INTRODUCTION TO AUTOCAD

Computer Aided Design (CAD) is the process of doing designs with the aid of computers. AutoCAD is a CAD (Computer Aided Design) software application for 2D and 3D design and drafting. It was developed by Autodesk, Inc. First released in December 1982, AutoCAD was one of the first CAD programs to run on personal computers. The two dimensional and three dimensional models created in AutoCAD can be transferred to other computer programs for further analysis and testing.

AutoCAD is a software program that allows us to build different AutoCAD drawings, CAD designs simply and professionally. Computer aided drafting and design (CADD) has brought about revolutionary changes in the field of mechanical engineering drawings. A technology that replaces manual AutoCAD drafting on paper by allowing us to formulate projects on screen, CAD drafting has reduced the efforts and increased the efficiency considerably.

WHY TO USE AUTOCAD:

CAD provides us with a wide range of benefits, in most cases; the result of using CAD is increased accuracy and productivity. First of all, the computer offers much higher accuracy than the traditional methods of drafting and design. Traditionally, drafting and detailing are the most expensive cost elements. With CAD systems, such as AutoCAD, the tedious drafting and detailing tasks are simplified through the use of many of the CAD geometric construction tools, such as grids, snap, trim and auto – dimensioning. Dimensions and notes are always legible in CAD drawings and in most cases, CAD systems can produce higher quality prints compared to traditional hand drawings.

CAD also offers much needed flexibility in design and drafting. A CAD model generated on a computer consists of numeric data that describe the geometry of the object. This allows the designers to see something tangible and to interpret the ramifications of the design. In many cases, it is also possible to simulate operating conditions on the computer and observe the results. Any kind of geometric shape stored in the database can be easily duplicated. For large and complex designs and drawings, particularly those involving similar shapes and repetitive operations, CAD approaches are very efficient and effective. Making changes to a CAD database is generally much faster than making changes to a traditional hand drawing. Only the affected components

of the design need to be modified and the drawings can be plotted again. In addition, the greatest benefit is that, once the CAD model is created, it can be used over and over again. The CAD models can also be transferred into manufacturing equipment such as machining centers, lathes, mills, or rapid prototyping machines to manufacture the product directly.

CAD does not replace every design activity. CAD will supplement our skills to ensure that the best design is obtained.

HARDWARE FOR CAD:

Hardware of a computer system includes all the physical equipment or devices associated with it. A system for CAD may contain different combinations of equipment or devices (hardware) regardless of the system application. A specific combination selected depends largely on the needs of the user. Generally, every equipment comes under any one of the following groups.

1. Central processing unit (CPU)
2. Peripherals
    a. Input devices
    b. Secondary storage devices
    c. display devices
    d. Output devices

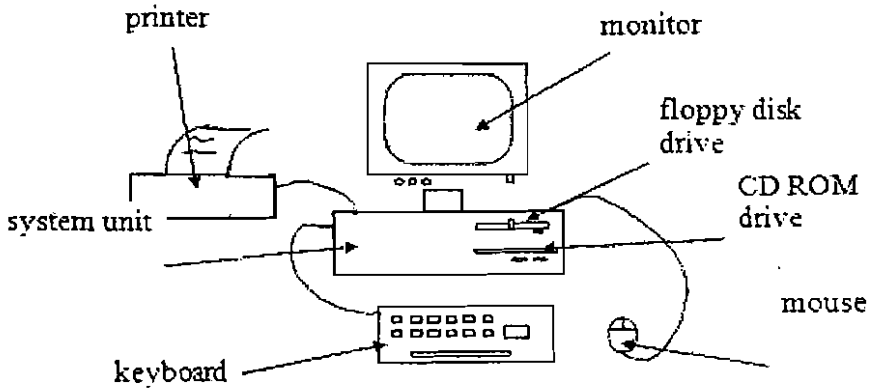


FIG: Computer pheripharals.

CENTRAL PROESSING UNIT (CPU):

The central processing unit (CPU) is the most important part of a personal computer hardware, since it is the 'brain' and all the processing are done in this unit.

This is an integrated circuit(IC) and sometimes referred to as a microprocessor. The CPU of a PC consists of the following sections.

1. Primary storage section
2. Arithmetic logic unit (ALU)
3. Computer section

RAM and ROM:

The primary storage component used in CPU is often referred to as Random Access Memory (RAM) chip. In a RAM, programmes can be stored, altered or retrieved. But once the power supply is switched off, the data and programmes are lost. But the control instructions that cause the machine to operate should be available at any time as a permanent storage. This permanent storage element is called Read only Memory (ROM), which is also a microprocessor (chip). ROM chips retain data when the power goes off. But it is a reads only memory so that it cannot accept any input data or instructions from the computer user.

INPUT DEVICES:

Input device is a mechanism used to present data to the processing unit in a readable form to the computer. Although key board is a common input device, there are other devices also in use. A list of the input devices generally used in the CAD systems are given below

1. Keyboard
2. Mouse
3. Graphic Tablet
4. Automatic scanners
5. Miscellaneous input scanners

SECONDARY STORAGE DEVICES:

In a computer there are two types of storage (memory).

1. Primary storage (inside CPU)
2. Secondary storage (inside peripherals)

   The secondary storage devices are generally fitted inside the system cabinet itself as drivers. The classification of the secondary storage devices is given below.

FLOPPY DISKS:

The new member of the floppy disk family is the compact 3.5 inch size disk called micro disk. They are packed in a non-bendable shirt pocket sized plastic case. The storage capacity is 1.44 MB.

HARD DISK:

Hard disks are rigid aluminum platens with a magnetic coating similar to floppy disks. They spin inside an air tight enclosure. The rotation may be as fast as 3600 rpm, so that storage and retrieval are faster. The storage capacity is in the order of Giga (10^9) bytes.

Hard disks are kept inside the system cabinet of a personal computer. For a CAD station, hard disk of higher storage capacity is preferred in order to keep large number and size of drawing files as well as the software's.

COMPACT DISK (CD):

The optical disks (called CD) used for storing data, are made of transparent glass or plastic and has a reflective metal film coating. Information is stored in the digital form by focusing a laser beam at the desired location.

Optical compact disks are generally available as 'read only memory' in computers and hence they are called as CD ROM. The storage density of CD ROM is about 800 MB and the diameter is 4.7 inches only. There are now optical disks available which can be written many times. They are called re-writable CD.

DISPLAY DEVICES:

The most common type of display unit used in computer is the Cathode Ray Tube (CRT). This can be compared with a television screen. The other modern method which is on the way to common use is liquid crystal display (LCD). These display units are called monitors. The size of the screen ranges from 12 to 20 inches.

OUTPUT DEVICES:

After preparing drawings with the help of monitor and saving them in the hard disk, they have to be converted into permanent drawings on sheets. The devices used to take the output of a CAD station is called printer or plotter. Lists of them are given below.

1. Dot matrix printers
2. Electro-mechanical pen plotters
   a. Drum plotter
   b. Flat bed plotter
3. Laser jet printers
4. Ink jet printers
5. Miscellaneous output devices

The dot matrix printer can produce only poor quality output. At the same time a laser printer of higher resolution can give prints of excellent finish. Ink jet printers and plotters can produce reasonably good quality drawings of A4 to A0 size, if required.

CAD WORKSTATION:

The workstation is a visible part of the CAD system which provides interaction between the operator and the system. There are three elements used by work station:

1. Graphic display terminal
2. Input unit
3. Output devices

ADVANTAGES OF COMPUTER AIDED DRAFTING:

There are many advantages of computer aided drafting system over the traditional drafting system. A few advantages are listed over here.

- ✓ The computer has tremendous speed and it has almost unlimited storage and rapid recall capabilities. This results in reduction of drafting labour and drafting time.
- ✓ The drawing can be stored in 'database' for future use by different programmers for variety of applications.
- ✓ With the ability to interact with the computer, we can quickly correct a drawing error and see a revised picture on the graphic screen.
- ✓ Visual modeling of any object or engineering component is possible.
- ✓ When we use computer aided drawing for a drawing containing the same component at several places, we can draw the component once and then insert it wherever needed.
- ✓ Color graphics help to display more distinct information on the screen, highlight certain important features, etc.

- ✓ The computer aided drawing can be exported to commercial analysis software for analysis purpose.
- ✓ Computer aided design and drafting is used in the creation, modification, analysis and optimization of designs for improved engineering productivity. An engineer can try out different design ideas by just sitting at the terminal without making any prototype.

APPLICATIONS OF COMPUTER AIDED DRAFTING:

Computer aided drafting applications in various fields of engineering are given here.

- ✓ Mechanical: Design of machine elements, CNC machine tools, Robotics.
- ✓ Automotive: Kinematics, Hydraulics, and Steering.
- ✓ Electrical: Circuit layout, Panel design, and control system.
- ✓ Electronics: Schematic diagrams of PCs, Ics, etc.
- ✓ Communication: Communication network, satellite transmitting pictures, T.V. telecasting.
- ✓ Civil: Mapping, contour plotting, building drawing, and structural design.
- ✓ Architectural: Town planning, interior decorations, multi storied complex.
- ✓ Aerospace: Design of spacecraft, flight simulator, lofting, etc.

There are various processes which can be performed by use of computer in the drafting process.

1. Automated Drafting: This involves the creation of hard-copy engineering drawings directly from CAD data base. Drafting also includes features like automatic dimensioning, generation of cross-hatched areas, scaling of the drawing and the capability to develop sectional views and enlarged views in detail. It has ability to perform transformations of image and prepare 3D drawings like isometric views, perspective views etc.
2. Geometric Modeling: Geometric modeling is concerned with the computer compatible mathematical description of the geometry of an object. The mathematical description allows the image of the object to be displayed and manipulated on a graphics terminal through signals from the CPU of the CAD system. The software that provides geometric modeling capabilities must be designed for efficient use both by the computer and the human image of the

object on the CRT screen of the ICG system by supplying following three types of commands to the computer.

GETTING STARTED WITH AUTOCAD:

How to start AutoCAD depends on the type of workstation and the particular software configuration we are using. With most windows systems, we may select the AutoCAD option on the start menu or select the AutoCAD icon on the desktop. The program takes a while to load. Eventually, the AutoCAD 2011 drawing screen and the AutoCAD today startup dialog box will appear on the screen.

AUTOCAD 2011 SCREEN LAYOUT:

The default AutoCAD drawing contains the pull – down menus, the standard tool bar, the object properties toolbar, the draw toolbar, the modify toolbar, the command prompt area, the status bar and the AutoCAD Active assistance.

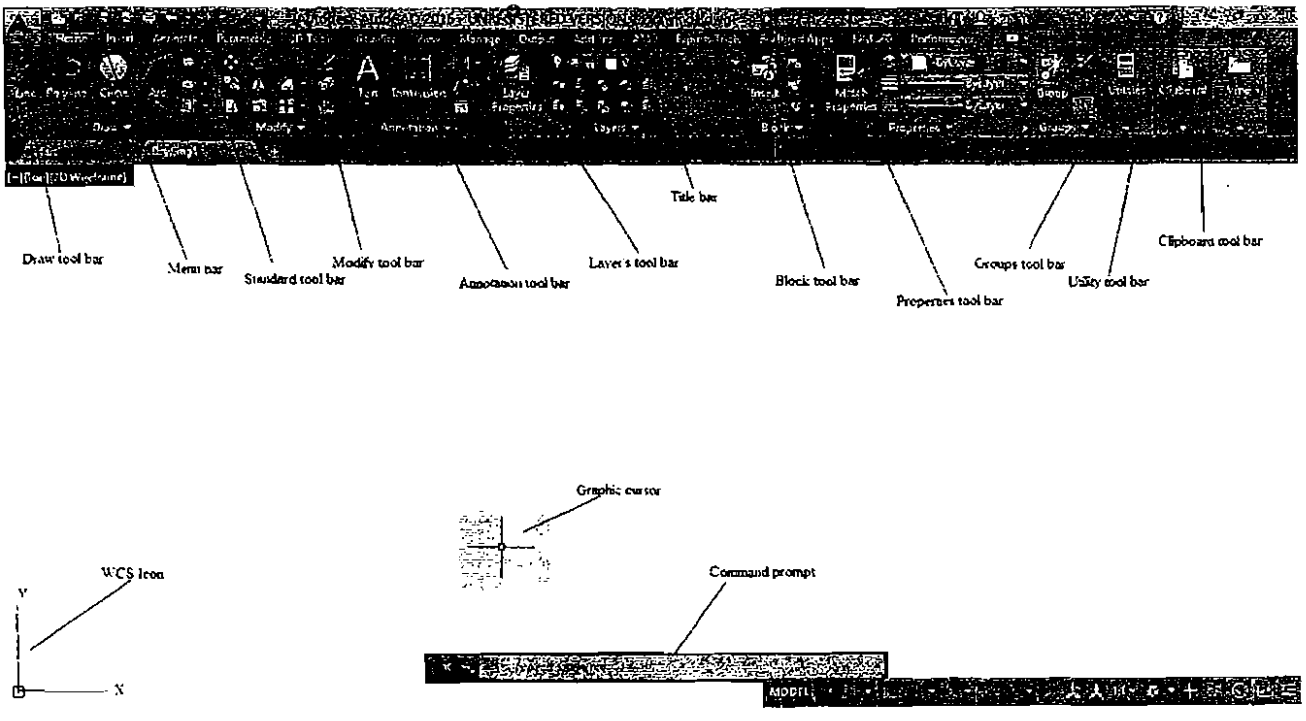


FIG: AutoCAD screen layout.

PULL DOWN MENUS:

The pull – down menus at the top of the main windows contain operations that we can use for all modes of the system.

STANDARD TOOLBAR:

The standard toolbar at the top of the AutoCAD window allows us quick access to frequently used commands. We can customize the toolbar by adding and removing sets of options or individual commands.

FIG: AutoCAD standard toolbar.

OBJECT PROPERTIES TOOLBAR:

Object properties toolbar contains tools to help manipulate the graphical object properties, such as color, line type and layer.

GRAPHICS WINDOW:

Graphics window is the area where models and drawing are displayed.

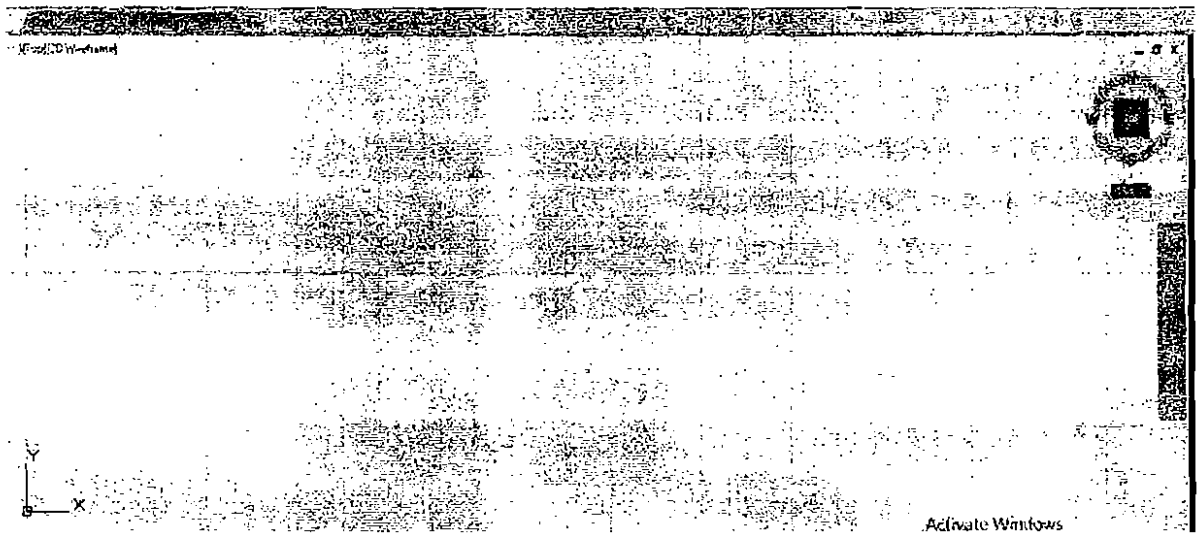


FIG: Graphic window.

GRAPHICS CURSOR:

Graphics cursor shows the location of the pointing device in the graphics window. The coordinates of the cursor are displayed at the bottom of the screen layout. The cursors appearance depends on the selected command or option.

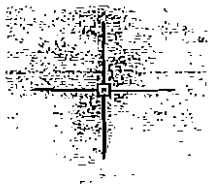


FIG: Graphic cursor.

COMMAND PROMPT AREA:

The bottom section of the screen layout provides status information for an operation and it is also the area for data input.

FIG: Command prompt area.

DRAW TOOLBAR AND MODIFY TOOLBAR:

Additional toolbars are available in AutoCAD 2002 and contain groups of buttons that allow us to pick commands quickly, without searching through a menu structure. The draw toolbar and modify toolbar contain icons for basic draw and modify commands.

[ESC] – CANCELLING COMMAND:

The [Esc] key is used to cancel a command in AutoCAD. The [Esc] key is located near the top – left corner of the keyboard. Sometimes, it may be necessary to press the [Esc] key twice to cancel a command. It depends on where we are in the command sequence. For some commands, the [Esc] key is used to exit the command.

LEAVING AUTOCAD:

To leave AutoCAD 2002, use the left – mouse button and click on File at the top of the AutoCAD 2002 screen window, then choose Exit from the pull – down menu or type QUIT in the command prompt area.

THE COORDINATE SYSTEM

The coordinate system is a method of locating points in the drawing area. It enables us to locate points by specifying distances from a fixed reference point. One can locate a point by giving its distance in the horizontal direction, vertical direction, measuring along an angle, etc.

The coordinate system is available when a function requires data input in the form of point locations. We may use it while drawing, editing or any time you need to locate a point. The most common coordinate systems are as follows:

- Cartesian coordinates
- Polar coordinates

Cartesian Coordinates

Cartesian coordinates is a rectangular system of measurement that enables us to locate points with the help of horizontal and vertical coordinates. The horizontal values, called X-coordinates, are measured along the X-axis. The vertical values, called Y-coordinates, are measured along the Y – axis. The intersection of the X- and Y-axes is called the origin point, which represents the 0, 0 location of the coordinate system.

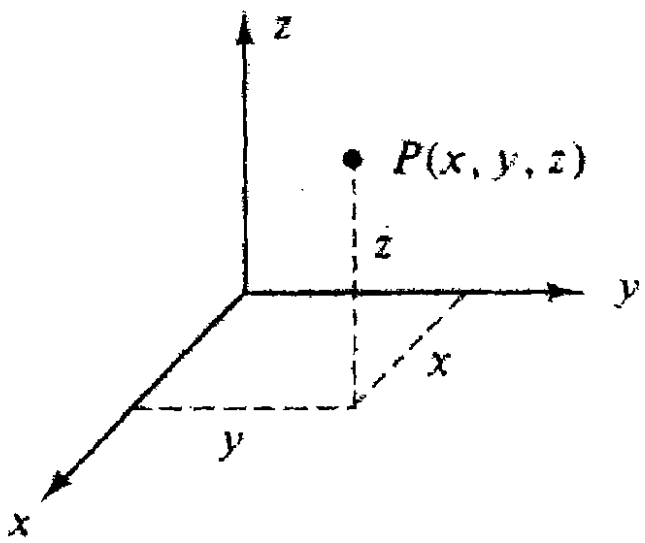


FIG: Cartesian coordinate system.

The positive X values are measured to the right and the positive Y values are measured above origin point. The negative X and Y values are measured to the left and below. To enter a coordinate, you need to enter both the X and Y values separated by a comma (X, Y).

Polar Coordinates

Polar coordinates allow us to define a point by specifying the distance and the direction from a given point. This mode of measurement is quite helpful in working with angles. To draw a line at an angle, we need to specify how long a line we want to draw and specify the angle.

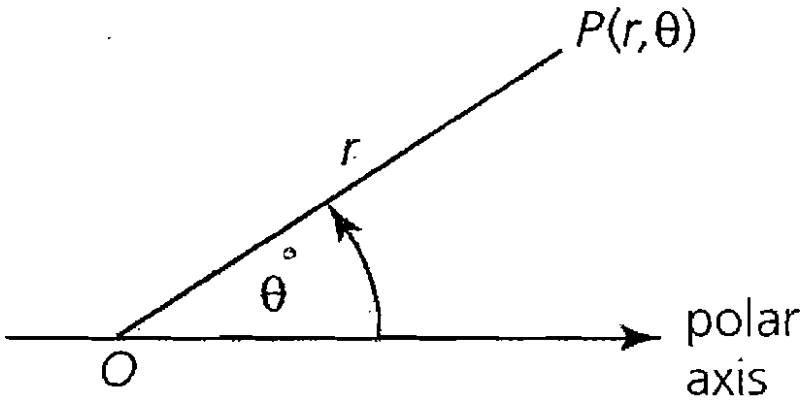


FIG: Polar coordinate system.

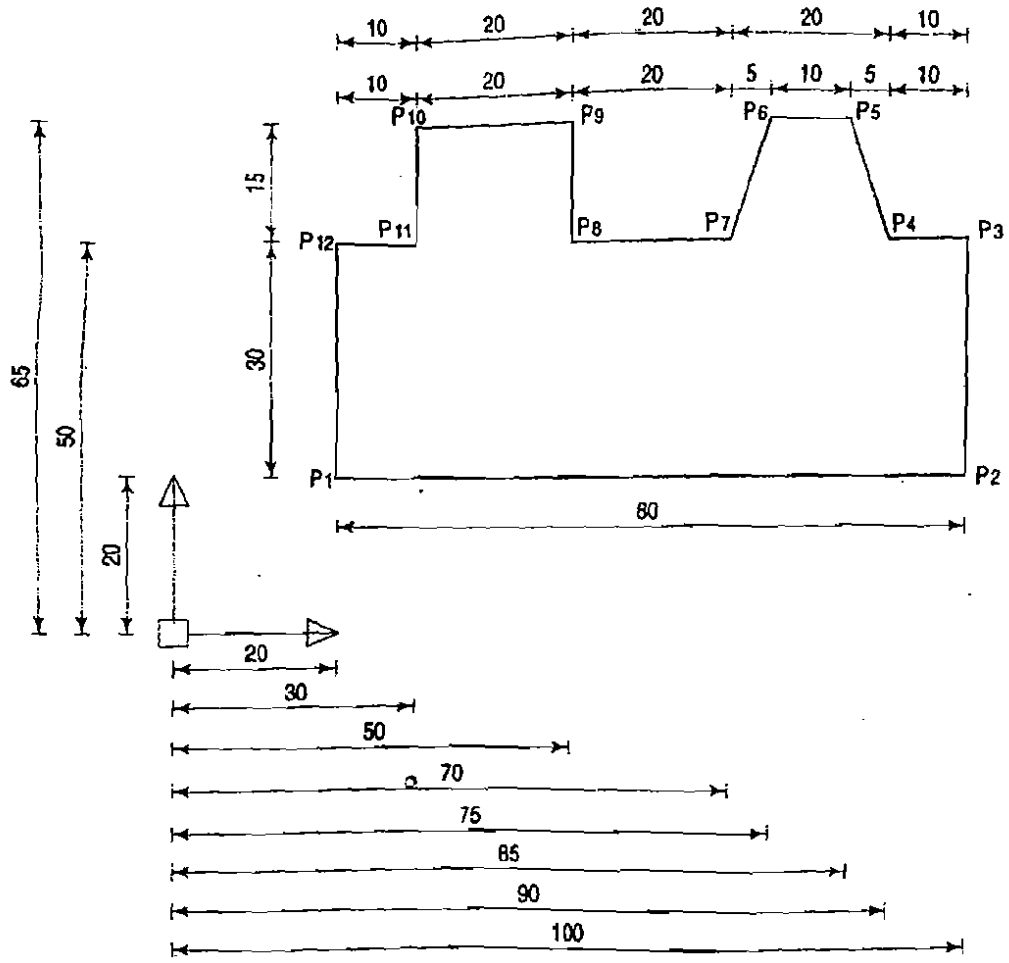
Different Systems to Enter Coordinates:

Cartesian or polar coordinate values can be entered in two formats:

- Absolute
- Relative

Absolute format is a way of measuring distances from a fixed reference location (origin point), which is the 0, 0 location of the coordinate system. Consider this point to be stationary all the time. In some CAD programs this point remains visible at the left bottom corner of the drawing area, while in others it is invisible.

We can use this point as a reference to measure any distance in the drawing. Absolute coordinates are primarily used to adjust the alignment of diagrams in a drawing, to align one drawing with another or to make plotting adjustments.

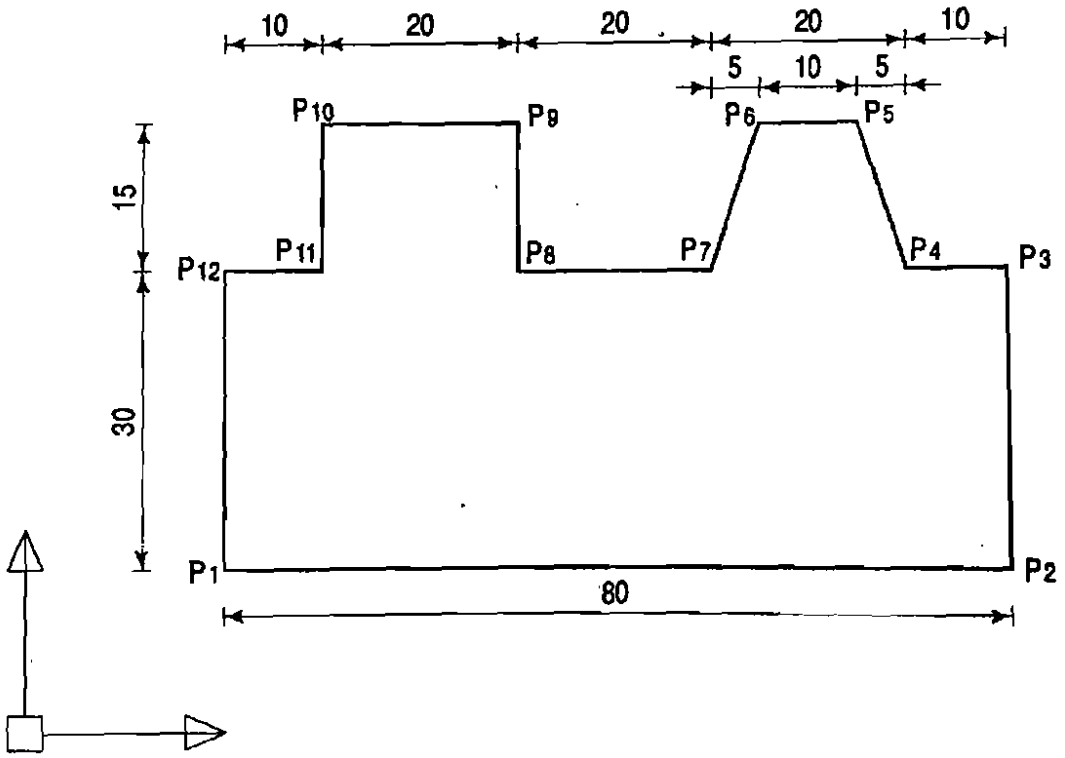


ABSOLUTE COORDINATES OF POINTS P_1 TO P_{12}

P_1 = 20,20	P_4 = 90,50	P_7 = 70,50	P_{10}= 30,65
P_2 = 100,20	P_5 = 85,65	P_8 = 50,50	P_{11}= 30,50
P_3 = 100,50	P_6 = 75,65	P_9 = 50,65	P_{12}= 20,50

FIG: Absolute coordinate format.

Relative format is a way of measuring distances from the last point entered. All measurements are taken the same way as the absolute coordinates; with the only difference being that the relative coordinates are measured from the last point entered instead of the origin point. When a point is entered, it becomes the reference for entering the next point and so on. This mode of measurement is frequently used for drawing because it is always convenient to place the drawing components relative to each other rather than a fixed reference point.



RELATIVE COORDINATES OF POINTS P_1 TO P_{12}

P1 = 20,20	P_4 = @-10,0	P_7 = @-5,-15	P_{10} = @-20,0
P2 = @80,0	P_5 = @-5,15	P_8 = @-20,0	P_{11} = @0,-15
P3 = @0,30	P_6 = @-10,0	P_9 = @0,15	P_{12} = @-10,0

FIG: Relative coordinate format.

Examples:

Cartesian Coordinates

- Input as either Absolute or Relative Coordinates
  - — Absolute X, Y
  - — Relative @X,Y

Polar Coordinates (Vector Coordinates)

- Used to input a distance and the direction angle
- Format: @Distance<Angle

WORLD COORDINATE SYSTEM:

In this system, the x, y and z co-ordinates of any point are measured from the origin, which is located at the lower left hand corner of the screen. This system is fixed and used in 2D drawings, wire frame models, and surface models.

However, the origin can be relocated/ the x, y and z axes can be re-oriented by the user co-ordinate system (UCS). The UCS is used in 3D drawings. It is also useful for relocating the origin or rotating z and y axes in 2D drawings. UCS command is also used to set a new co-ordinate system by shifting the working plane to the desired location. This makes dimensioning of the object easier. One can also fix the screen size equal to the standard sheet size chosen, by setting the x, y and z co-ordinate to any given scale.

USER COORDINATE SYSTEM IN AUTOCAD:

Designs and drawings created in a CAD system are usually defined and stored using sets of points in what is called world space. In most CAD systems, the world space is defined using a three-dimensional Cartesian coordinate system. Three mutually perpendicular axes, usually referred to as the X, Y, and Z axes, define this system. The intersection of the three coordinate axes forms a point called the origin. Any point in world space can then be defined as the distance from the origin in the X, Y and Z directions. In most CAD systems, the directions of the arrows shown on the axes identify the positive sides of the coordinates.

A CAD file, which is the electronic version of the design, contains data that describe the entities created in the CAD system. Information such as the coordinate values in world space for all endpoints, center points, etc., along with the descriptions of the types of entities is all stored in the file. Knowing that Aut0CAD stores designs by keeping coordinate data helps US understand the inputs required to create entities.

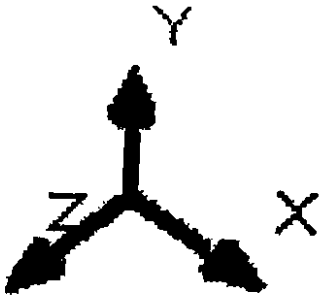


FIG: UCS in AutoCAD.

CHANGING TO THE 2D UCS ICON DISPLAY:

In AutoCAD, the UCS icon is displayed in various ways to help us visualize the

orientation of the drawing plane.

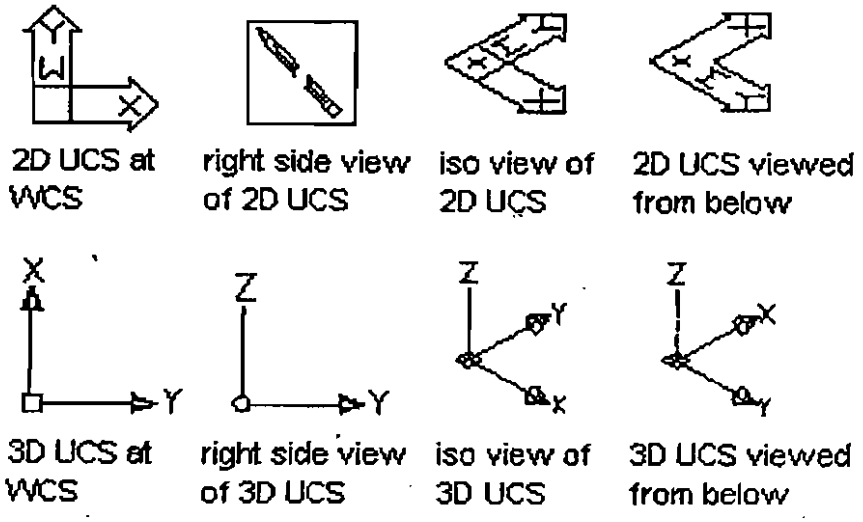


FIG: Types of UCS in AutoCAD.

ABSOLUTE AND RELATIVE COORDINATES

AutoCAD also allows us to use absolute and relative coordinates to quickly construct objects. Absolute Coordinate values are measured from the current coordinate system's origin point. Relative coordinate values are specified in relation to previous coordinates. In AutoCAD, the absolute coordinates and the relative coordinates can be used in conjunction with the Cartesian and polar coordinate systems. By default, AutoCAD expects us to enter values in absolute Cartesian coordinates, distances measured from the current coordinate system's origin point. We can switch to using the relative coordinates by using the @ symbol. The @ symbol is used as the relative coordinates specifier, which means that we can specify the position of a point in relation to the previous point.

DEFINING POSITIONS:

In AutoCAD, there are five methods for specifying the locations of points when we create planar geometric entities.

- ✓ Interactive method: Use the cursor to select on the screen.
- ✓ Absolute coordinates (Format: X, Y): Type the X and Y coordinates to locate the point on the current coordinate system relative to the origin.
- ✓ Relative rectangular coordinates (Format: @X, Y): Type the X and Y coordinates relative to the last point.

- ✓ Relative polar coordinates (Format: @Distance<angle) Type a distance and angle relative to the last point.
- ✓ Direct Distance entry technique: Specify a second point by first moving the cursor to indicate direction and then entering a distance.

USES OF FUNCTION KEYS:

Eleven function keys of the key board have special uses in AutoCAD. They are toggle switches to make the function On/OFF by pressing.

F1 This key opens window for "Help: User documentation". Help provides complete information for using AutoCAD.

F2 Opens AutoCAD text window of commands used in the command area.

F3 Opens or closes running OSNAP

F4 Makes the Tablet ON/OFF (if calibrated)

F5 This key changes the "Isoplane" of isometric view to left, top and right sides

F6 By pressing this key coordinate value display changes to active or inactive, or changes the display mode from rectangular to polar coordinates

F7 This turns Grid ON or OFF

F8 This turns the Ortho ON or OFF

F9 This turns the Snap On or OFF

F10 This turns the Polar ON or OFF

F11 This turns the OSNAP tracking ON or OFF

1. DRAWING OBJECTS USING BASIC DRAWING COMMANDS

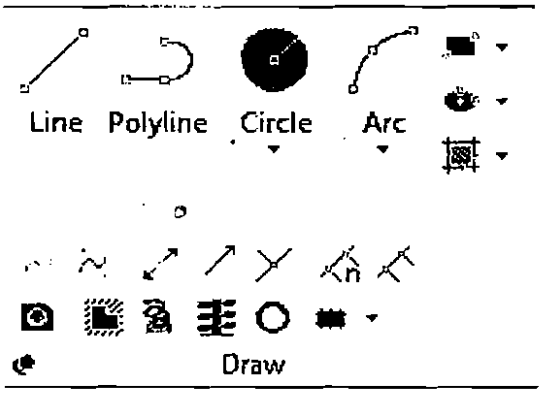
Date: 16/8/16

Aim: To learn and practice the commands used to draw different objects using AutoCAD 2007 like line, poly line, rectangle, arc, circle, spline etc.

Commands used:

Draw tool bar:

The Draw commands can be used to create new objects such as lines and circles. Most AutoCAD drawings are composed purely and simply from these basic components. A good understanding of the Draw commands is fundamental to the efficient use of AutoCAD.



DRAWING ENTITIES

An entity is a drawing element, namely, point, line, circle, am, etc. AutoCAD provides a set of entities using DRAW ENTITY COMMANDS for constructing the drawing.

1. Drawing Entity - POINT

The point command places a point in the drawing. Enter the command POINT.

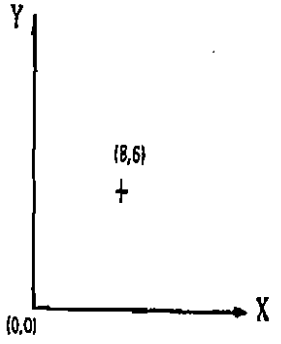
Example:

Plot a point at the location (8, 6).

Command: POINT

Point: 8, 6

This places the given point in the drawing at location (8,6).



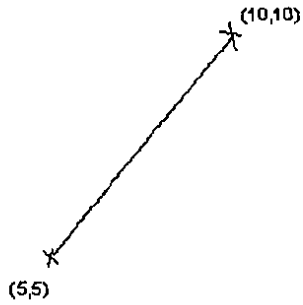
2. Drawing Entity - LINE

Lines can be drawn by anyone of the following three methods using LINE command.

a) Using Absolute Co-ordinates:

Example:

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



Command: LINE

From point: 5, 5 (Select the point by mouse or enter the Co-ordinates by keyboard)

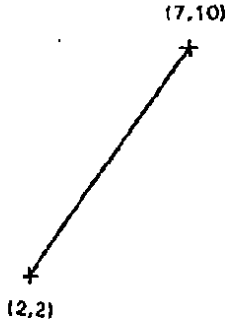
To point: 10, 10

To point: (Press ENTER)

b) Using relative Co-ordinates:

Example:

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



Command: LINE

From point: 2, 2

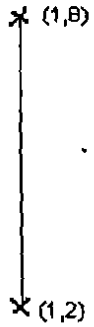
To point: @5, 8

To point: (press ENTER)

c) Using Polar Co-ordinates:

Example:

Draw a line from point (1, 2) to a length of 6 units at 90 degree.



Command: LINE

From point: 12

To point:@6 < 90

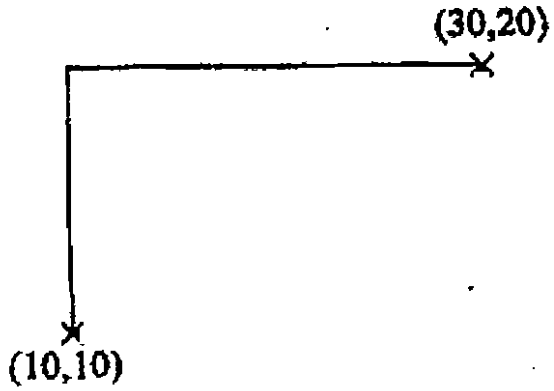
To point :(Press ENTER)

3. Drawing Entity - PLINE

A polyline is a connected sequence of line and arc segments. The command is PLINE.

Example:

Draw a thick line of width 2 units from point to point using PLINE command.



4. Drawing Entity - RECTANGLE

A rectangle is a polyline based on two opposite corner points called diagonal points. (A polyline is a connected sequence of line / arc segments)

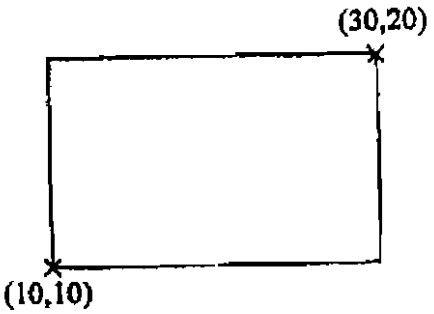
Example:

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

Command: RECTANGLE

First corner: 10, 10

Second Corner: 30, 20



5. Drawing Entity - CIRCLE

Circle can be drawn by any one of following five methods using CIRCLE command

a) Using Centre and Radius:

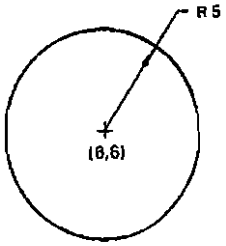
Example:

Draw a circle with centre (6, 6) and radius 5 units.

Command: CIRCLE

3P/2P/TTR/ <Centre point>: 6,6

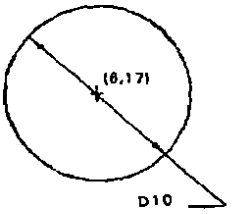
Diameter/ <Radius>:5



b) Using Centre and Diameter:

Example:

Draw a circle with centre (6, 17) and diameter 10 units.



Command: CIRCLE

3P/2P/TTR <Centre point>: 6, 17

Diameter/ < Radius>: D

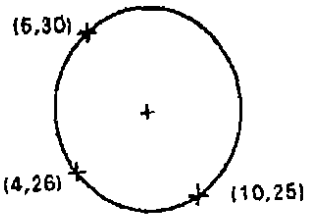
Diameter: 10

c) Using Three given Points: (3P)

Example:

Draw a circle using the given 3points: (5,30), (4,26), (10,25).

By entering 3 given points to be on the circumference of the circle:



Command: CIRCLE

3p/2P/TTR/ <Centre point>: 3P

First point: 5, 30

Second point: 4, 26

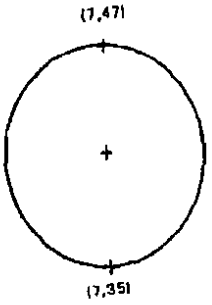
Third point: 10.25

d) Using Two given Points: (2 P)

Example:

Draw a circle using the given 2 points: (7, 35), (7, 47).

By entering 2 end points of the circle diameter:



Command: CIRCLE

3P12P/flR/ <Centre point>:2P

First point on diameter: 7, 35

Second point on diameter: 7, 47

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

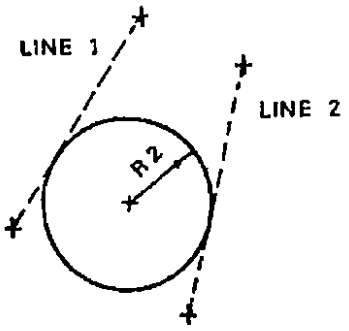
We can draw a circle by specifying two lines or two circles or a combination of line and circle, and also radius of circle. These two lines or two circles act as tangents to the circle.

Example:

(i) Draw a circle with radius 2 units and two existing lines as tangents.

Take: For line 1 - From point (16, 4) to point (19, 9).

For line 2 - From point (20, 2) to point (21, 7).



Command: CIRCLE

3P/2P/TTR/ <Centre point>: TTR

Enter Tangent Spec: line 1 (pick up using mouse): I

["Spec" means Specification]

Enter Second Tangent Spec: line 2 (pick up using mouse)

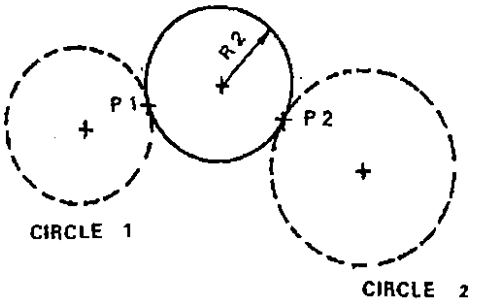
Radius: 2

(ii) Draw a circle with radius 2 units and two existing circles as tangents.

Take

For circle I - Centre (15, 15) and radius 2 units.

For circle 2 - Centre (23, 14) and radius 2.5 units.



Command: CIRCLE

3P/2P/TTR/ <Centre point> :TTR

Enter Tangent Spec: Point P1 (on circle 1)

Enter Second Tangent Spec: Point P2 (on circle 2)

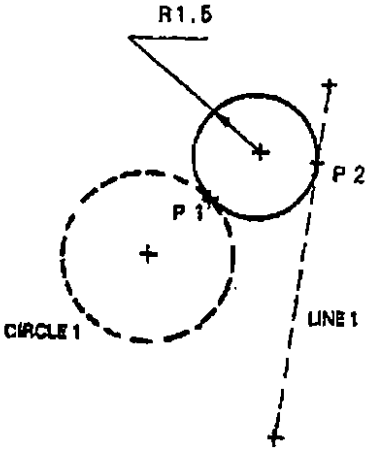
Radius: 2

(iii) Draw a circle with radius 1.5 units and an existing line and a circle as tangents.

Take:

For line 1 - From point (21, 20) to point (22, 30)

For circle I - Centre (18, 25) and radius 2 units.



Command: CRCLE

3 P/2P/TTR/ <Centre point>: TTR

Enter Tangent Spec: Point Pl (on circle 1)

Enter Second Tangent Spec: Point P2 (on tine I)

Radius: 1.5

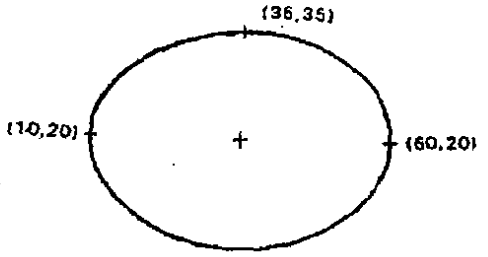
6. Drawing Entity - ELLIPSE

Ellipse can be drawn by any one of following four methods using ELLIPSE command.

a) Using First axis end points and other axis distance:

Example:

Draw an ellipse using major axis endpoints (10, 20), (60, 20) and minor axis end point (35, 35).



Command: ELLIPSE

<Axis endpoint1 >/Center: 10, 20

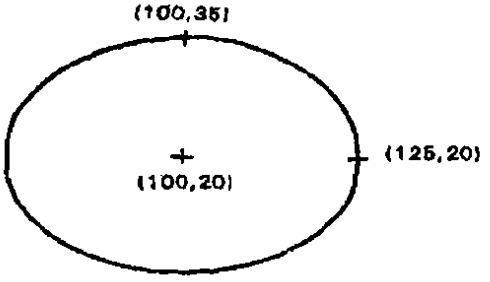
Axis endpojnt2: 60, 20

<Other axis distance> /Rotation: 35, 35

b) Using Centre of ellipse axis, End point and other axis distance:

Example:

Draw an ellipse with centre (100, 20), major axis endpoint (125, 20) and minor axis end paint (100.35).



Command: ELLIPSE

<Axis endpoint1 >/ Centre: C

Centre ofEtlipse: 100, 20

Axis endpoint2: 125.20

<Other axis distance> /Rotation: 100, 35

c) Using First axis endpoints and Rotation angle of circle around the axis:

Example:

Draw an ellipse using major axis end points (8, 80), (58, 80) and 55 degree rotation around major axis.

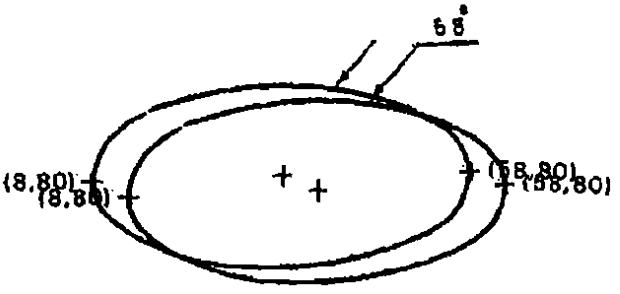
Command: ELLIPSE

<Axis endpoint1 >/Center: 8, 80

Axis endpoint2: 58, 8O

<Other axis distance> /Rotation: R

Rotation around major axis: 55



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

d) Using Centre, End point and Rotation angle of circle around the axis:

Example:

Draw an ellipse with centre (35, 48), major axis endpoint (60, 48) and 65 degree rotation around the major axis.

Command: ELLIPSE

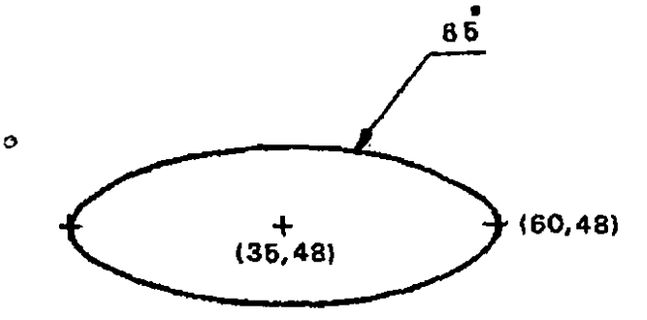
<Axis endpoint! >/Centre: C

Centre of Ellipse: 35, 48

Axis endpoint2: (60, 48)

<Other axis distance> /Rotation: R

Rotation around major axis: 65



7. Drawing Entity – ARC

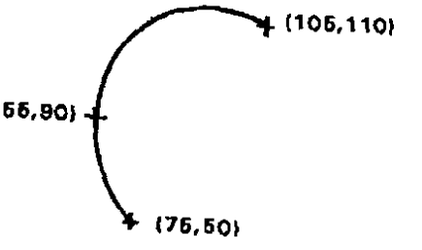
Arcs are partial circles and can be drawn in eight different methods using ARC command.

a) Using Three given Points:

Example:

Draw an arc using the given three points:

(75, 50), (55, 90), (105,110).



Command: ARC

Center/<Start point>:75, 50

Center/End/<Second point>: 55, 90

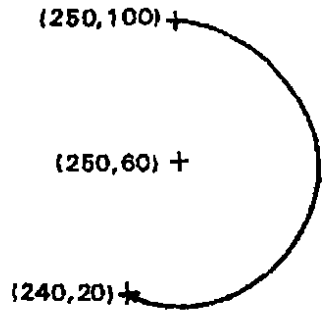
Endpoint: 105,110

b) Using Start point, Centre and Endpoint: (S, C, E)

Example:

Draw an arc using start point (240, 20),

centre point (250, 60) and endpoint (250, 100).



Command: ARC

Center/<Start point> :240,20

Center/End/ <Second point>: C

Center point: 250, 60

Angle/Length of chord! <Endpoint>:250,100

c) Using Start point, Centre and Included Angle: (S, C, A)

Example:

Draw an arc using Stan point (100,190),

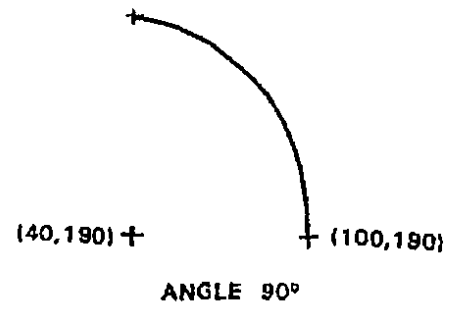
centre point (40,190) and included angle 90 degrees.

Command: ARC

Center/<Start point>: 100, 190

Center/End/ <Second point>: C

Center: 40, 190



NOTE: Positive "included angle" draws arc in clockwise and negative in anti-clockwise direction.

d) Using Start point, Centre and Length of chord :(S, C, L)

Example:

Draw an arc using start point (140.10),

centre point (100, 10) and chord length 45 units.

Command: ARC

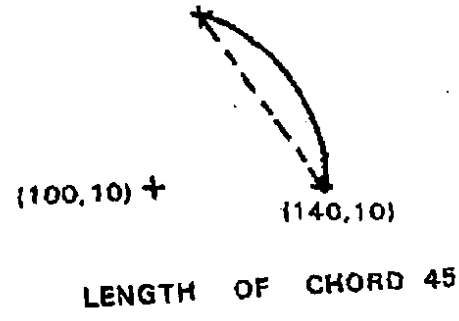
Center/<Start point>: 140, 10

Center/End/< Second point>: C

Center: 100, 10

Angle/Length of chord/ <Endpoint>: L

Length of chord: 45



NOTE: A chord is a straight line connecting an arc's start point and endpoint; always such arcs are drawn in anti-clockwise direction.

e) Using Start point, End point and Radius :(S, E, R)

Example:

Draw an arc using start point (230, 80), endpoint (190, 80) and radius 22 units.

Command: ARC

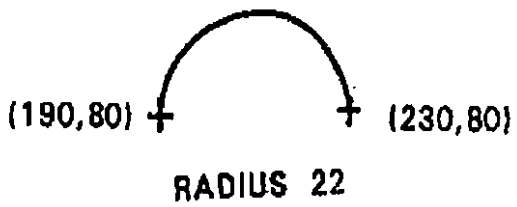
Center/<Start point> :230, 80

Center/End/ <Second point> E

Endpoint: 190, 80

Angle/Direction/Radius/ <Centre point>: R

Radius: 22



f) Using Start point, End point and Included Angle :(S, E, A)

Example:

Draw an arc using start point (300, 60), end point (340,120) and included angle 90 degrees.

Command: ARC

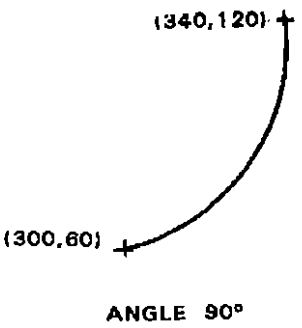
Center/<Start point>:300, 60

Center/End/ <Second point>: E

End point: 340, 120

Angle/Direction/Radius/ <Centre point>: A

Included Angle: 90



NOTE: This type of arc is normally drawn anti-clockwise from start point to end point. If negative sign is specified for included angle, the arc is drawn clockwise.

g) Using Start point, End point and Starting Direction: (S, E, D)

Example:

Draw an arc with start point (40, 170), endpoint (70,230) and direction from start point 120 degrees.

Command: ARC

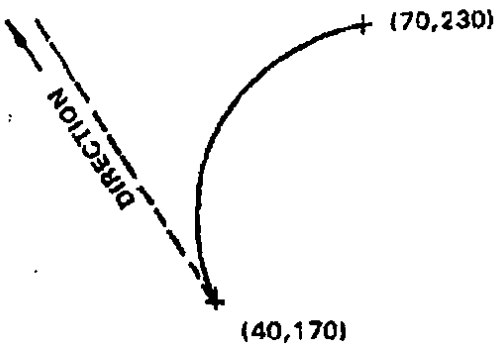
Center/ <Start point>:40,170

Center/End/ <Second point>: E

End point: 70, 230

Angle/Direction/Radius/ <Center point>: D

Direction from Start point: 120



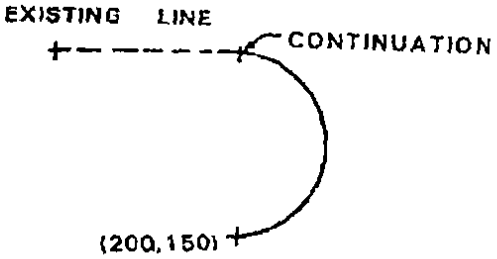
h) Using Line/Arc Continuation:

Example:

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

Take:

Existing line from point (150, 200) to point (200,200).



Command: ARC

Center/< Start point>: (Press ENTER)

End point: 200, 150

NOTE: This command is also used to draw a continuous arc after an arc is drawn.

8. Drawing Entity - POLYGON

The Polygon command draws regular 2D polygons with 3 to 1024 sides. Any polygon can be drawn by the following three methods using POLYGON command.

a) Using Radius of given Circle in which polygon is inscribed:

Example:

Draw a polygon of eight sides with centre (50, 50) inscribed in a circle of radius 40 units.

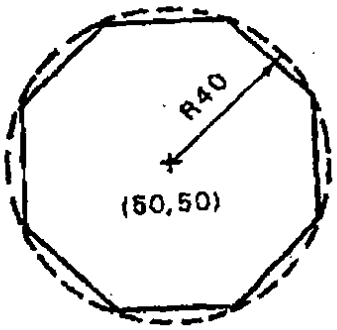
Command: POLYGON

Number of sides: 8

Edge/ <Center of polygon>: 50, 50

Inscribed in circle/Circumscribed about circle (I/C): I

Radius of circle: 40



b) Using Radius of given Circle on which polygon is circumscribed:

Example:

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

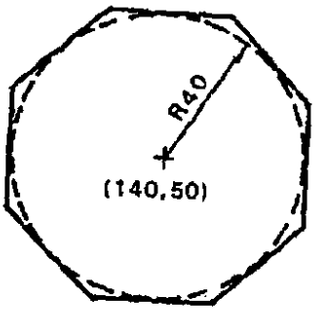
Command: POLYGON

Number of sides: 8

Edge/<Center of polygon>: 140, 50

Inscribed in circle/Circumscribed about circle (I/C): C

Radius of circle: 40



c) Using Edge Method:

Example:

Draw a polygon of ten sides using "edge method." The first end point of the edge is (90,100) and second end point of the edge is (120,100).

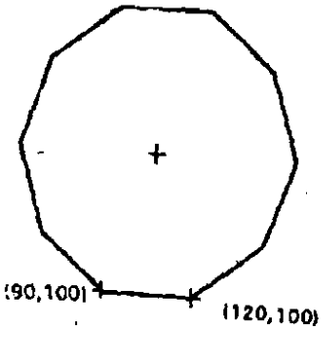
Command: POLYGON

Number of sides: 10

Edge/ <Center of polygon>: E

First end point of edge: 90, 100

Second end point of edge: 120, 100



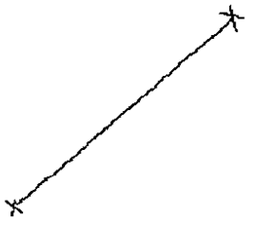
9. Drawing Entity – RAY

The RAY command can create a line starts at a point and continues to infinity in any specified direction.

Command: RAY

Specify starting point: X1, Y1

Specify through point: X2, Y2.



10. Drawing Entity – SPLINE

The SPLINE command can create a curved line with number of curve on it. A SPLINE con be drawn by two commands.

a) Using fit points

Specify the fit points for the curve. The SPLINE is created by passing through all the fit points.

Command: SPLINE

Select method: M, FIT

Specify first point: X1, Y1

Specify next point: X2, Y2

Specify next point: X3, Y3

FIG: Spline using FIT points

b) Using control vertices

Specify the fit points for the curve. The SPLINE is created by passing through all the fit points.

Command: SPLINE

Select method: M, FIT

Specify first point: X1, Y1

Specify next point: X2, Y2

Specify next point: X3, Y3

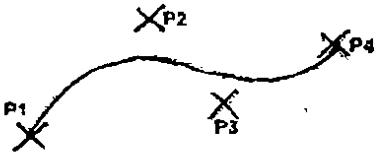


FIG: Spline using CONTROL vertices.

Result: The commands used to draw different objects in AutoCAD like line, poly line, rectangle, arc, circle, spline etc. were learned and practiced.

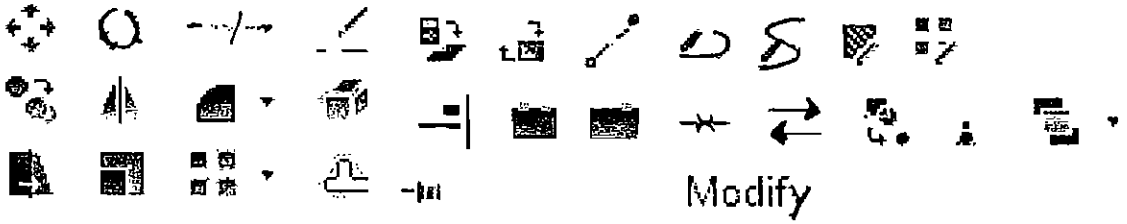
2. DRAWING OBJECTS USING BASIC MODIFYING COMMANDS

Date: 22/8/16

Aim: To learn and practice the commands used to modify different objects using AutoCAD like move, copy, rotate, mirror, trim, region etc.

Commands used:

The MODIFY command is used to modify the existing drawings or complete the drawing easily. AutoCAD drawings are rarely completed simply by drawing lines, circles etc. Most likely you will need to MODIFY these basic drawing objects in some way in order to create the image you need. AutoCAD provides a whole range of modify tools such as Move, Copy, Rotate and Mirror. A good understanding of the Draw commands is fundamental to the efficient use of AutoCAD.



1. Modifying Entity – MOVE

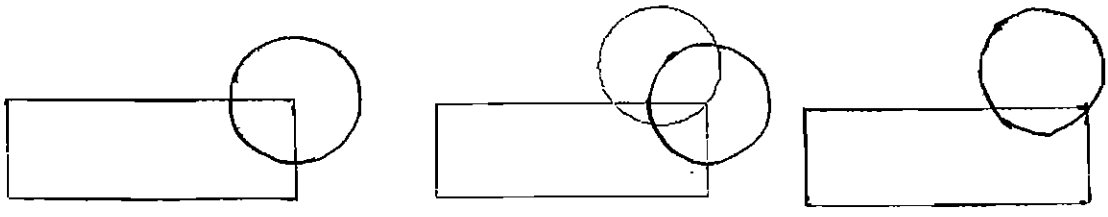
The modify entity MOVE command is used to move an object from one place to another place in the work plane.

Command: MOVE

Select object to move:

Select through point:

Specify through distance.



Before During After

2. Modifying Entity – COPY

The modify entity COPY command copies the selected objects and creates any number of copies on the workspace at specified distance with specified rotation.

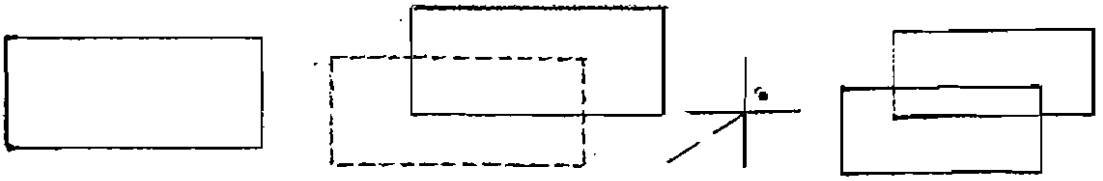
Command: COPY

Select an object to copy:

Select a through point:

Select through distance:

Esc to close command.



Before During After

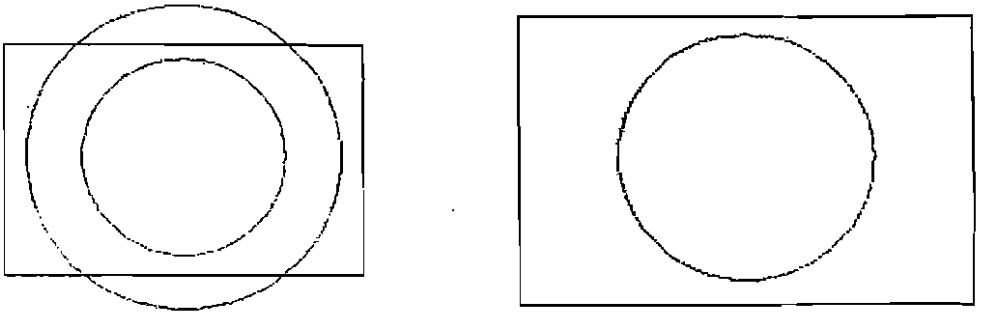
3. Modifying Entity – ERASE

The modify entity ERASE command is used to delete an object or some part of the object from the drawing permanently. It can be also done by selecting an object and pressing delete button on the keyboard.

Command: ERASE

Select an object to erase:

Close the command.



Before After

4. Modifying Entity – MIRROR

The modify entity MIRROR command is used to create a mirror image of the selected object. The symmetric objects can be easily created using this command.

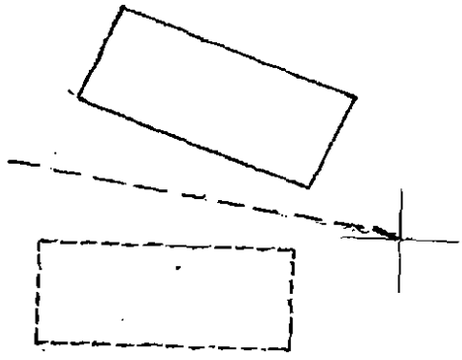
Command: MIRROR

Select an object:

Specify first point of mirror line:

Specify second point of mirror line:

Close the command.



5. Modifying Entity – TRIM

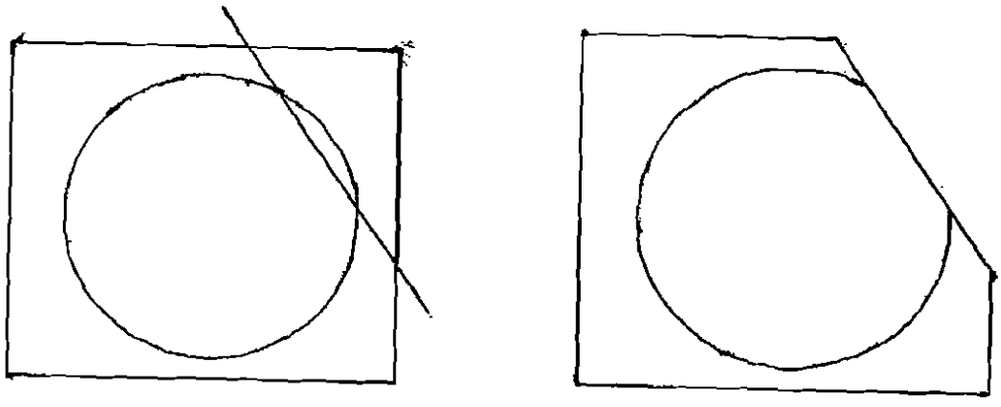
The modify entity TRIM command is remove the unnecessary parts of the drawing corresponding to the other drawing entities. It is also used to meet the edges of one object to the other in the drawings.

Command: TRIM

Select the objects to trim:

Trim unnecessary parts:

Close the command.



Before

After

6. Modifying Entity – OFFSET

The modify entity OFFSET command is used to OFFSET the objects through specified distances. This command duplicates the same object around itself with some distance.

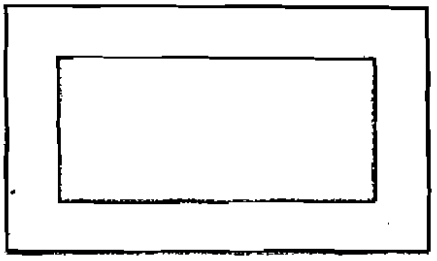
Command: OFFSET

Select the objects to offset:

Specify offset distance:

Specify direction to offset:

Close the command.



7. Modifying Entity – ARRAY

The modify entity ARRAY command creates the selected objects into a number of objects arranged in a specified order. ARRAY command can be used in two different ways.

a) Rectangular array:

The rectangular array is used to create the selected objects into number of objects and arrange them in a rectangular order.

Command: RA

Select the object:

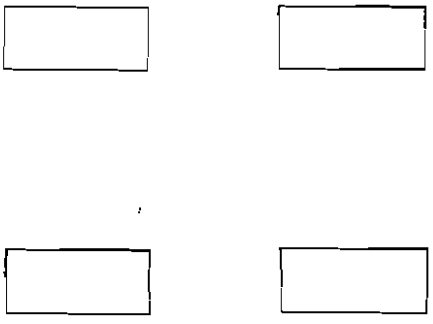
Select number of rows:

Specify distance between two consecutive rows:

Select number of columns:

Specify distance between two consecutive columns:

Close the command.



b) Polar array:

The polar array is used to create the selected objects into number of objects and arrange them in a circular order.

Command: PA

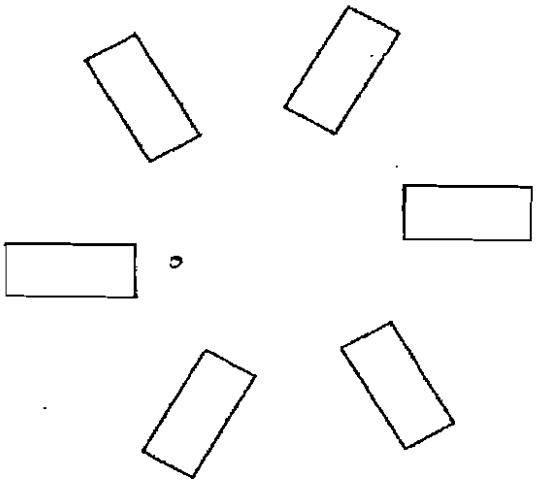
Select the object:

Select number of rows:

Specify distance between two consecutive rows:

Select angle of rotation:

Close the command.



8. Modifying Entity – ROTATE

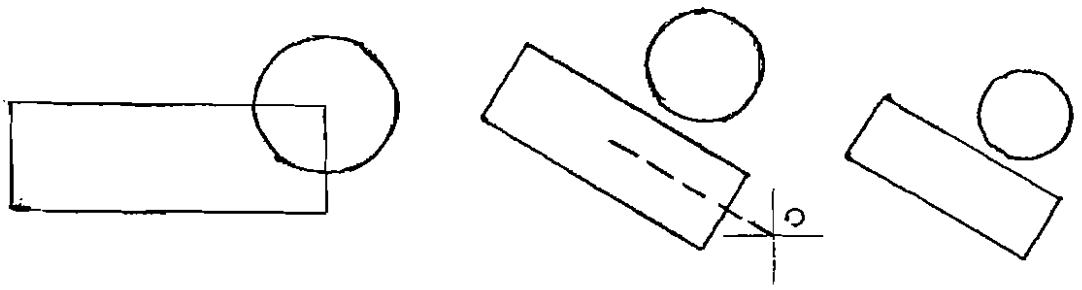
The modify entity ROTATE command is used to rearrange the selected object in the required order or at the specified angle.

Command: ROTATE

Select the object to rotate:

Specify the through point to rotate:

Close the command.



Before During After

9. Modifying Entity – SCALE

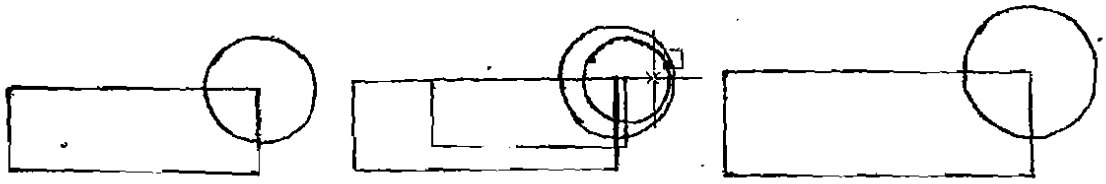
The modify entity SCALE command is used to resize the selected objects to specified scaling ratio. This command is used to view the large objects into a considered size.

Command: SCALE

Select an object to be scaled:

Specify scaling ratio:

Close the command.



Before During After

10. Modifying Entity – STRECH

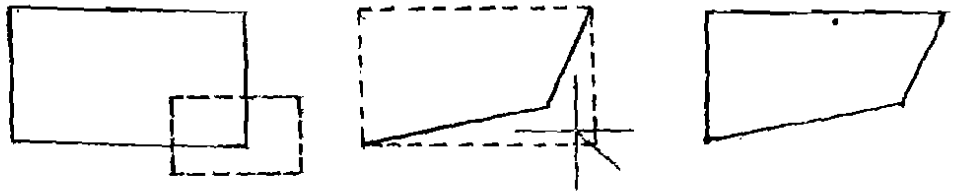
The modify entity STRECH command is used to strech the required object to the required length. This command is also used to make join one entity with the other.

Command: STRECH

Select an object to strech:

Specify the distance or end object:

Close the command.



Select Vertex During Stretch After Stretch

11. Modifying Entity – LENGTHEN

The modify entity LENGTHEN command is used to increase the length of the line command.

Command: LENGTHEN

Select an object to lengthen:

Specify the distance or end object:

Close the command.

Before

After

12. Modifying Entity – EXTEND

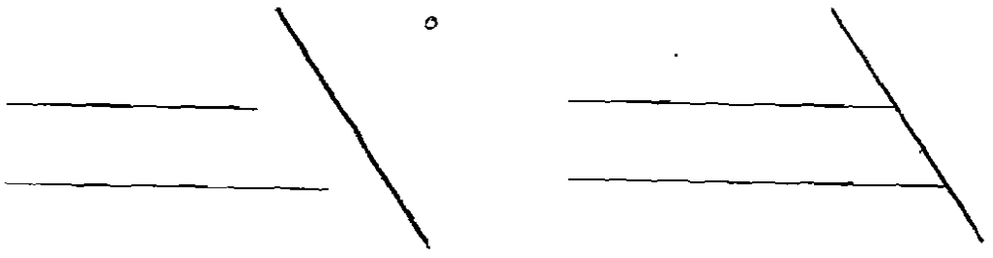
The modify entity EXTEND command is used to extend the required object to the required length. This command is also used to make join one entity with the other.

Command: EXTEND

Select an object to strech:

Specify the distance or end object:

Close the command.



Before

After

13. Modifying Entity – BREAK

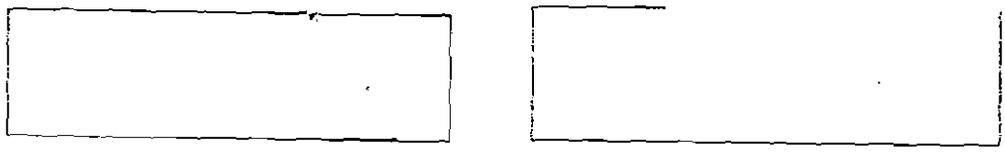
The modify entity BREAK command is used to create breaks in the objects. Any entity can be breaked at any place using this break command.

Command: BREAK

Select an object to break:

Specify break position:

Close the command.



Before

After

14. Modifying Entity – CHAMFER

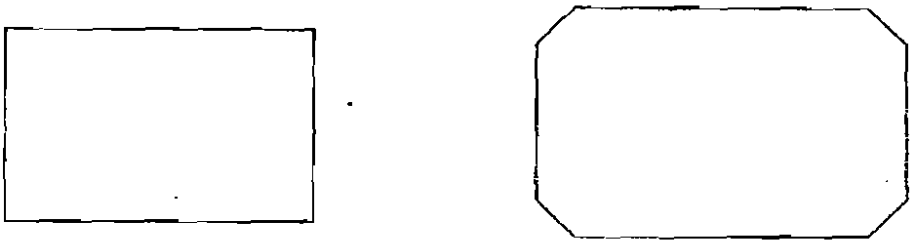
The modify entity CHAMFER command is used to round off the sharp edges of the object.

Command: CHAMFER

Select an end to chamfer:

Select the chamfer radius:

Close the command.



Before After

15. Modifying Entity – FILLET

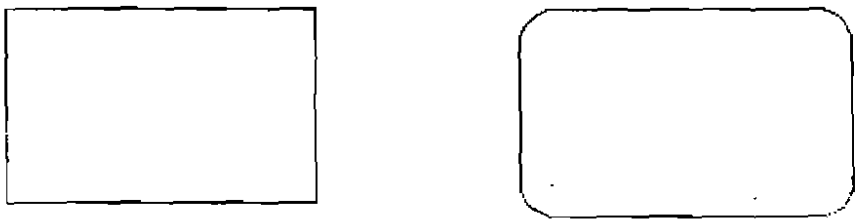
The modify entity FILLET command is used to remove the sharp edges of the object.

Command: FILLET

Select an end to chamfer:

Select the chamfer radius:

Close the command.



Before After

16. Modifying Entity – EXPLODE

The modify entity EXPLODE command is used to return blocks, polylines etc. (which may be composed of a number of component objects) back to their individual component parts. The change has no visible effect.

Command: EXPLODE

Select the object to explode:

Close the command.

17. Modifying Entity – HATCH

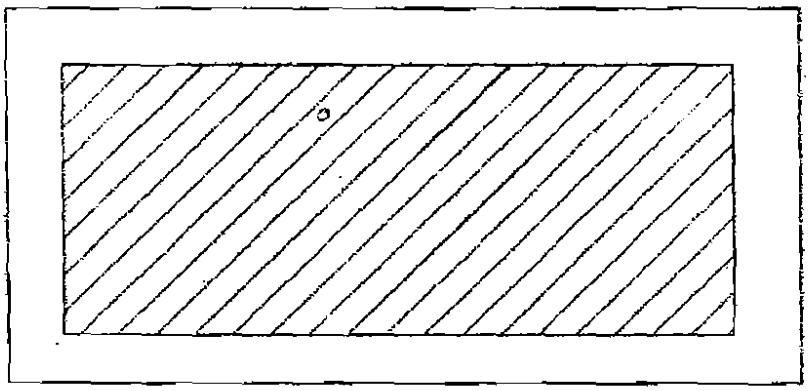
The modify entity HATCH command is used to define hatching to an object. Hatch is used to clearly identify the different parts of an object and different sectional views of object. Different types of hatches can be used by changing properties of hatch.

Command: HATCH

Select the area to hatch:

Edit hatch properties:

Close the command.



Result: The commands used to modify different objects using AutoCAD like move, copy, rotate, mirror, trim, region etc. were learned and practiced.

3. DIMENSIONING OBJECTS USING BASIC ANOTATION COMMANDS

Date: 30/6/16

Aim: To learn and practice the commands used to dimension different objects using AutoCAD.

Dimensioning:

The correct use of AutoCAD's dimension tools is the key to producing clear and concise measured drawings. Any drawing is incomplete without specifying its dimensions. Dimensions are necessary to understand the drawings and also to manufacture the parts in the drawings. In this experiment, we are going to learn how to use different types of dimensioning commands.

1. Linear Dimension command:

The LINENEAR DIMENSION command is used to generate the horizontal and vertical dimensions in the drawing. The dimensions of vertical and horizontal lines are represented in the drawing using this command.

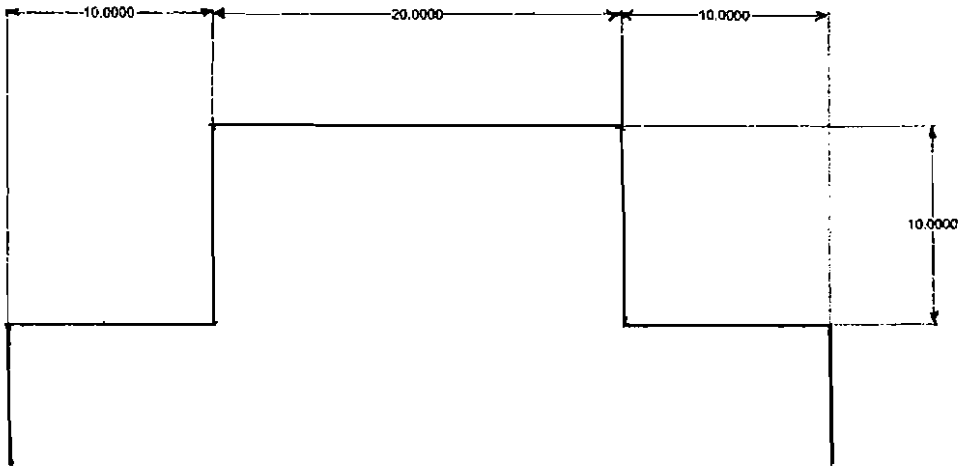
Command: DIMLINEAR

Select a line or first point:

Select end point:

Select dimension direction:

Close the command.



2. Baseline Dimension command:

The BASELINE DIMENSION command is also used to generate the horizontal and vertical dimensions in the drawings. But the only difference is that every dimension is started from a fixed basic line unlike to linear dimension.

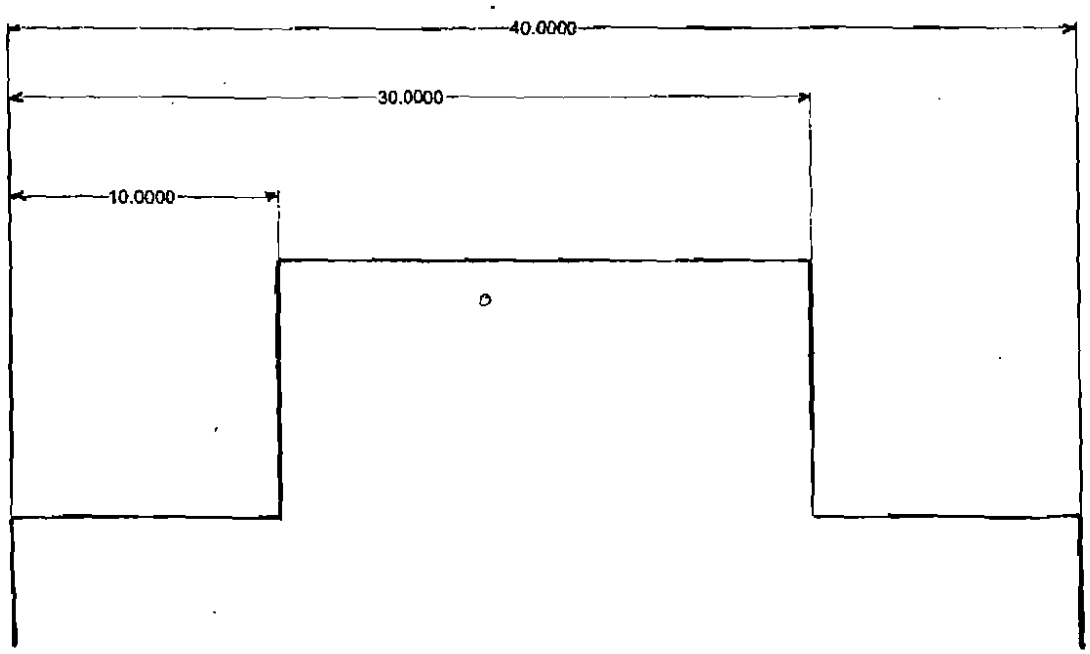
Command: DIMBASELINE

Select the base line:

Select second point for dimension:

Select dimension direction:

Close the command.



3. Aligned Dimension command:

The ALIGNED DIMENSION command is used to generate the dimensions of the aligned lines and aligned portions of the drawings. By using this command, we can generate the dimensions of the aligned lines parallel to them.

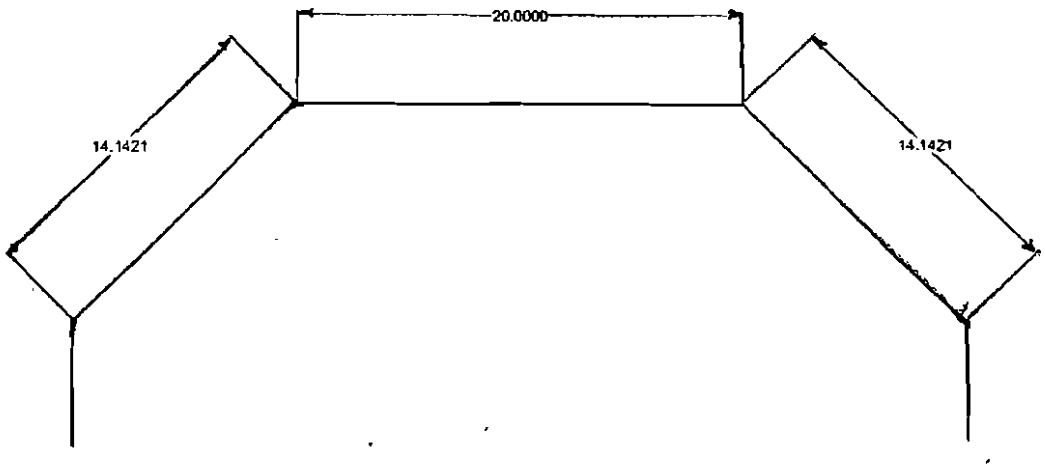
Command: DIMALIGNED

Select a line or first point:

Select end point:

Select dimension direction:

Close the command.



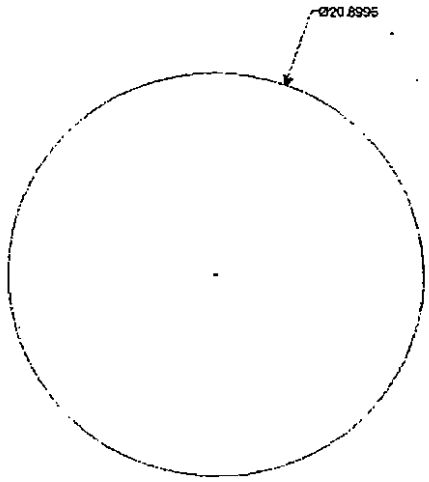
4. Diameter Dimension command:

The DIAMETER DIMENSION command is used to specify the diameter of the circles or arcs that are present in the drawings.

Command: DIMDIAMETER

Select the circle to annotate:

Close the command.



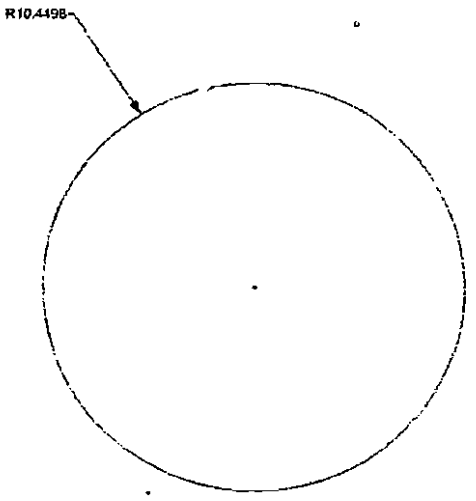
5. Radius Dimension command:

The DIAMETER DIMENSION command is used to specify the diameter of the circles or arcs that are present in the drawings.

Command: DIMDIAMETER

Select the circle to annotate:

Close the command.



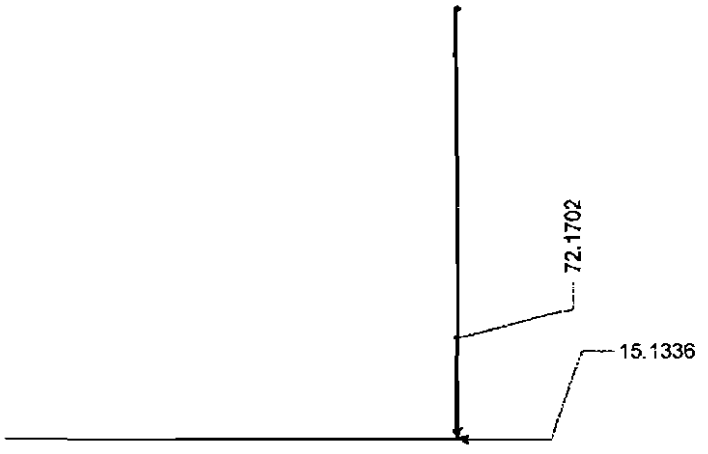
6. Ordinate Dimension command:

The ORDINATE DIMENSION command is used to annotate co-ordinate points with X or Y values of any point in the drawing. This may be useful for setting-out on site plans.

Command: DIMORDINATE

Select the point to dimension ordinates:

Close the command.



7. Angular Dimension command:

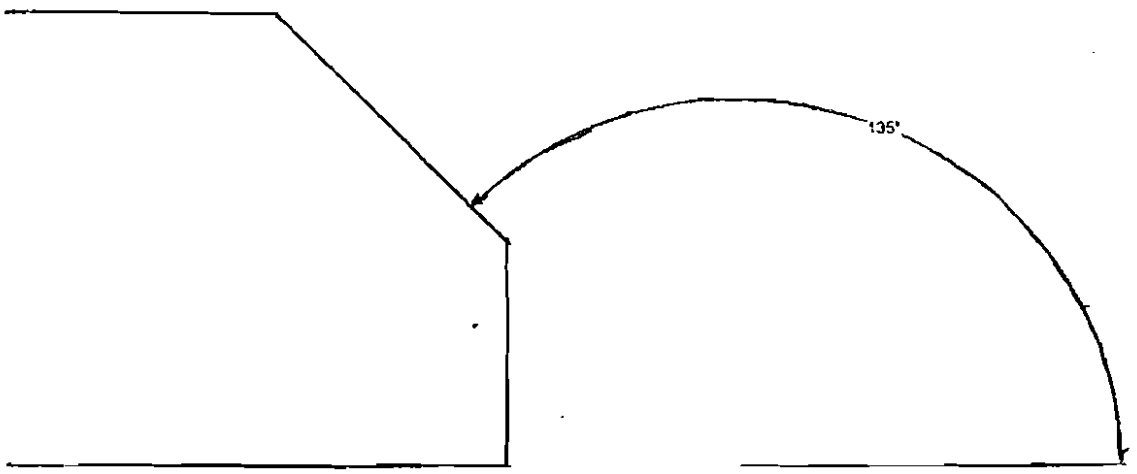
The ANGULAR DIMENSION command is used to represent the angles of angular lines with respect to a base line in the drawings.

Command: DIMANGULAR

Select base line:

Select the line to represent angle:

Close the command.



8. Dimension Text Edit command:

DIMENSION TEXT EDIT command is used to edit the text in the dimension. This command option is used to edit the dimension to represent the repetitive dimensions.

Command: DIMTEDIT

Select the dimension to edit:

Edit dimension:

Close the command.

9. Dimension Edit command:

DIMENSOIN EDIT command is used to edit the properties of the dimensions such as dimension line type, line thickness, arrow size, text height etc.

Command DIMEDIT

Select the dimensions to edit properties:

Edit required properties:

Close the command.

Result: The commands used to dimension different types of objects using AutoCAD software were learned and practiced.

4. 2D WIREFRAME MODELLING

Date: 7/7/16

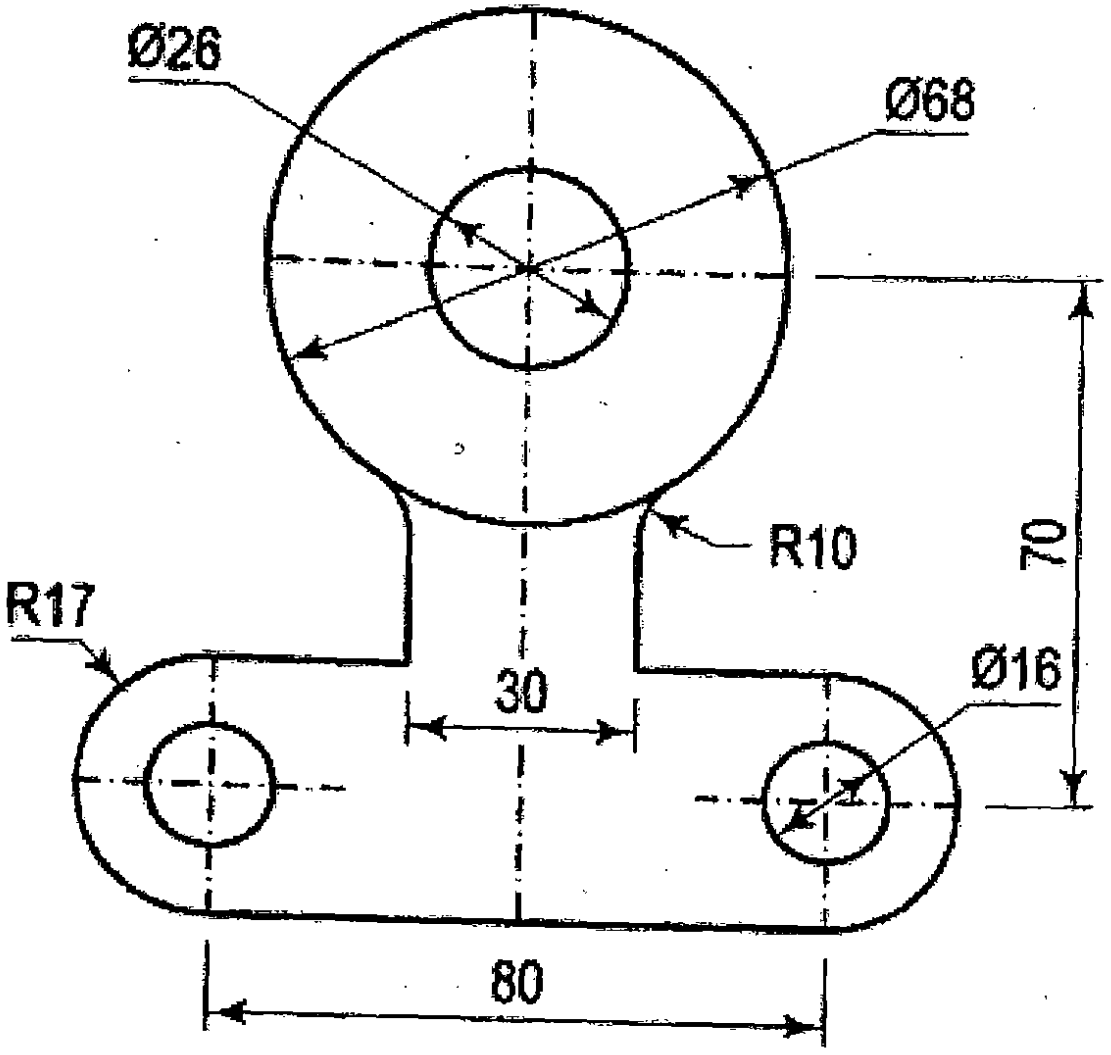
Aim: To draw the 2D wireframe model of the given diagram using AutoCAD software.

Commands used: Line, Circle, Arc, Trim, Chamfer, Fillet, dimension.

Procedure:

- Start with the axis line first. Draw the axis lines.
- Draw the line starting from the axis line horizontally and then to vertically and so on till completing the polygon.
- Using circle command, draw the circles with given dimensions.
- Now, using trim command, remove the unnecessary portions of the drawing.
- Using fillet command, round off the edges.
- Now change the axis lines to dashed lines by changing line properties.
- Dimension the drawing using annotation commands.
- Adjust the dimension text height and arrow size by changing dimension properties.

Result: The 2D isometric drawing of a given diagram is drafted using AutoCAD software.


Ø26
Ø68
R10
70
R17
Ø16
30
80

5. 3D WIREFRAME MODELLING

Date: 21/7/16

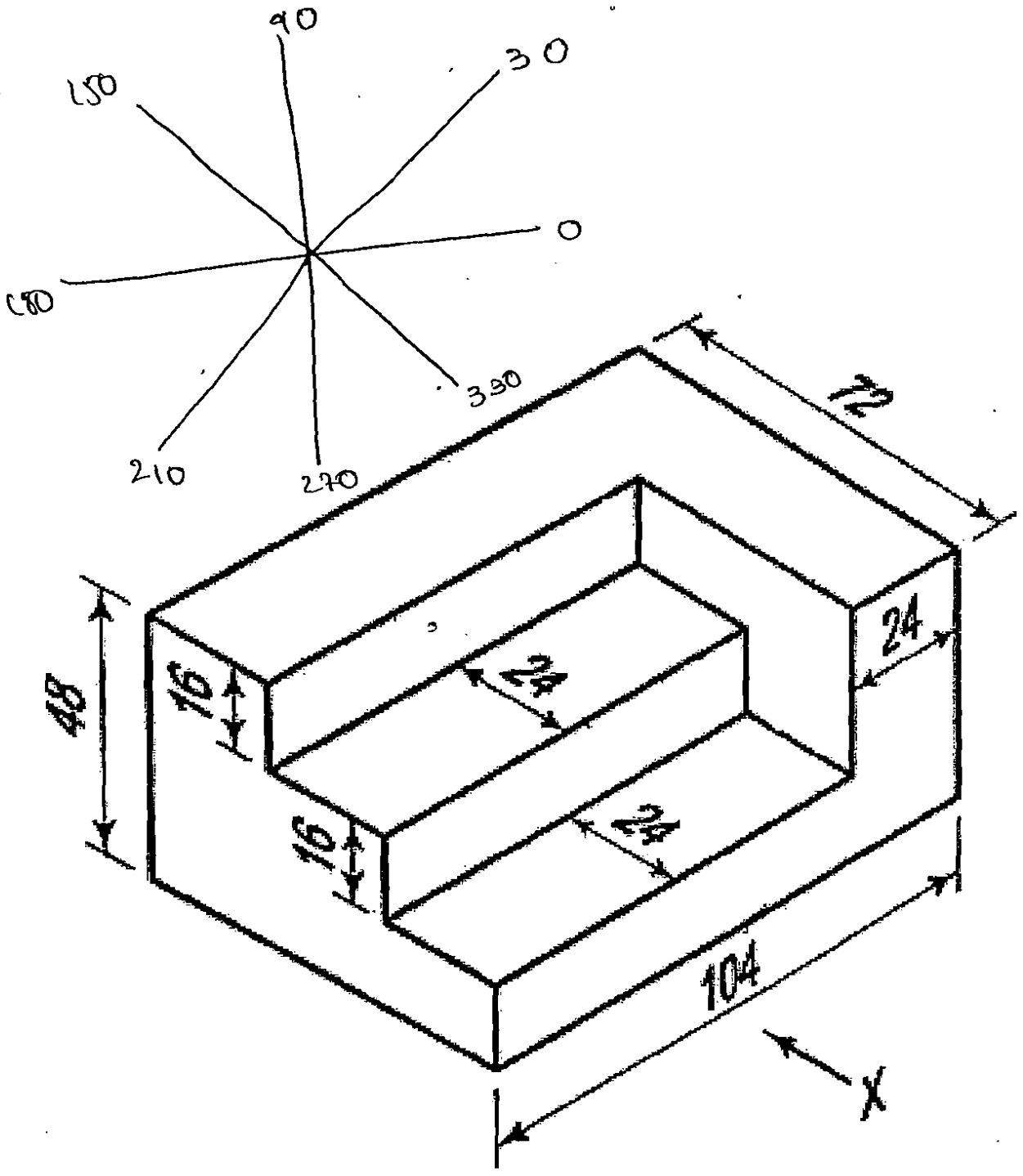
Aim: To draw the 3D wireframe model of the given diagram using AutoCAD software.

Commands used: Line, Circle, Arc, Trim, Chamfer, Fillet, dimension.

Procedure:

- The diagram is started with isoplane cross hair which is changed in snap settings, and then lines are started for isometric drawing.
- Lines are started with line command in required isoplane, and then it could be changed to another by pressing F5.
- Wedge lines are drawn at required distances by changing the isoplane.
- At the corners of base chamfer is applied to remove sharp edges.
- Using ellipse option isocircles are drawn to required dimensions.
- Two isocircles are drawn with an offset distance of 6mm and tangents are drawn to join both the isocircles of dia 18mm.
- Using trim option the unnecessary lines are trimmed at the chamfering portion.
- Dimensioning for the length and height of the diagram is kept using "aligned dimensioning" Command from dimensioning tool bar.

Result: The 3D isometric drawing of a given diagram is drafted using AutoCAD software.


90
150
30
180
0
210
270
330
72
24
48
16
24
16
24
104
X

6. ISOMETRIC DRAWING USING AUTOCAD

Date: 28/7/16

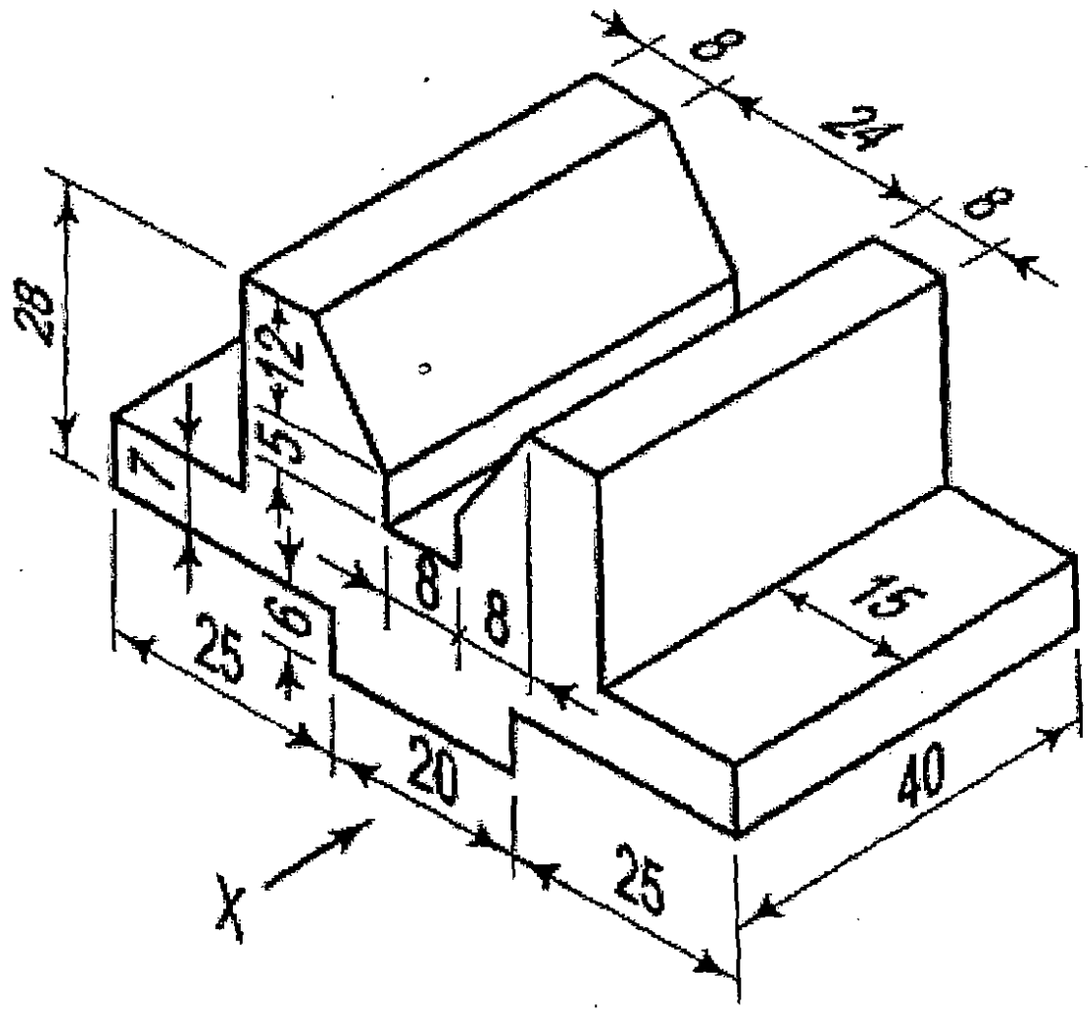
Aim: To draw the isometric model of the given diagram using AutoCAD software.

Commands used: Line, Circle, Arc, Trim, Chamfer, Fillet, dimension.

Procedure:

- The diagram is started with isoplane cross hair which is changed in snap settings, and then lines are started for isometric drawing.
- Lines are started with line command in required isoplane, and then it could be changed to another by pressing F5.
- Wedge lines are drawn at required distances by changing the isoplane.
- At the corners of base chamfer is applied to remove sharp edges.
- Using ellipse option isocircles are drawn to required dimensions.
- Two isocircles are drawn with an offset distance of 6mm and tangents are drawn to join both the isocircles of dia 18mm.
- Using trim option the unnecessary lines are trimmed at the chamfering portion.
- Dimensioning for the length and height of the diagram is kept using "aligned dimensioning" Command from dimensioning tool bar.

Result: The isometric drawing of a given diagram is drafted using AutoCAD software.


8
24
8
28
12
15
7
8
8
15
6
25
20
40
25
X

7. ISOMETRIC DRAWING USING AUTOCAD

Date: 8/9/16

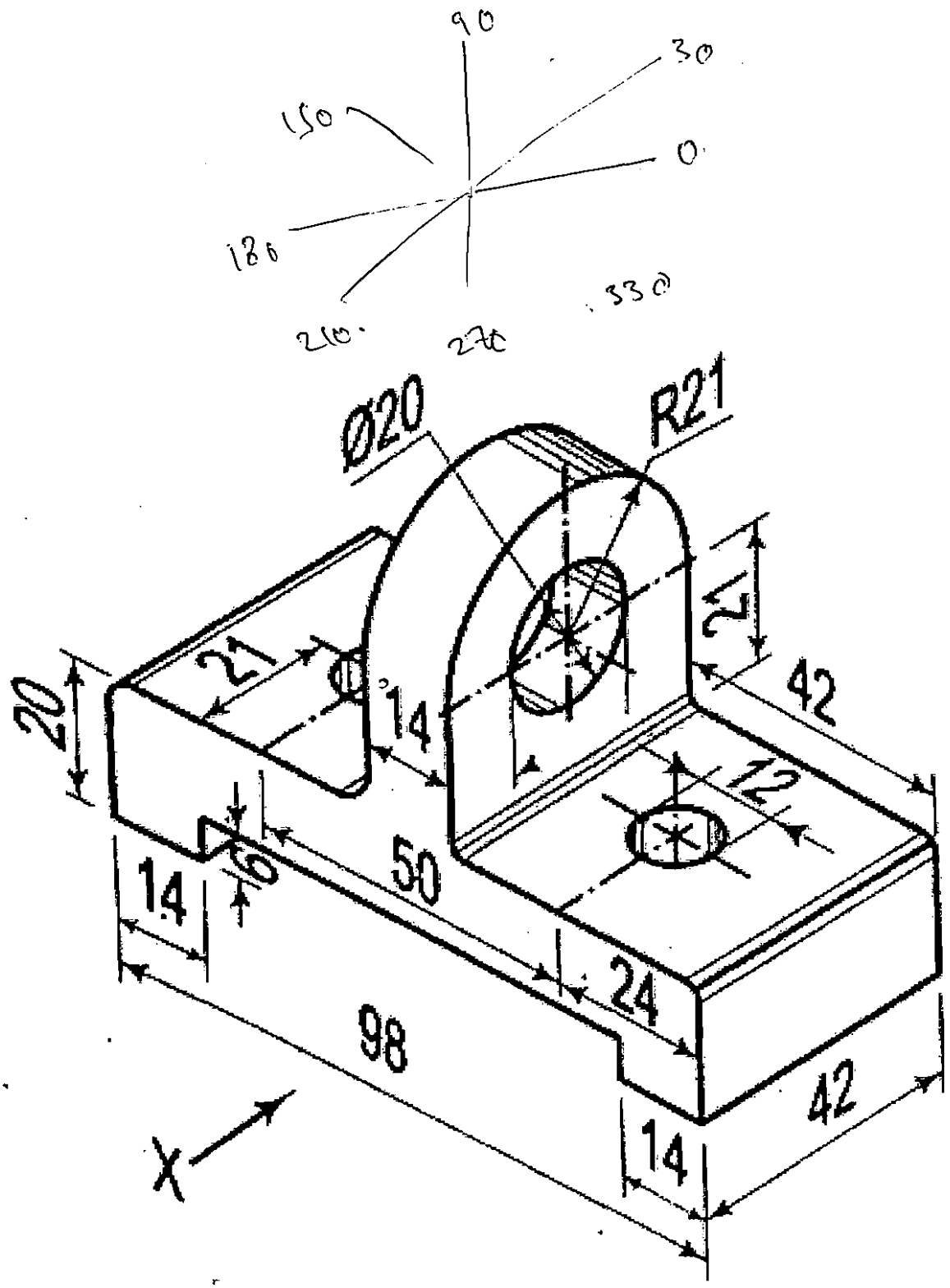
Aim: To draw the isometric drawing of the given diagram using AutoCAD software.

Commands used: Line, Circle, Arc, Trim, Chamfer, Fillet, dimension.

Procedure:

- The diagram is started with isoplane cross hair which is changed in snap settings, and then lines are started for isometric drawing.
- Lines are started with line command in required isoplane, and then it could be changed to another by pressing F5.
- Wedge lines are drawn at required distances by changing the isoplane.
- At the corners of base chamfer is applied to remove sharp edges.
- Using ellipse option isocircles are drawn to required dimensions.
- Two isocircles are drawn with an offset distance of 6mm and tangents are drawn to join both the isocircles of dia 18mm.
- Using trim option the unnecessary lines are trimmed at the chamfering portion.
- Dimensioning for the length and height of the diagram is kept using "aligned dimensioning" Command from dimensioning tool bar.

Result: The isometric drawing of a given diagram is drafted using AutoCAD software.


90
30
150
0
180
330
210
270
Ø20
R21
21
14
42
12
20
6
50
14
24
98
42
14
X

8. ORTHOGRAPIC PROJECTIONS OF ISOMETRIC DRAWING

Date: 15/9/16

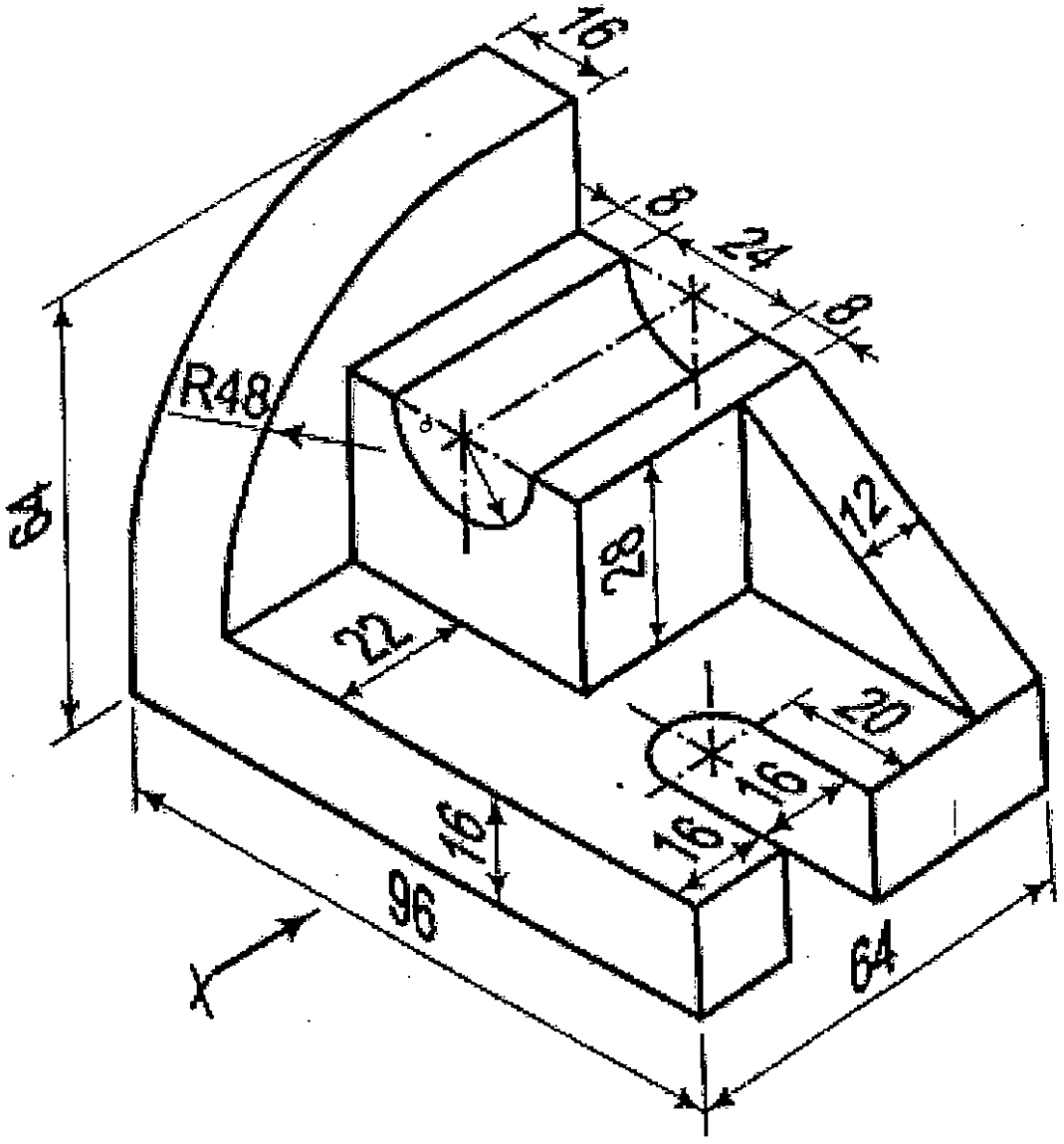
AIM: To draw the orthographic projections of a given isometric drawing with given dimensions.

COMMANDS USED: Line, Circle, Arc, Trim, Chamfer, Fillet, dimension.

PROCEDURE:

- The diagram is started with axis line; rectangles are drawn using rectangle command for the given dimensions.
- The remaining portion is drawn using line command.
- Trim command is used to remove the excess lines.
- Front view of machine block is obtained.
- Rectangle is drawn usfhg rectangle command for the given dimensions.
- Circle command is used to draw circles.
- The remaining portion is drawn using line command.
- Trim command is used to remove the excess lines.
- Top view of the machine block is obtained.
- The diagram is started with axis line.
- Rectangles are drawn using rectangle command for the given dimensions.
- Circle command is used to draw circle and half of the circle is trimmed using trim command.
- The remaining portion is drawn using line command.
- The side view of the machine block is obtained.

Result: The orthographic projection of the given drawing is drafted using AutoCAD software.


16
8
24
8
R48
38
28
12
22
20
16
16
16
96
64
X

9. ORTHOGRAPIC PROJECTIONS OF ISOMETRIC DRAWING

Date: 22/9/16

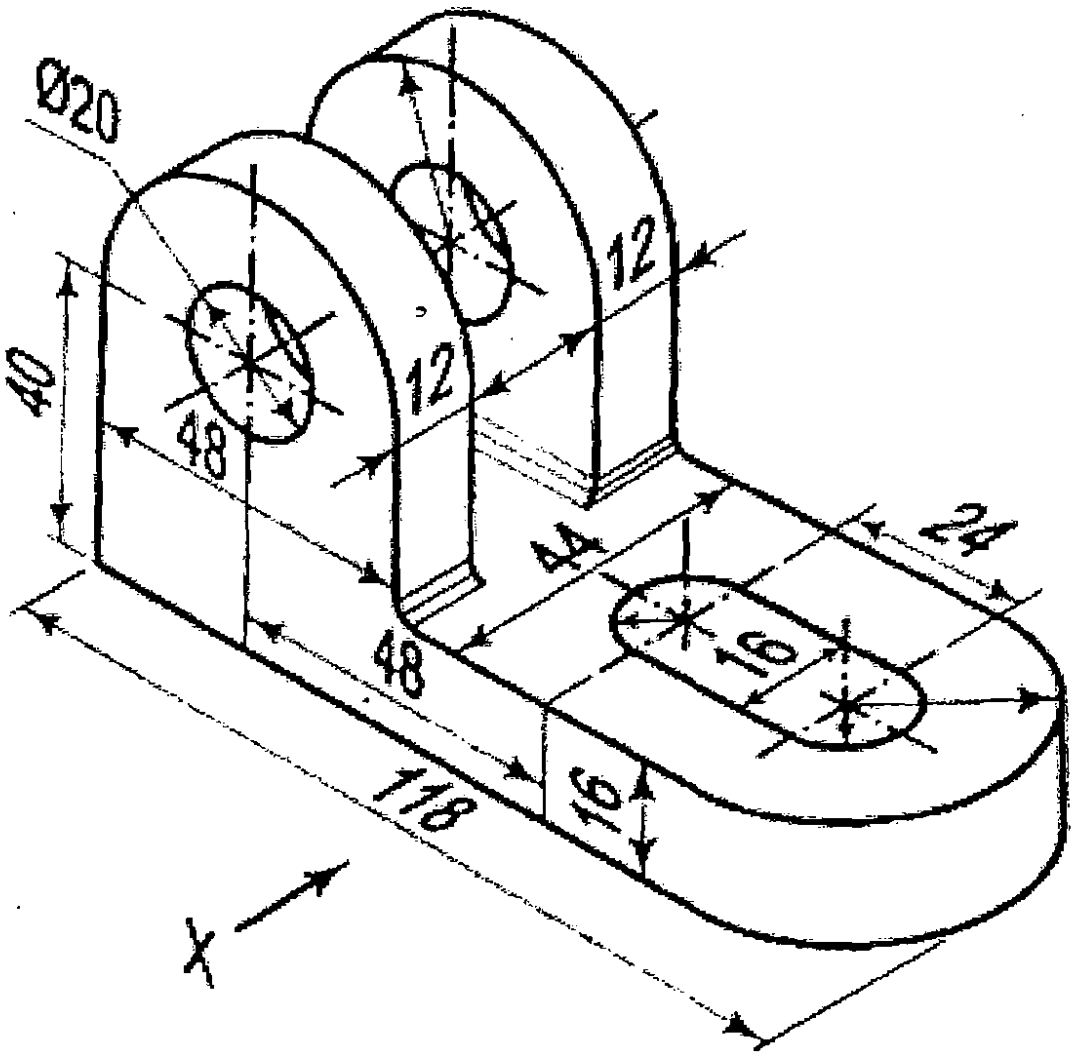
AIM: To draw the orthographic projections of the given isometric drawing.

COMMANDS USED: Line, ellipse, chamfer, aligned dimension, snap settings.

PROCEDURE:

- The diagram is started with axis line; rectangles are drawn using rectangle command for the given dimensions.
- The remaining portion is drawn using line command.
- Trim command is used to remove the excess lines.
- Front view of machine block is obtained.
- Rectangle is drawn using rectangle command for the given dimensions.
- Circle command is used to draw circles.
- The remaining portion is drawn using line command.
- Trim command is used to remove the excess lines.
- Top view of the machine block is obtained.
- The diagram is started with axis line.
- Rectangles are drawn using rectangle command for the given dimensions.
- Circle command is used to draw circle and half of the circle is trimmed using trim command.
- The remaining portion is drawn using line command.
- The side view of the machine block is obtained.

Result: The orthographic projection of the given drawing is drafted using AutoCAD software.


Ø20
40
48
12
12
44
24
48
16
118
16
X

10.3D MODELLING OF SIMPLE OBJECTS

Date: 29/9/16

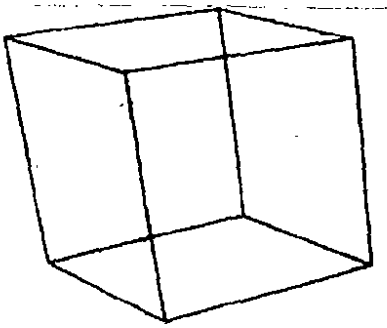
AIM: To model simple 3D objects in AutoCAD.

COMMANDS USED: Line, ellipse, chamfer, aligned dimension, snap settings.

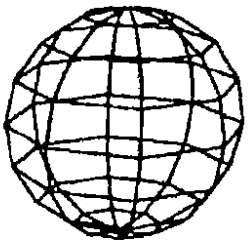
PROCEDURE:

- By using workspaces tool bar 2D settings are converted in to 3D modelling.
- From the view tool bar front view is selected and is drawn by using draw commands
- View is changed from front view to southwest isometric.
- The rectangle is extruded by using extrude command with a length of 50mm from the modelling tool bar.
- By using UCS command and selecting sub option "face" the face is selected on which the hole and slot are to be inserted.
- A cylinder and cuboid are drawn in that view and moved to the required position.
- Both of them are subtracted from the main block using subtract command from modelling tool bar.
- By using same UCS procedure top view is selected and two square block are inserted and subtracted.
- Dimensioning for the length and height of the diagram is kept using "linear dimensioning" command from dimensioning tool bar.

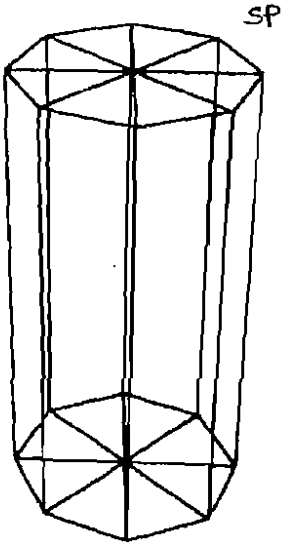
Result: Simple 3D solids are modelled using AutoCAD software.



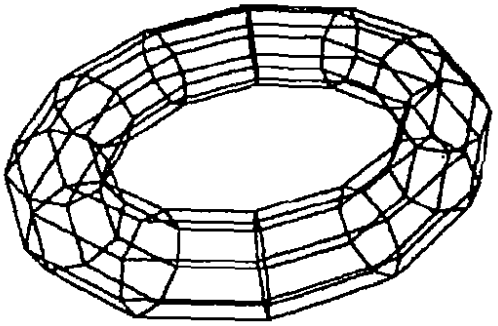
CUBE



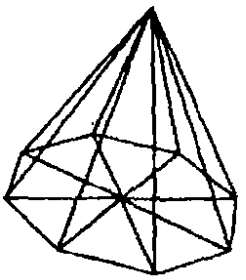
SPHERE



CYLINDER



TORUS



CONE

7, 10, 11, 12

11.3D MODELLING OF A GEAR

Date: 2/9/16.

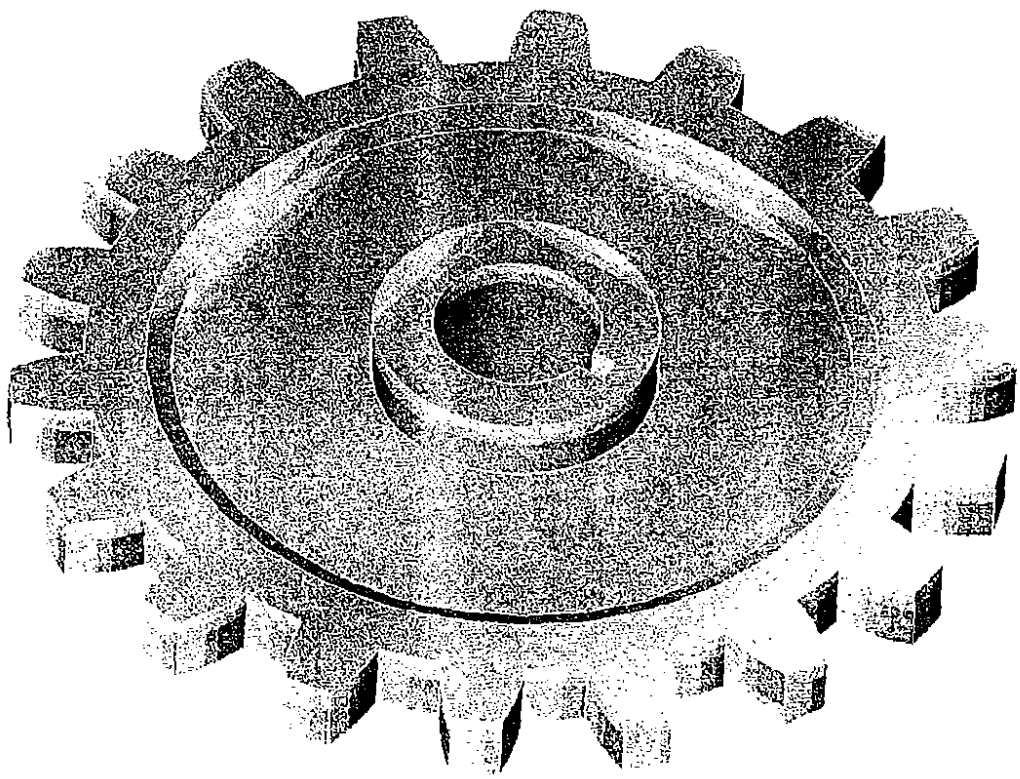
AIM: To model a gear in 3D using AUTO CAD.

COMMANDS USED: Line, ellipse, chamfer, aligned dimension, snap settings.

PROCEDURE:

- By using workspaces tool bar 2D settings are converted in to 3D modelling.
- From the view tool bar front view is selected and is drawn by using draw commands.
- View is changed from front view to southwest isometric.
- The rectangle is extruded by using extrude command with a length of 50mm from the modelling tool bar.
- By using UCS command and selecting sub option "face" the face is selected on which the hole and slot are to be inserted.
- A cylinder and cuboid are drawn in that view and moved to the required position.
- Both of them are subtracted from the main block using subtract command from modelling tool bar.
- By using same UCS procedure top view is selected and two square block are inserted and subtracted.
- Dimensioning for the length and height of the diagram is kept using "linear dimensioning" command from dimensioning tool bar.

Result: A 3D model of a gear is generated using AutoCAD software.



SHAFT WITH MOUNTINGS

12. MODELLING OF ~~BOLT AND NUT~~

Date: 29/9/16

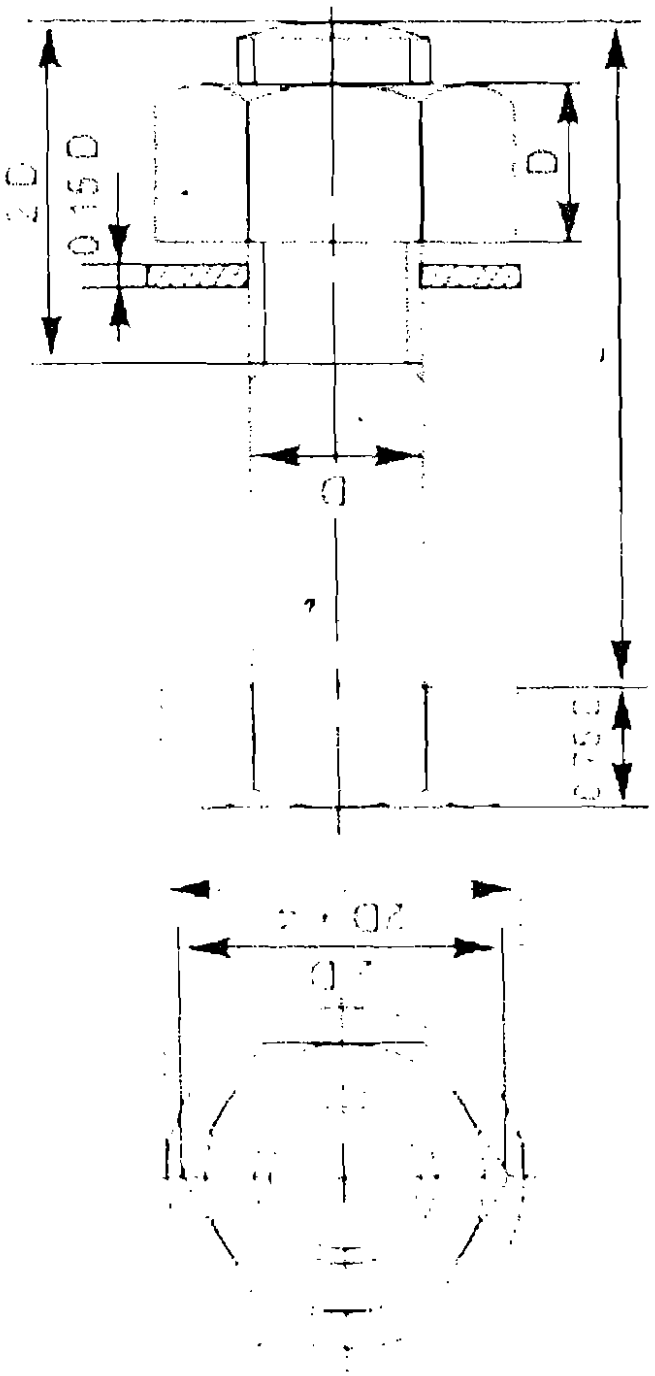
AIM: To draw the BOLT & NUT diagram with given dimensions.

COMMANDS USED: Line, Rectangle, Arc, Circle, Polygon.

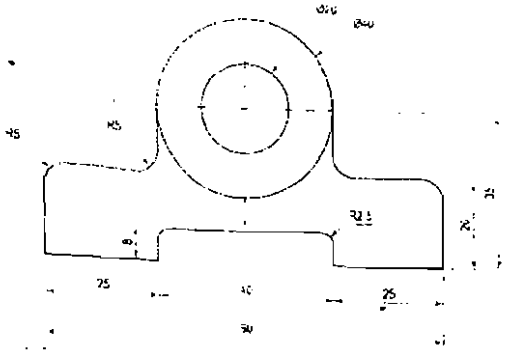
PROCEDURE:

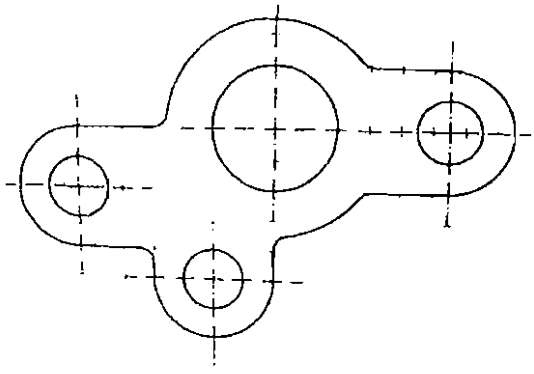
- The diagram is started with axis line, and then the bolt head is drawn using rectangle command for the dimensioning of 20x48. The shank portion is drawn using line command.
- Arcs are drawn on the bolt head to indicate the chamfering. The chamfering thickness is taken as 2mm.
- Nut and washer are drawn using rectangle command. Arcs are drawn on the nut to indicate the chamfering.
- The threading sign on the bolt is indicated by using line command, and end of the threading is ends with an arc.
- Using trim option the unnecessary lines are trimmed at the chamfering portion and threading portion.
- The side view of the bolt head is drawn using polygon command with six sides, circumscribing circle of diameter 42. And washer side view is drawn using circle command with dia 52.
- The side view for the shank portion is drawn using circle command with dia 24. The properties of the circle of dia 24 are changed to dashed lines.
- The top view of the bolt and nut is started from the bolt head portion with a rectangle command.
- The chamfering, shank, threading, washer, and nut for top view are drawn using relevant commands as practiced above.
- Dimensioning for the length and height of the diagram is kept using "linear dimensioning" command from dimensioning tool bar. The side view of the washer and shank are dimensioned using "diameter" command from the dimensioning tool bar.

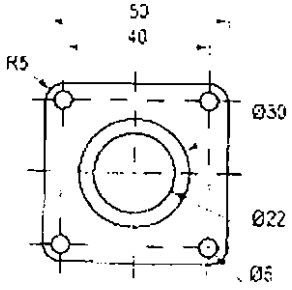
RESULT: The bolt and nut diagram was drawn using AutoCAD software.

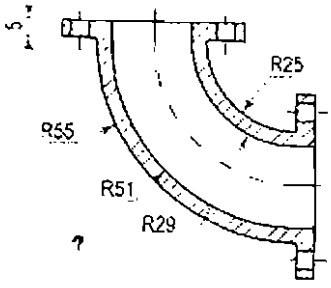


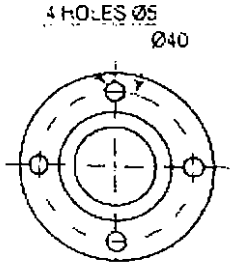
EXERCISES

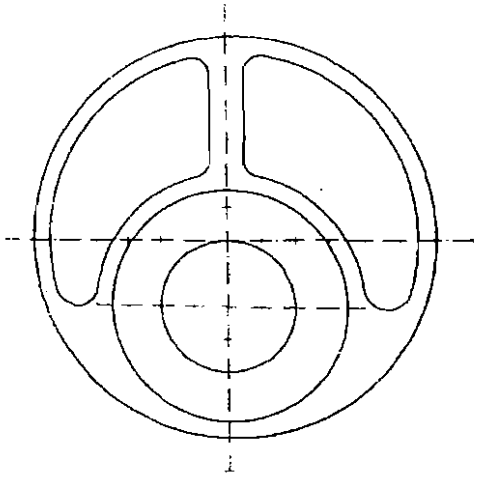


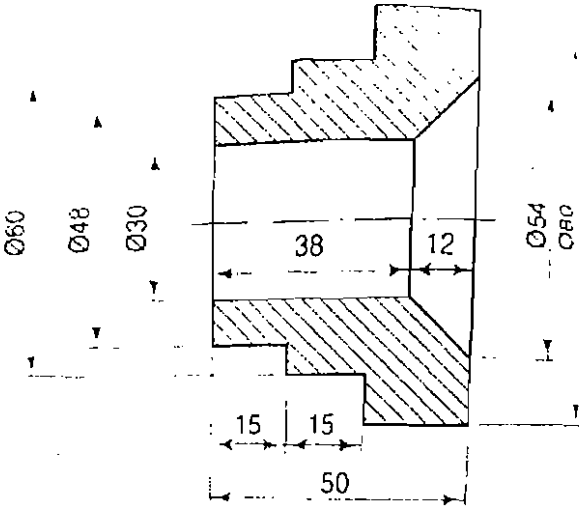


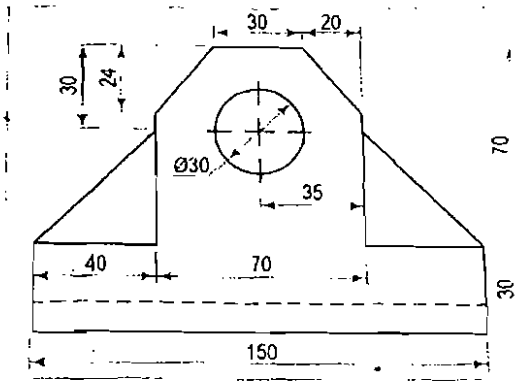

50
40
R5
Ø30
Ø22
Ø6

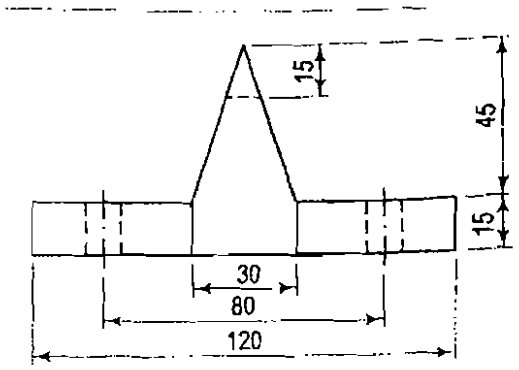

R25
R55
R51
R29

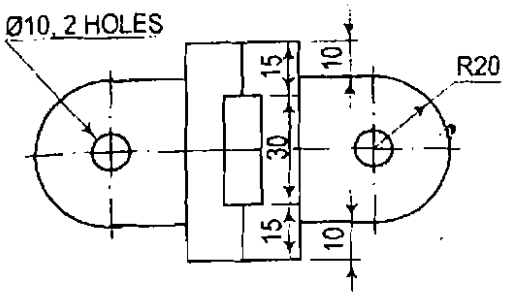

4 HOLES Ø5
Ø40

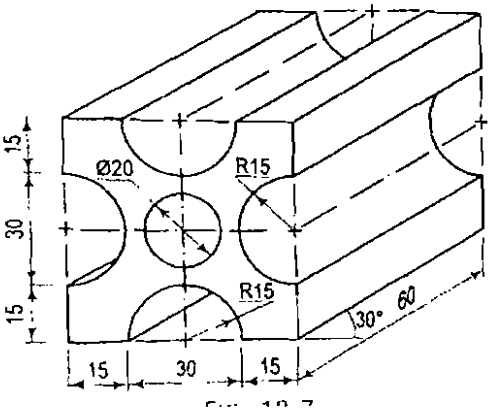


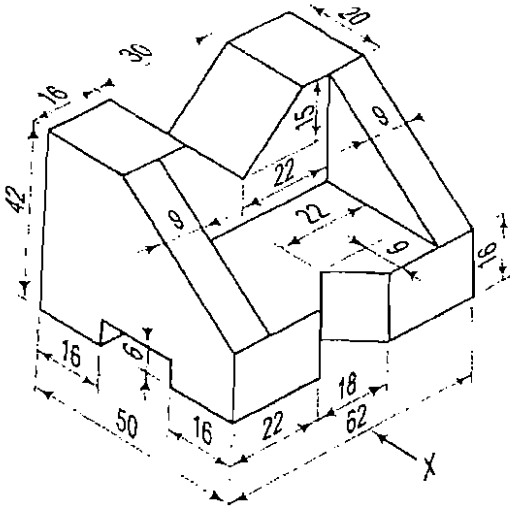

Ø60
Ø48
Ø30
38
12
Ø54
Ø80
15
15
50

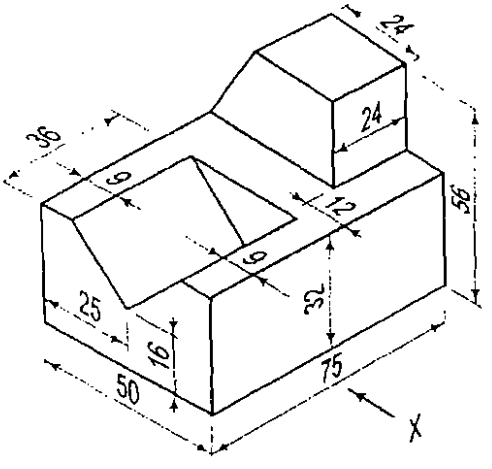

30
20
30
24
Ø30
35
70
40
70
30
150

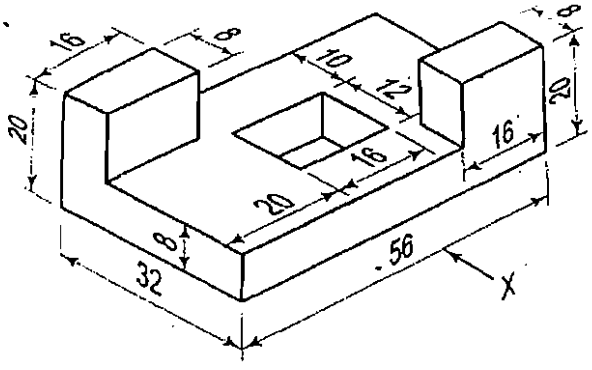

15
45
15
30
80
120

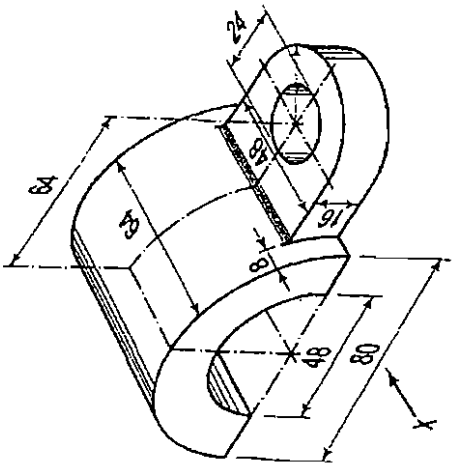

Ø10, 2 HOLES
R20
15
10
30
15
10

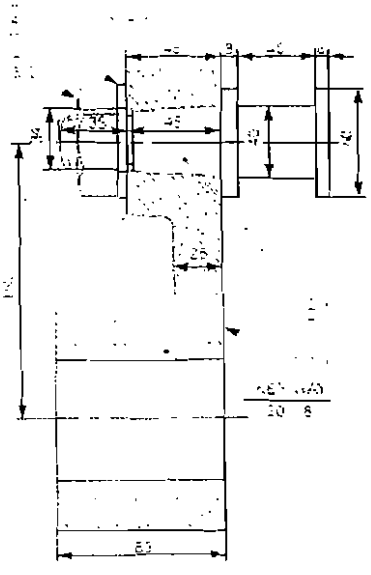

15
30
15
Ø20
R15
R15
30°
60
15
30
15

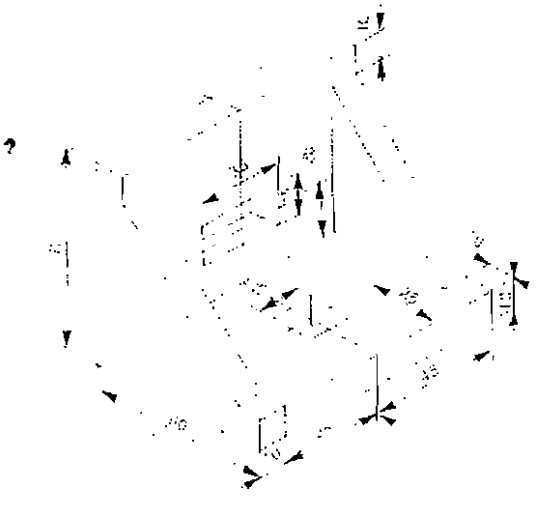

20
30
16
42
15
9
22
22
9
9
16
16
6
50
16
22
18
62
X

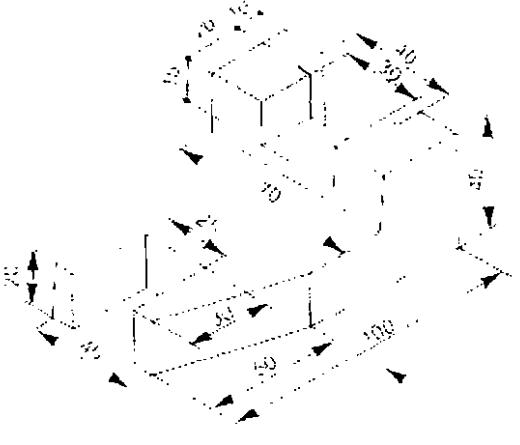

24
24
36
9
12
56
9
52
25
16
50
75
X

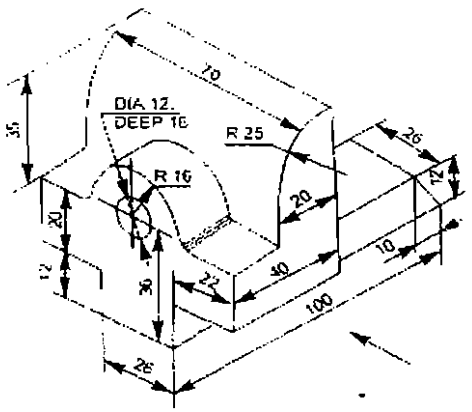

16
8
10
12
20
20
16
16
20
20
8
32
56
X

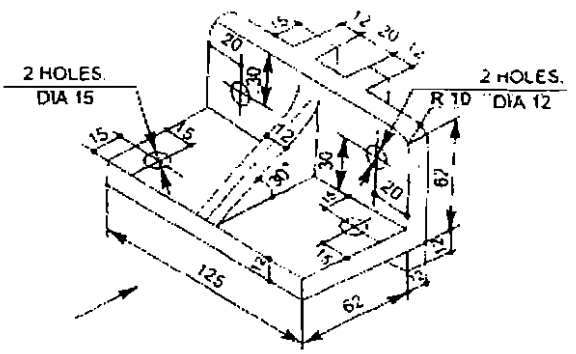

24
84
56
8
91
48
80
X

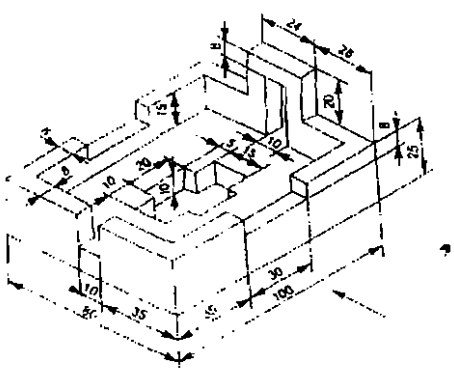


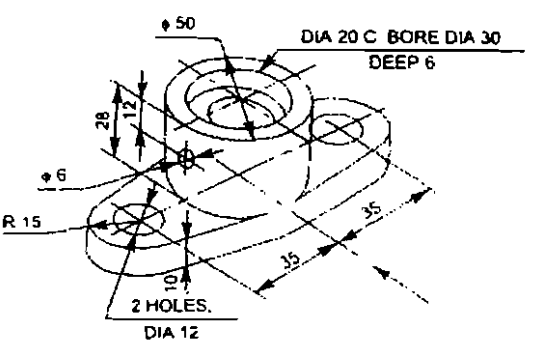


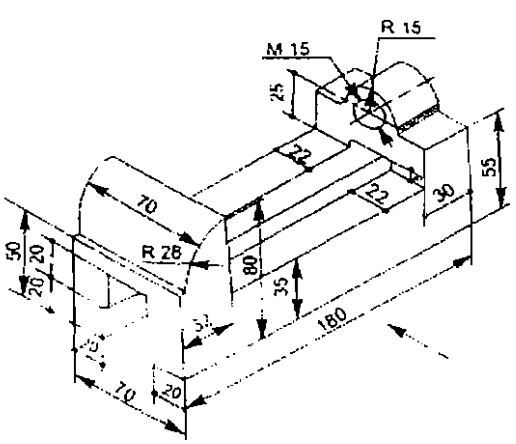


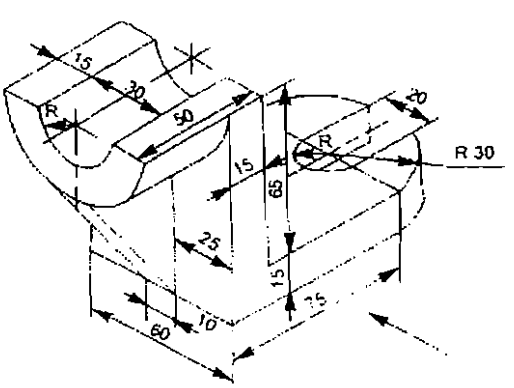

DIA 12.
DEEP 16
R 25
R 16
70
35
20
12
30
40
100
26
10

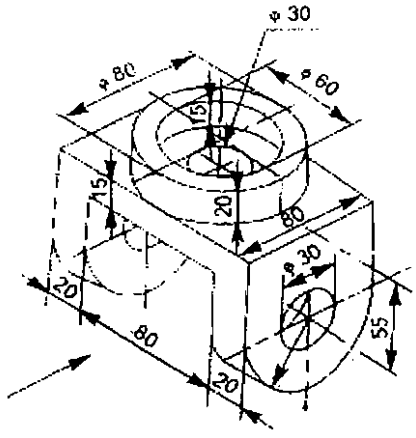

2 HOLES.
DIA 15
2 HOLES.
R 10
DIA 12
125
62

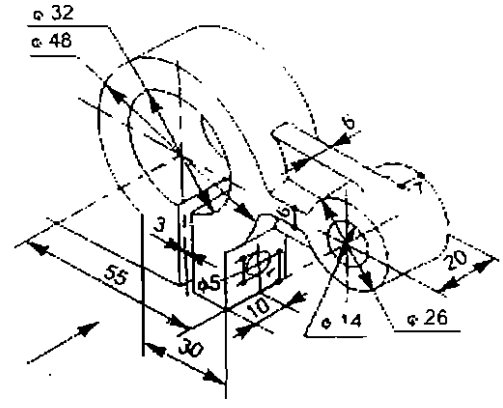

24
28
25
30
35
109

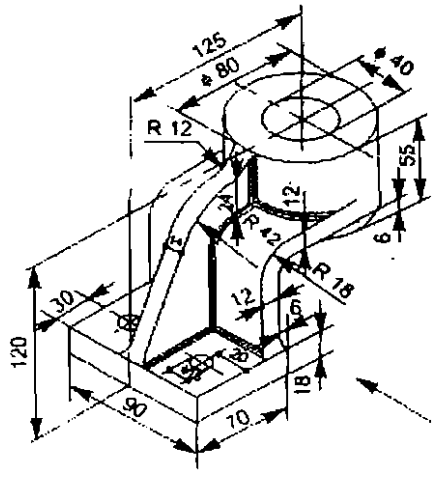

φ50
DIA 20 C BORE DIA 30
DEEP 6
φ6
R 15
2 HOLES.
DIA 12

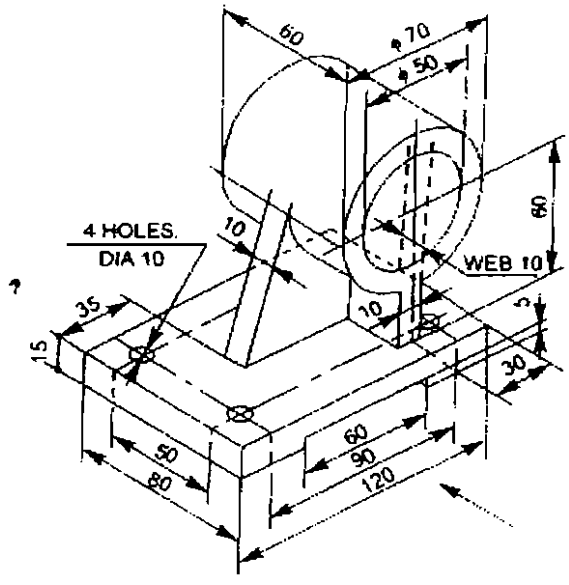

M 15
R 15
70
R 28
50
20
80
35
55
30
180

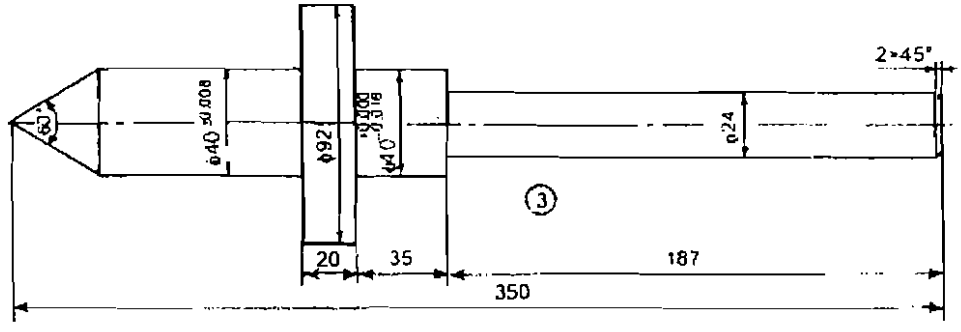

R 30
65
50
30
60


φ 30
φ 80
φ 60
15
15
20
80
20
80
20
φ 30
55


φ 32
φ 48
6
3
6
55
45
10
30
20
φ 14
φ 26


125
φ 80
φ 40
R 12
55
6
14
12
R 12
R 18
12
6
30
120
90
70
18


60
φ 70
φ 50
4 HOLES.
DIA 10
10
WEB 10
60
35
10
15
3
30
50
80
60
90
120


2×45°
60°
φ40 -0.008
φ92
φ40 +0.000 +0.016
φ24
③
20
35
187
350