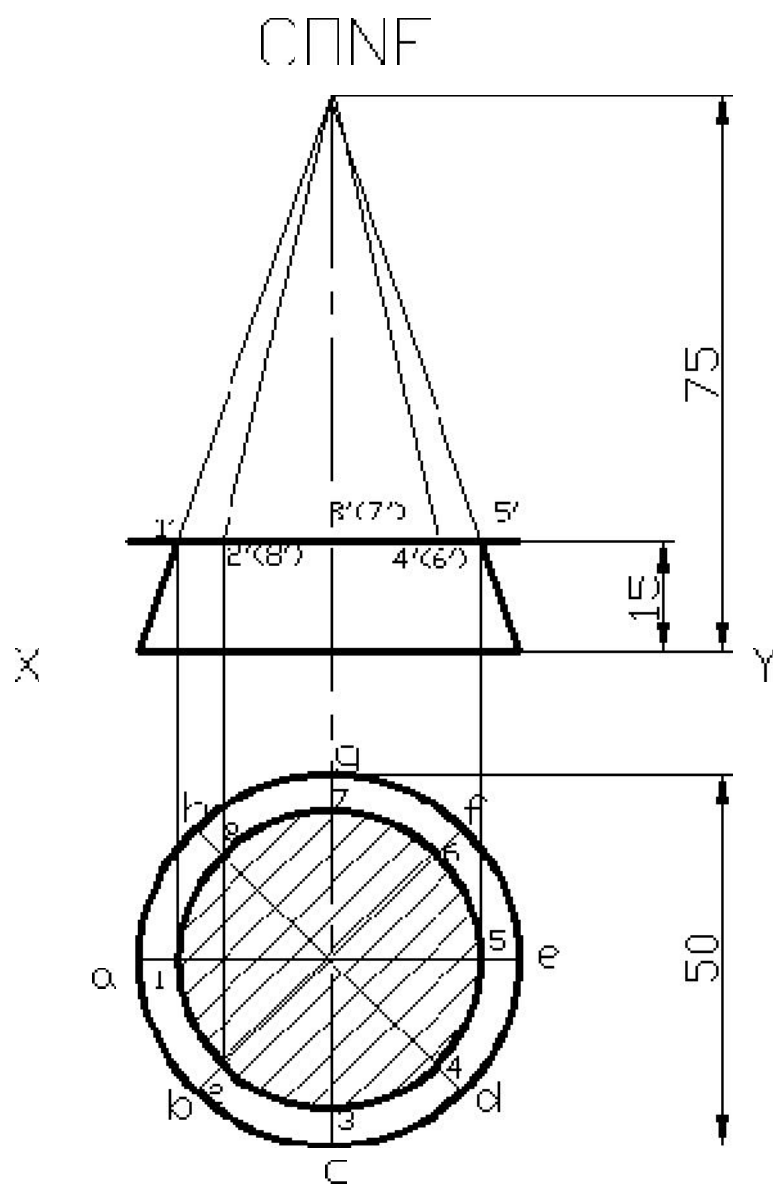
Command: LTS (Line type scale)

  Enter a new line type scale factor <1>:2

Command: LT (Line type)

Select load option
Select required line type
Select objects: enter

CПNF

`

Hexagonal Prism

Command: limits

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

Specify upper right corner <12.0000, 9.0000>: 297,420

Command: line

Specify first point: As per limits

Specify next point or [Undo] : <Polar on> <Ortho on>

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]:

Specify next point or [Close/Undo]: enter

Command: trim

Current settings: Projection=UCS, Edge=None

Select cutting edges:

Select objects or <select all>:

Select object to trim or shift-select to extend or

[Fence/Crossing/Project/Edge/erase/Undo]:

Command: hatch

Pick internal point or [Select objects/Undo/settings]:

Command: dimlinear

Dissociative dimension priority = endpoint

Select geometry to dimension:

Specify dimension line position (continuous /stage greed/base line /ordinate/radius)

Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:

Specify rotation angle of corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200, 10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all

Command: LTS (Line type scale)

Enter a new line type scale factor <1>:2

Command: LT (Line type)

Select load option

Select required line type

Select objects: enter

PRISM

All Dimensions are in mm

RESULT:

The sectional view hexagonal prism is drawn and detailed as per given dimensions.

`

EX NO 9: Drawing isometric projection of simple object

AIM:

To create the isometric projection of the given drawing.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

PROCEDURE:

- Open new file with the standard template and set metric units
- Using limits command set the screen limits
- Using zoom command fit the drawing to window.
- Using commands like line, circle and trim to complete the drawing.
- Set snap style to isometric using SNAP command.
- Keep the cursor in isoplane left by pressing F5 key.

   Command: limits

   Reset Model space limits:

   Specify lower left corner or [ON/OFF] <0.0000, 0.0000>:

   Specify upper right corner <12.0000, 9.0000>: 297,420

   Command: line

   Specify first point: As per limits

   Specify next point or [Undo]: <Polar on> <Ortho on>

   Specify next point or [Close/Undo]:

   Specify next point or [Close/Undo]:

   Specify next point or [Close/Undo]: enter

   Command: trim

   Current settings: Projection=UCS, Edge=None

   Select cutting edges:

   Select objects or <select all>:

   Select object to trim or shift-select to extend or

   [Fence/Crossing/Project/Edge/erase/Undo]:

   Command: dimlinear

   Dissociative dimension priority = endpoint

   Select geometry to dimension:

   Specify dimension line position (contionous /stage greed/base line /ordinate/radius)

Diameter dates print (edit/setting)

Command: text

Current text style: "Standard" Text height: 0.2000 Annotative: No

Specify start point of text or [Justify/Style]: Specify height <0.2000>:
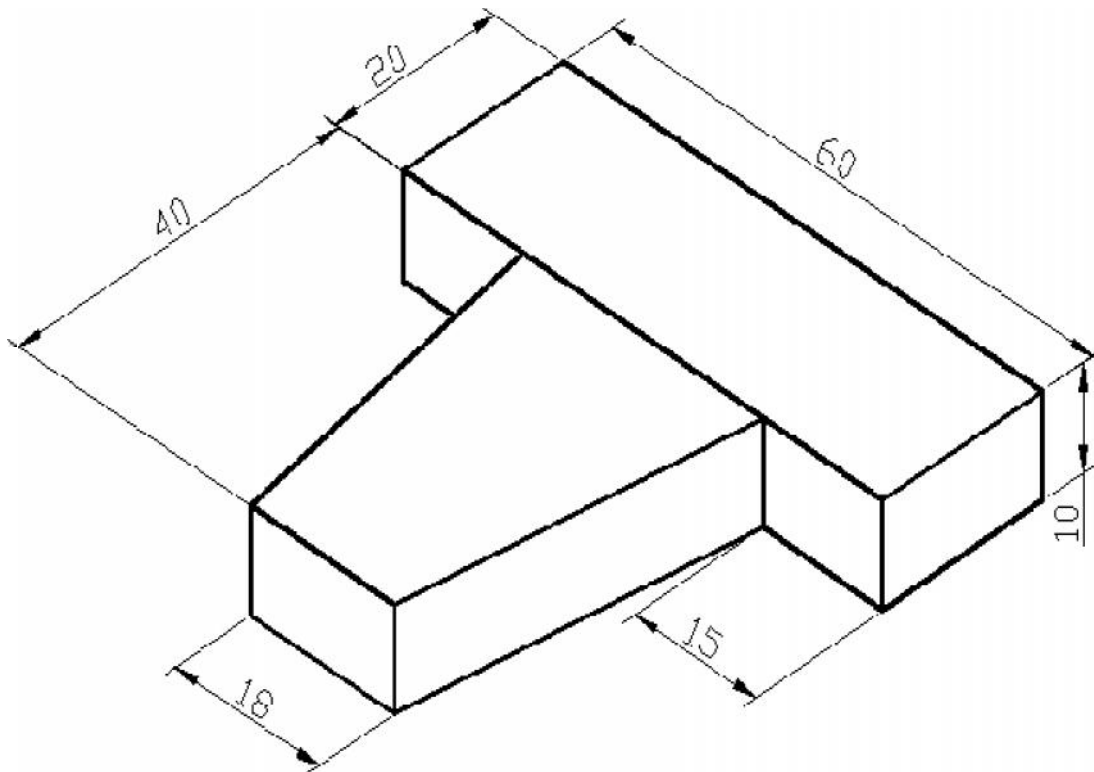
Specify rotation angle of corner: 20, 20

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: 200,10

Type the word of "ALL DIMENSIONS ARE IN mm"

Command: zoom

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <Real time>: all

All Dimensions are in mm

RESULT:

The isometric projection of the given drawing is drawn and detailed as per given dimensions.

EX NO 10:Creation of 3-D models of simple objects and obtaining 2-D multi-view drawings from 3D model

AIM:

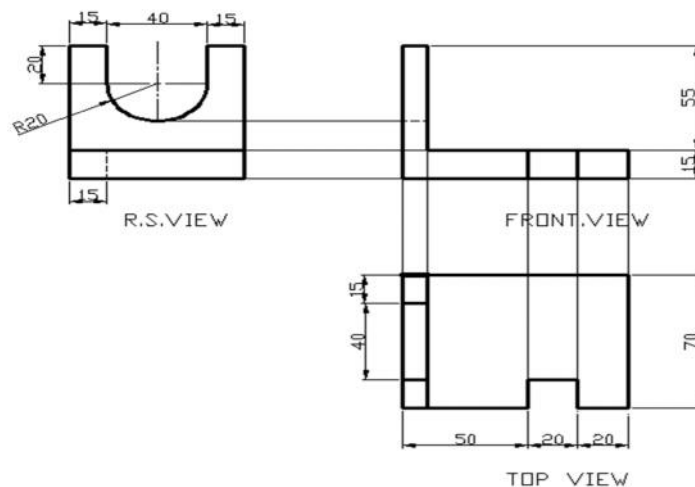To draw 3-D view of the given object and then draw 2D multi view drawing.

HARDWARE USED:

Intel(R) Core (TM) i3-3220 CPU @3.30GHz, 500 GB Hard Disk, 4 GB Ram.

SOFTWARE USED:

Windows 8, AutoCAD 2014.

PROCEDURE:

* Draw the 3-D view by setting view point to 1,-1, 1 using VPOINT command.
* Set Shade mode to 3d by using SHADEMODE command.
* Draw the cross section using LINE command and extrude it using REGION, EXTRUDE commands.
* Draw the front view, top view and side view using LINE, CIRCLE commands.
* Detail the drawing as per dimensions.

All Dimensions are in mm

RESULT:

The 3-D view of the given object, 2D multi view drawing is drawn and detailed as per given dimensions.