

# AutoCAD

## MODULE 5.1



**Centre for Electronics Design & Technology of India**

A Scientific Society under Department of Electronics,  
Govt. of India, New Delhi

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## FOREWORD

The Information Technology and Telecom sectors have suddenly opened up avenues, which require a very large specially trained manpower. These sectors are highly dynamic and need training and re-training of manpower at a rapid rate. The growing gap of requirement of the industry and its fulfilment has created a challenging situation before manpower training institutes of the country. To meet this challenge most effectively, Centre for Electronics Design and Technology of India (CEDTI) has launched its nation-wide franchising scheme.

Centre for Electronics Design and Technology of India (CEDTI) is an Autonomous Scientific Society under the Govt. of India, Department of Electronics with its Headquarters at New Delhi. It operates seven centres located at Aurangabad, Calicut, Gorakhpur, Imphal, Mohali, Jammu and Tezpur. The scheme will be implemented and coordinated by these centres.

The scheme endeavours to promote high quality computer and information technology education in the country at an affordable cost while ensuring uniform standards in order to build a national resource of trained manpower. Low course fees will make this education available to people in relatively small, semi urban and rural areas. State-of-the-art training will be provided keeping in view the existing and emerging needs of the industrial and Govt. sectors. The examinations will be conducted by CEDTI and certificates will also be awarded by CEDTI. The scheme will be operated through all the seven centres of CEDTI.

The CEDTI functions under the overall control and guidance of the Governing Council with Secretary, Department of Electronics as its Chairman. The members of the council are drawn from scientific, government and industrial sectors. The Centres have separate executive committees headed by Director General, CEDTI. The members of these committees are from academic/professional institutes, state governments, industry and department of electronics.

CEDTI is a quality conscious organisation and has taken steps to formally get recognition of the quality and standards in various activities. CEDTI, Mohali was granted the prestigious ISO 9002 certificate in 1997. The other centres have taken steps to obtain the certification as early as possible. This quality consciousness will assist CEDTI in globalizing some of its activities. In keeping with its philosophy of 'Quality in every Activity', CEDTI will endeavour to impart state of the art – computer and IT training through its franchising scheme.

The thrust of the Software Courses is to train the students at various levels to carry out the Management Information System functions of a medium sized establishment, manufacture Software for domestic and export use, make multimedia presentations for management and effectively produce various manufacturing and architectural designs.

The thrust of the Hardware Courses at Technician and Telecommunication Equipment Maintenance Course levels is to train the students to diagnose the faults and carry out repairs at card level in computers, instruments, EPABX, Fax etc. and other office equipment. At Engineer and Network Engineer levels the thrust is to train them as System Engineers to install and supervise the Window NT, Netware and Unix Networking Systems and repair Microcontrollers / Microprocessor based electronic applications.

An Advisory Committee comprising eminent and expert personalities from the Information Technology field have been constituted to advise CEDTI on introduction of new courses and revising the syllabus of existing courses to meet the changing IT needs of the trade, industry and service sectors. The ultimate objective is to provide industry-specific quality education in modular form to supplement the formal education.

The study material has been prepared by the CEDTI, document centre. It is based on the vast and rich instructional experience of all the CEDTI centres. Any suggestions on the improvement of the study material will be most welcome.

**(R. S. Khandpur)**  
Director General (CEDTI)

## **HISTORY & OVERVIEW OF AutoCAD**

CAD been known around 1964, however it wasn't widely used until around 1982, when CAD system such as AutoCAD became available on microcomputer.

### **FEATURES OF CAD**

#### **(1) *Ease of Use***

1. It is a clean & comfortable approach to drawing.
2. CAD is a valuable tool because it makes the tedious and inevitable task of revising drawings more pleasant.

#### **(2) *CAD as an Editor***

1. When you edit a drawing, you can “ cut and paste “ objects to other parts of the drawing or to an entirely different drawing.
2. When you use CAD for a drawing containing the same object in several places, you can draw the object once, then insert it wherever it is needed.

#### **(3) *Creating Visual Models***

1. CAD can be used to create a visual model of an imagined object.
2. You can quickly see whether an idea will work by using CAD to construct a graphic model, then checking in detail the design's critical elements. Exp.- gears & cams.

#### **(4) *Accuracy***

CAD allows greater accuracy than drawing by hand. Rather than Depending on a graphic scale and the clarity of your sight to determine dimensions, you can use CAD's built-in measuring capabilities to check the dimensions of the drawings you create.

#### **(5) *Intelligence***

Another CAD feature is the ability to attach information to an object or An entire drawing. Such a piece of information is called an attribute, and the ability to attach attributes to a drawing is often referred to as intelligence. An attribute can be visible or invisible. It can be a part number attached to a gear, the no. of phone lines attached to a desk, etc. These attributes can be extracted and manipulated in a data-base-management or spreadsheet program.

#### **(6) *Customization***

A customization feature enables you to automate repetitive tasks. Exp. While creating architectural design, you may find that you have to draw the symbols for doors, windows, sinks, & other features repeatedly. CAD enables to stores such frequently used symbols & later insert them into any drawing.

## GETTING STARTED

As we know AutoCAD13 is a window based package, so to start AutoCAD you have to open Windows95 or higher version.

In Windows95, you have to click on the **START** button at the status bar and move on the Program and then AutoCAD13. The options available in AutoCAD13 are

- **AutoCAD13**
- AutoCAD Readme
- Online Help
- Quick Tour
- What's New

Click on **AutoCAD13** and soon you will get a screen as shown in fig.

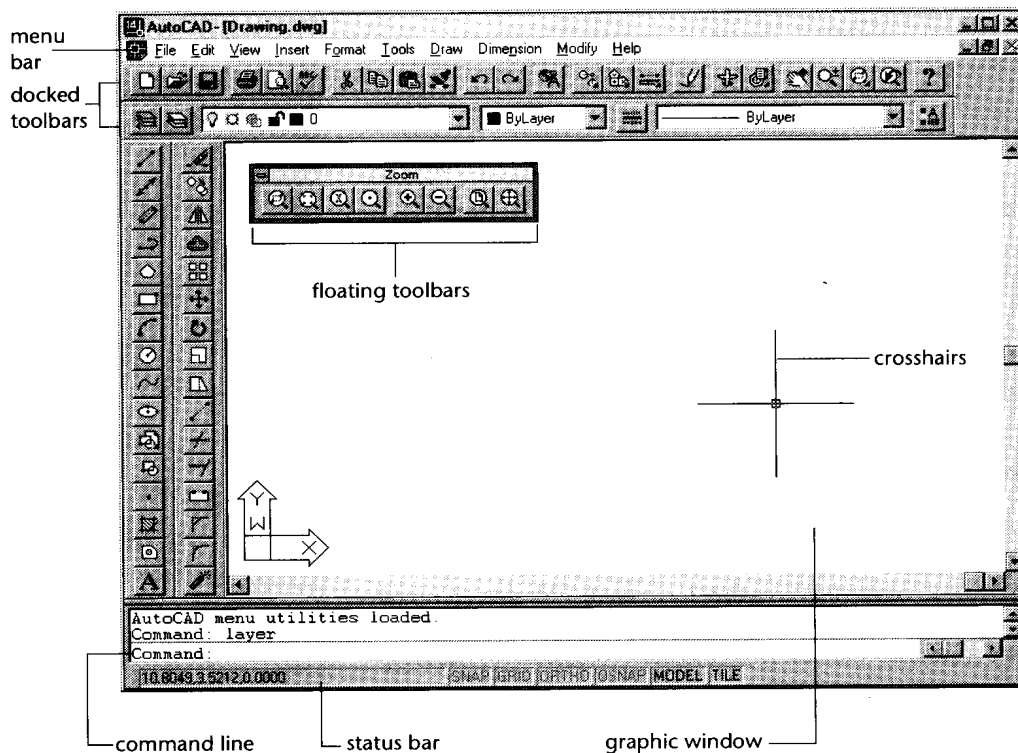


Fig. Courtesy From Autodesk Release-14 Manual

The drawing area- your workspace- occupies most of the screen. Everything you draw appears in this area. You will see crosshairs appear to move within the drawing area. This is the drawing cursor, which lets you point to locations in the drawing area.

At the bottom of the drawing area, the status bar gives you information at a glance about the drawing. The status bar gives you the information like coordinates, snap, ortho, model, tile & time.

In the lower-left corner of the drawing area, you see a thick, L-shaped arrow outline. This is the User Coordinate System (UCS) icon, which tells you your orientation in the drawing. This icon becomes helpful while drawing complex 2D & 3D drawings.

At the bottom of the screen, just above the status bar, is a small horizontal window, which is the *command window*.

**Command:** *tells you that AutoCAD is waiting for your instructions.*

At the top of the screen, the message written like *AutoCAD UNNAMED* on blue line is known as title bar. Below that is the main menu bar, also known as Pull-Down Menus. The pull-down menus available on the menu bar offer an easy-to-understand way to access the general controls and settings for AutoCAD.

The pull-down menu options perform for basic functions:

- Display additional menu choices
- Display a dialog box that contains settings you can change
- Issue a command that requires keyboard or drawing input
- Offer an expanded set of the same tools found in the Draw and Modify toolbars.

There is a second set of options and is called *Cascading menu*. Whenever you see a pull down item with the triangular pointer, this item opens a cascading menu offering a more detailed set of options.

### **Flyouts:**

Some tools will display a set of additional tools that are related to the too you have selected. This is shown in fig. This set of additional tools is called a toolbar **flyout**.



**The UCS flyout**

Fig. Courtesy From Autodesk Release-14 Manual

Below the main menu are toolbars. These toolbars have mobility. They can either be floating anywhere on the Windows screen or in a *docked* position. The toolbars can be moved to any location on the desktop, thus turning it into floating toolbar.

The various toolbars available in tools menu are:

1. draw
2. modify
3. dimension
4. solids
5. surfaces
6. preferences
7. render
8. external database
9. select objects
10. object snap
11. UCS
12. Object properties
13. Standard toolbar
14. Inquiry
15. Insert
16. Viewpoint
17. Zoom

There is a screen menu option from where various commands can be assessed. This screen menu will be displayed on the right side of your screen. You may or may not be seeing screen menu. If you are not, it can be displayed by going through the following steps.

1. Go to **OPTIONS** in main menu.
2. Select **PREFERENCES**. A dialog box will be seen on your screen. On **AUTOCAD GRAPHICS WINDOWS** click on the **SCREEN MENU** check button.
3. Then click **OK**.

Soon you will see screen menu on your screen.



## HOW TO GIVE COMMANDS TO AutoCAD

There are three ways to give commands

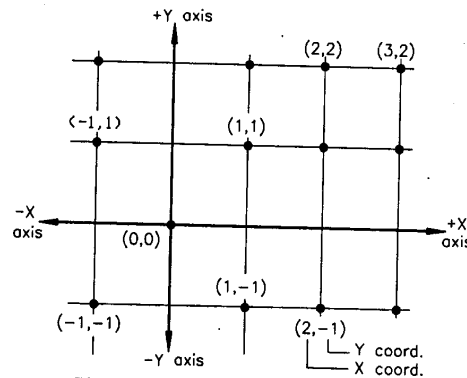
1. Through **TOOLBARS**
2. Through **SCREEN MENU**
3. Through **COMMAND: PROMPT**

## HOW TO DRAW A LINE ON SCREEN

To draw a **LINE**, you, first of all, understand the various **COORDINATE SYSTEMS** used to generate a line in AutoCAD.

### (1) **CARTESIAN COORDINATE**

To specify a point a plane, we take two perpendicular lines. The horizontal line represents the X-axis and vertical line represents Y-axis. The meeting point of these lines is called as Origin. The X coordinate is positive if measured to the right of the origin & negative if measured to the left of the origin. The y coordinate is positive if measured above the origin & negative if measured below the origin as shown in fig . This method of specifying points in plane is called Cartesian coordinates.



AutoCAD uses following coordinate systems to locate a point in X-Y plane.

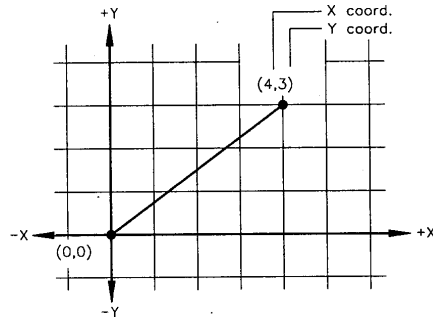
- a. Absolute coordinate system
- b. Relative coordinate system
- c. Polar coordinate system

## ABSOLUTE COORDINATE SYSTEM

In the absolute coordinate system the points are located with respect to the origin (0,0) as shown in fig.5  
Exp. = (1,1), here X=1 unit & Y=1 unit.

## RELATIVE COORDINATE SYSTEM

In relative coordinate system the displacement along X & Y axes are measured with reference to the previous point rather than the origin. The relative coordinate system is designated by the symbol @. Exp. = @4,0 here Distance in X-axis is 4 units & distance in Y-axis is 0. Now we will give another point @0,3 this displacement will be from point (4,0) and X=0 & Y=3 units.



## POLAR COORDINATE SYSTEM

In the polar coordinate system a point can be located by defining the distance of the point from the current point, and the angle that the line between the two points makes with the positive X-axis. Exp. = @5<30. In this @5 gives the 5 units distance & <30 gives the angle or direction in which 5 units will be drawn.

Now we will draw a rectangle using all the three above-mentioned coordinate systems.

Type **LINE** at the command prompt & the computer will ask you **FROM POINT**: here you have to specify the point from where you have to start your line to make rectangle. Then computer asks you **TO POINT**: means the destination point. The computer continues to ask **TO POINT**: unless press enter or clicked right button of your mouse to terminate the command.

### **Command: line**

From point: 1,1

To point: 4,1

To point: 4,3

To point: 1,3

You will see a rectangle on your screen. This is drawn with the use of Absolute coordinate system.

### **Command: line**

From point: 1,1

To point: @4,0

To point: @0,3

To point: @-4,0

To point: @0,-3

This will draw a rectangle using Relative coordinate system.

**Command: *line***

From point: 1,1

To point: @4<0

To point: @3<90

To point: @4<180

To point: @3<270

This will draw a rectangle using Polar coordinate system.

The line command has three following options:

- continue
- close
- undo

\* **Continue:** - After exiting from the **LINE** command you may want to draw another line starting from the point where the previous line ended. In this case you may use Continue option. Perform the following steps.

**Command: *line***

From point: 2,2

To point: 6,2

To point: Enter

**Command: *line***

From point: *Either Enter here to continue or type @ to continue from point 6,2*

To point: 6,5

To point: *Enter*

\* **Close:** - The close option can be used to join the current point with the initial point of the first line when two or more lines are drawn in continuation. Type **C** at the last **TO POINT:** and see the result on your screen.

\* **Undo:** - If you draw a line and want to erase, you can use **UNDO** option. Type **U** at the **TO POINT :** from where you want to go back.

## SAVING YOUR WORK

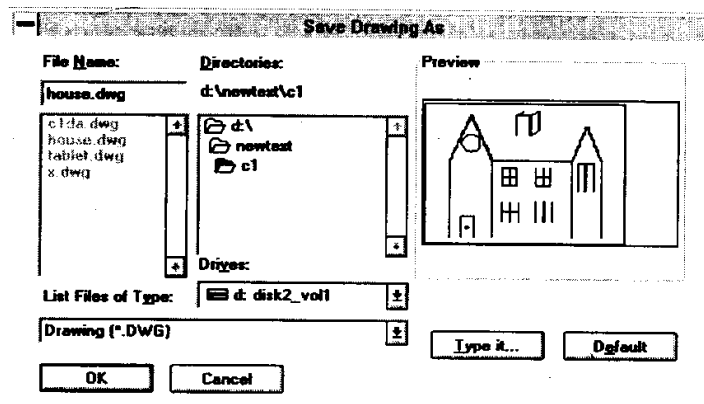
In AutoCAD you must save your work before you exit from the drawing editor or turn the system off. There are three options available in AutoCAD to save your work on Hard disk or on floppy diskette:

- **SAVE**
- **SAVEAS**
- **QSAVE**

**SAVE :-** The SAVE command allows you to save your drawing by writing it to a permanent storage device, such as hard disk, or derive A or B.

### Command: save

In the save dialog box as shown in fig., type the name in the **FILE NAME edit box** in which the drawing will be saved



**SAVEAS :-** The SAVEAS works in the same way as SAVE command.

You can choose the above options from the **MAIN MENU**.

**FILE----- SAVE or SAVEAS**

**QSAVE:-** The QSAVE command saves the current named drawing without asking you to enter a filename, thus allowing you to do a quick save.

**Command: qsave**

## OPENING AN EXISTING FILE

### FILE----OPEN

To open an existing file either type OPEN at the command prompt or use MAIN MENU. The dialog box will be seen on your screen as shown in fig. You can either enter the name at the FILE NAME edit box or choose file name by using scroll bars. The individual files will be highlighted on by one.

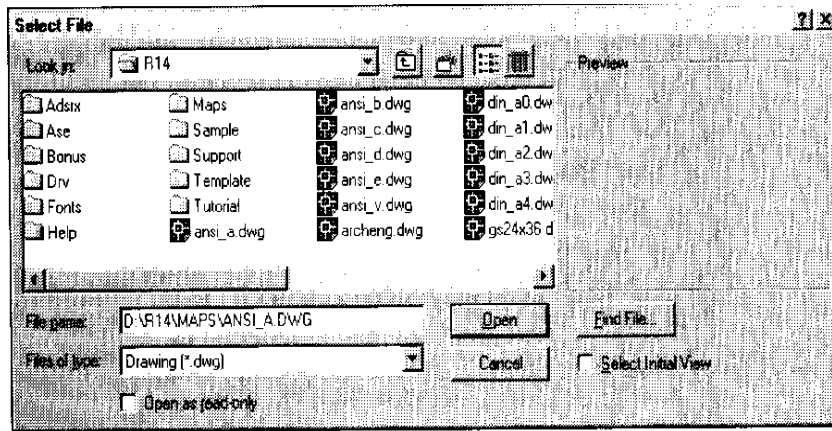


Fig. Courtesy From Autodesk Release-14 Manual

## STARTING A NEW FILE

### FILE-----NEW

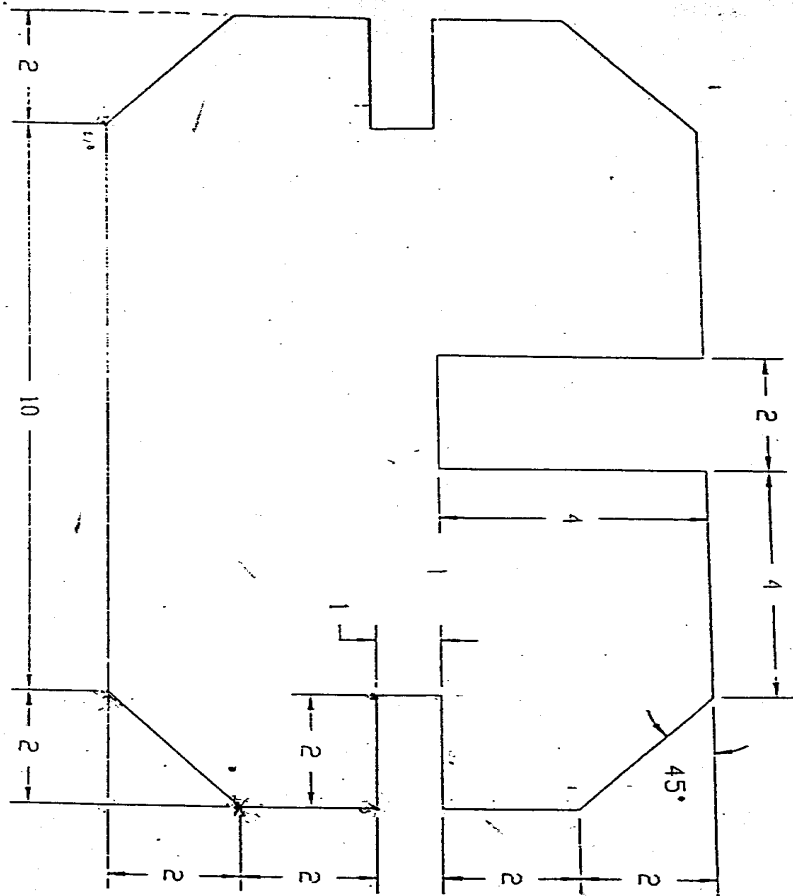
When you are running an AutoCAD, you are already in drawing editor and the default name is **UNNAMED**. The **NEW** command can be invoked from the Standard toolbar by selecting the **NEW** icon, the pull-down menu (**select FILE, NEW.....**), or the screen menu, or by entering **NEW** at the **Command: prompt**.

## QUITTING A DRAWING

### FILE-----EXIT

You can exit from the drawing editor by using the **EXIT**, **QUIT**, or **END** commands.

**Exercise No.1 -Draw the following fig. By using all the three coordinate systems.**



You have successfully created your first drawing. Now we will move further towards other features of AutoCAD drawing various other important drawing functions. First we will learn how to draw a circle.

## **DRAWING A CIRCLE**

### ***DRAW-----CIRCLE***

Command: **circle or C**

The following is the prompt sequence for the CIRCLE command

**3P/2P/TTR/<Center Point>:**

#### **1. Center & Radius Option**

In this option you draw a circle by defining the center and the radius of the circle.

**Command: C**

3P/2P/TTR/<Center Point>: 3,2

Diameter/<Radius>: 1 (or @1<0, or @1,0)

#### **2. Center & Diameter Option**

In this option you can draw a circle by defining the center & diameter of the circle.

**Command: C**

3P/2P/TTR/<Center Point>: 4,2

Diameter/<Radius>: D

Diameter<current>: 3

#### **3. Two-Point Option**

You can draw a circle using the two-point option.

**Command: C**

3P/2P/TTR/<Center Point>: 2P

First point on Diameter: 2,2

Second point on Diameter: 3,2

#### **4. Three-point option**

For drawing a circle, you can use the Three-point option by defining three points on the circumference of the circle.

**Command: C**

3P/2P/TTR/<Center Point>: 3P

First point : 4,4

Second point : 4,1

Third point: 5,3

## 5. Tangent Tangent Radius option

A tangent is an object (line, circle, or arc) that contacts the circumference of a circle at only one point. In this option you specify two objects that are to be tangent to the circle, and the radius of the circle as shown in fig.

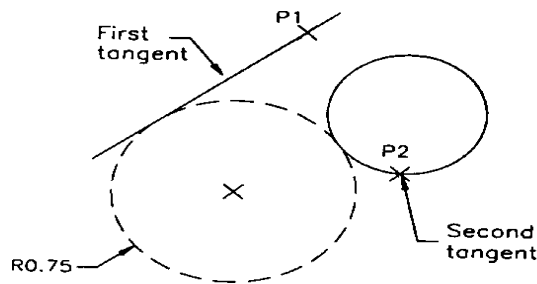
### Command: C

3P/2P/TTR/<Center Point>: TTR

Enter Tangent Spec: *select first line, circle, or arc*

Enter second Tangent Spec: *select second line, circle, or arc*

Radius<current>0.75



While editing you may find that at many times and at many places you want to erase lines or circles or you may observe + objects on your screen and you want to have a clear screen or want to see a part of your drawing in large, the following session will discuss the above mentioned topics.

## ERASING OBJECTS

### Modify ----- Erase

To erase the object use ERASE command. You can invoke the ERASE command by entering Erase at the command prompt. A small box will be seen on your screen instead of the screen cursor. This box is known as PICK BOX. To erase an object move the pick box so that it touches the object.

### Command: E

Select objects: select first object

Select objects: select second, third,--- objects

Select objects:



## **CANCELLING A COMMAND**

If you are in a command and you want to cancel or get out of that command, press the ECS (escape) key on the keyboard.

## **REDRAW COMMAND**

This command redraws the screen, thereby removing the cross marks that appears when a point is picked on the screen. These marks are known as BLIPS.

Command: REDRAW or R

## **ZOOM COMMAND**

Creating drawings on the screen would not be of much use if you could not get a magnified view of your drawing to work on minute details. You can zoom in and zoom out by using ZOOM command.

**Command: ZOOM or Z**

**All/Center/Dynamic/Extents/Left/Previous/Vmax/Window/<scale(X/XP)>:**

### **Window Option**

It lets you specify the area you want to zoom in by letting you pick two opposite corners of a rectangular window.

**Command: ZOOM or Z**

All/Center/Dynamic/Extents/Left/Previous/Vmax/Window/<scale(X/XP)>: W

First corner: *pick first point*

Second corner: *pick second point.*

### **Previous Option**

When you have completed the editing you may want to return to the previous view. This can be done by using the PREVIOUS option of the zoom command.

**Command: ZOOM or Z**

All/Center/Dynamic/Extents/Left/Previous/Vmax/Window/<scale(X/XP)>: P

### **All Option**

This option of the ZOOM command displays the drawing limits or extents, whichever is greater. Even if the objects are not within the limits, they are still included in the display. With the help of the ALL option, you can view the entire drawing in the current viewport .

### **Extents Option**

This option lets you zoom to the extents of the drawing. The extents of the drawing comprise that area that has the drawing in it. The rest of the empty area is neglected.

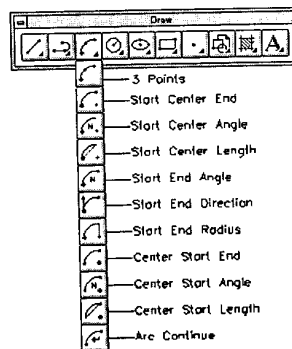
## HOW TO USE DRAW COMMANDS

So far we have use **LINE & CIRCLE** commands which comes under **DRAW** commands. In this session we will cover various other draw commands like **ARC, ELLIPSE, POLYGON, POLYLINE, POINT & TEXT.**

### DRAWING ARC

#### *Draw---ARC*

An ARC can be drawn in 11 distinct ways using the options listed under the ARC command as shown in fig.



1. **Three-point option:-** The three points option requires the Start point, second point, & end point.
2. **Start, Center, End option:** - In this option, instead of entering the second point, you enter the center point of the arc.
3. **Start, Center, and Angle option:** - This option is best suited if you know the included angle of the arc. The INCLUDED ANGLE is the angle formed by the start point and the end point of the arc with the center.
4. **Start, Center, Length of Chord option:-** In this option you are required to specify the start point, center point, and length of the chord. A CHORD is defined as the straight line connecting the start point and the end point of an arc.
5. **Start, End, Angle option:-** You can draw an arc by specifying the start point of the arc, the end point, and the included angle.
6. **Start, End, Starting Direction:-** You can draw an arc by specifying the start point, end point, and the starting direction of the arc in degrees.
7. **Start, End, Radius:-**You can draw the arc by specifying the start point, end point, & the radius of the arc.
8. **Center, Start, End option;-** This is a modification of the start, center, end option.
9. **Center, Start, Angle option:-** This is a modification of the start, center, angle option
10. **Center, Start, length of Cord option:-** This is a modification of the start, center, length option
11. **Continue option:-** With the help of this option you can continue drawing an arc from a previously drawn arc or line.

## DRAWING RECTANGLES

### *Draw---Polygon (Rectangle)*

Command: *Rectangle*

First corner: 4,3 ( lower left corner )

Other corner: 7,5 ( upper right corner location )

## DRAWING ELLIPSES

### *Draw---Ellipses*

**Command:** *ellipse*

Arc/Center/<Axis endpoint 1>: 5,4 (Exp. Of drawing an Ellipse using axis & Eccentricity option)

Axis endpoint 2: 8,3

<Other axis distance>/Rotation: 2

### **Drawing Ellipse Using the Center & Two Axes option**

In this option you can construct an ellipse by specifying its center point, end point of one axis, and length of the other axis.

**Command:** *ellipse*

Arc/Center/<Axis endpoint 1>: C

Center of ellipse: 5,4

Axis endpoint: 7,4

<Other axis distance>/Rotation: 2

### **Drawing Elliptical Arcs**

AutoCAD will prompts you to enter information about the geometry of the ellipse and the arc limits. You can define the arc limits by using the following options:

1. Start & End angle of the arc;
2. Start and Included angle of the arc;
3. Specifying Start & End parameters.

## DRAWING REGULAR POLYGONS

### *Draw---polygons*

A regular polygon is a geometric figure with equal sides and equal angles. The default option prompts you to select a point that is taken as the center point of the polygon. The next prompt is **Inscribed in circle/Circumscribed about circle (I/C) <I>:.** A polygon is said to be Inscribed when it is drawn inside an imaginary circle and its vertices touch the circle and circumscribed when it is drawn outside the imaginary circle as shown in fig.

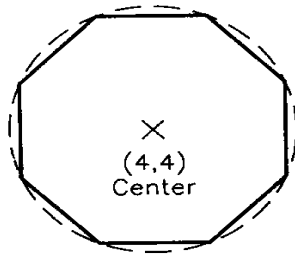
**Command: Polygon**

Number of sides <4>: 8

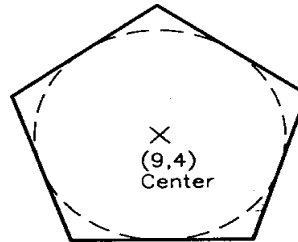
Edge/<Center of polygon>: 4,4

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

Radius of circle: 1.5



Inscribed octagon



Circumscribed pentagon

## DRAWING POLYLINES

### *Draw (POLYLINE)*

A **POLYLINE** is defined as a line that can have different characteristics.

Command: *PL*

From Point: *Pick the starting point or enter its coordinates.*

Current line width is 0.000

**Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>:**

### **End Point of Line:-**

This Option is maintained as the default, and is used to specify the end point of the current polyline segment.

### **Command: PL**

From Point: *Pick the starting point or enter its coordinates.*

Current line width is 0.000

Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>: *Specify the end point of 1<sup>st</sup> polyline*

Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>: *Specify the end point of 2<sup>st</sup> polyline*

### **Width**

If the current polyline width needs to be changed, you can do this by entering W (width option) at the last prompt.

Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>: W  
Starting Width<0.0000>: *specify the starting width (3)*.  
Ending Width<starting width>: *specify the ending width (0)*.

### **Close**

This option closes the polyline by drawing a polyline segment from the most recent end point to the initial start point, and on doing so exits from the PLINE command.

Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>: C

### **Halfwidth**

With this option specify the starting & ending halfwidth of a polyline. This halfwidth distance is equal to half of the actual width of the polyline.

Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>: H  
Starting halfwidth<0.0000>: *specify the starting halfwidth*.  
Ending halfwidth<starting width>: *specify the ending halfwidth*

### **Length**

This option prompts you to enter the length of a new polyline segment. It will be drawn at the same angle as that polyline segment or tangent to the previous polyarc segment.

Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>: L  
Length of line: *specify the desired length of the Pline*.

### **Undo**

This option erase the most recently drawn polyline segment

Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>: U

### **Arc**

This option is used to switch from drawing polyline to drawing polyarc.

Arc/Close/Half width/Length/Undo/Width/<Endpoint of line>: A  
**Angle/CENTER/Close/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/<Endpoint of arc>:**

**Angle:** This option prompts you to enter the Included Angle for the arc.

Included angle: *specify the included angle*.  
Center/Radius/<Endpoint>:

The center refers to the center of the arc segment, Radius refers the radius of the arc, and Endpoint draws the arc.

**Center:** This option prompts you to specify the center of the arc to be drawn.

Angle/Length/<Endpoint>:

Angle refers to the included angle, Length refers the length of the chord, and Endpoint refers to the end point of the arc.

**Close:** This option closes the polyline by drawing a polyarc segment from the previous end point to the initial start point, and on doing so exits from the PLINE command.

**Direction:** The Direction option allows you to specify Tangent Direction of your choice for the arc segment to be drawn. The prompt will be:

Direction from start point:

End point: *specify the end point of arc.*

**Halfwidth:** This option is the same for the Line option and prompts you to specify the starting and ending halfwidth of the arc segment.

**Line:** This option takes you back to the Line Mode.

**Radius:** This option prompts you to specify the Radius of the arc segment.

Radius: specify the radius of the arc segment.

Angle/<Endpoint>

**Second pt:** This option selects the second point of the arc in the three-point arc option.

Second point: *specify the second point on the arc.*

End point: *specify the third point on the arc.*

**Undo:** This option reserves the changes made in the previously drawn segment.

**Width:** This option prompts you to enter the width of the arc segment. The prompt sequence is identical to that of the polyline.

## DRAWING DOUGHNUTS

### *Draw---Circle (DONUT)*

The DONUT command is issued to draw an object that looks like a filled circle ring called a doughnut.

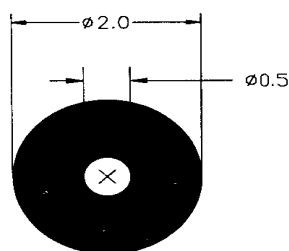
#### **Command: DONUT**

Inside diameter <current>: 0.75

Outside diameter <current>: 2

Center of doughnut: 2,2

Center of doughnut:



## DRAWING POINTS

The point is the basic drawing object.

Command: POINT

Point: specify the location where you want to plot the point.

When a point is drawn a mark appears on the screen. This mark is known as a Blip.

### Changing the Point Type

#### Format-----Point style

The point style Dialog box will seen on the screen as shown in fig. You can select a point style in this box by clicking your pointing device on the point style of your choice. A box is formed around that particular point style to acknowledge the selection made. Next, click on the OK Button.

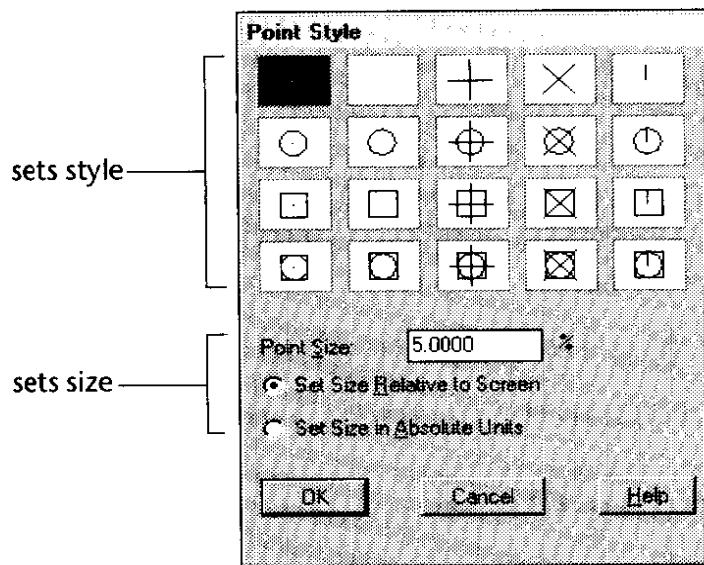
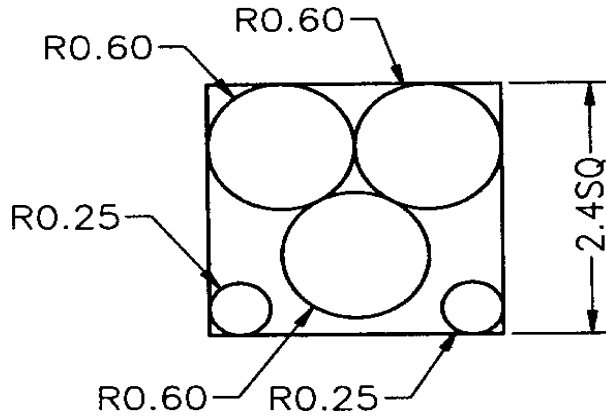


Fig. Courtesy From Autodesk Release-14 Manual

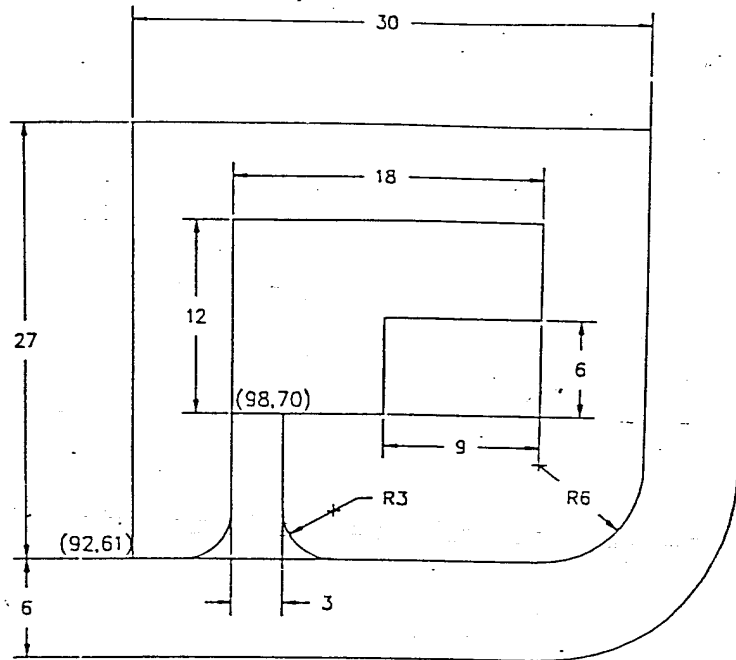
**Exercise No.2-Draw the following drawing by giving the specifications as given**

Drawing Limits: (1,1) & (10,7)

Draw option used: Line, Circle



**Exercise No.3-Draw the following drawing using Line & Arc commands**





The various editing commands are:

1. Move
2. Copy
3. Offset
4. Rotate
5. Scale
6. Fillet
7. Chamfer
8. Trim
9. Extend
10. Stretch
11. Lengthen
12. Array
13. Mirror
14. Break

All these commands are available in single Toolbar known as **MODIFY** Toolbar.

## **MODIFY**

**1. Move:-** Sometimes the objects may not be located where they should be. In such situation use the MOVE command.

**Command: *Move***

Select objects: *Choose objects individually or use the selection set.*

Base point or displacement: *Specify any point on or near the object.*

Second point of displacement: *Select the new location by specifying a point on the screen.*

**2. Copy:-** This command is used to copy an existing objects. This command is similar to the MOVE command in the sense that it makes copies of the object and places them at specified location, but the originals are left intact.

**Command: *Copy***

Select objects: *Choose objects to copy.*

<Base point or displacement>/Multiple: *Pick the base point.*

Second point of displacement: *Specify new location on the screen.*

The option of the COPY command is used to make MULTIPLE copies of the same object.

**Command: Copy**

Select objects: Choose objects to copy.

<Base point or displacement>/Multiple: M

Second point of displacement: Specify new location on the screen.

Second point of displacement: Specify new location on the screen.

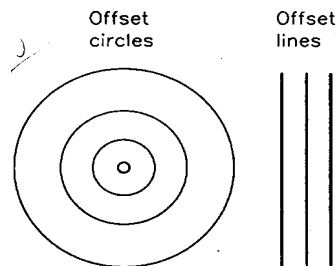
Second point of displacement: Specify new location on the screen.

**3. Offset:-** If you want to draw parallel lines, polylines, concentric circles, arcs, curves, etc., use the OFFSET command as shown in fig.

**Command: Offset**

Offset distance or through<current>: Enter a value

Select object to offset: Pick the object offset. Side to offset? Specify the side for offsetting.



**4. Rotate:-** Sometimes when making drawings you may need to rotate an object or a group of objects. It can be accomplished by using ROTATE command.

**Command: Rotate**

Select objects: *Choose objects to rotate.*

Base point : *Specify a base point on or near the object.*

<Rotation angle>/Reference: *Enter a positive or negative rotation angle, or pick a point.*

**5. Scale:-** Many times you will need to change the size of a drawing and this can be done by the SCALE command. This command enlarges or shrinks the entire object or a part of it in the same ratio for the X & Y dimensions.

**Command: Scale**

Select objects: Choose objects to scale.

Base point : Specify a base point on.

<Scale factor>/Reference: Enter any value

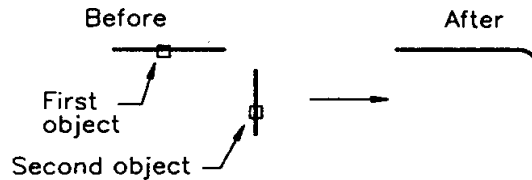
0.25 For scaling  $\frac{1}{4}$  of the actual size

2.0 For scaling double of the actual size.

**6. Fillet:-** This command is used to create smooth round arcs to connect two objects.

Command; Fillet

Polyline/Radius/Trim/<Select first object>:



**7. Chamfer:-** In drawing, chamfer is defined as the taper provided on a surface. Sometimes the chamfer is used to avoid a sharp corner.

**Command: Chamfer**

Polyline/Distance/Angle/Trim/method/<Select first object>:

**8. Trim:-** The TRIM command trims objects that extend beyond a required point of intersection. With this command you must select the cutting edge or boundary first. There can be more than one cutting edge. After the cutting edge or edges are selected you must select the object to be trimmed.

**Command: Trim**

Select cutting edges:

Select objects:

<select object to trim>/Project/Edge/Undo:

**9. Extend:-** This command may be considered the opposite of the TRIM command. In the EXTEND command you can lengthen or extend lines or arcs to meet other lines, arcs, polylines, circles, and rays.

**Command: Extend**

Select boundary edges:

Select objects:

**10. Stretch:-** The term stretching an object means changing only one of its dimensions; i.e., changing its length, width, or height .

**Command: Stretch**

Select objects to stretch by crossing –window or –polygon.

Select objects: *Pick the objects using crossing window or polygon.*

Select objects: *Enter*

**11. Lengthen:** -Like the TRIM & EXTEND commands, the LENGTHEN command can be used to extend or shorten lines. The LENGTHEN command has several options that allow you to change the length of objects by dynamically dragging the object end point, entering the delta value, entering percentage value, or entering total length of the object.

**Command: Lengthen**

Delta/Percent/Total/Dynamic/<Select object>:

**12. Array:** - In some drawings you may need to specify an object multiple times in rectangular or circular arrangement.

- a. *Rectangular array*
- b. *Polar array*

a. **Rectangular array:** A rectangular array is formed by making copies of the selected objects along the X and Y-axes. The command allows you to choose the number of rows and columns. The rows & columns must be whole numbers as shown in fig.

**Command: Array**

Select object:

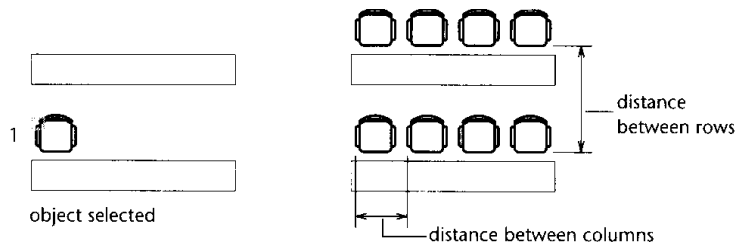
Rectangular or Polar array(R/P)<current>: R

Number of rows(---)<1>:

Number of columns(||||)<1>:

Unit cell or distance between rows(---):

Distance between columns(||||):



b. **Polar array:** A polar array is an arrangement of objects around a point in a circular pattern as shown in fig.

**Command: Array**

Select object:

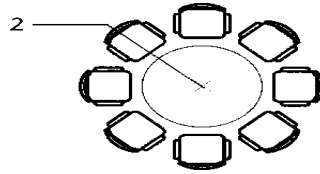
Rectangular or Polar array(R/P)<current>: P

Center point of array:

Number of items:

Angle to fill (+=ccw,-==cw)<360>:

Rotate objects as they are copied?<Y>:



**13. Mirror:-** The MIRROR command creates a mirror copy of the selected objects; the objects can be mirrored at any angle. This command is helpful in drawing symmetrical figures.

**Command: Mirror**

Select objects: *Pick the objects to be mirrored.*

First point of mirror line: *Specify the first point.*

Second point: *specify the second end point.*

Delete old objects?<N>: *Enter Y for deletion.*

**14. Break:-** This command cuts existing objects into two or erases portions of the object.

a. 1 Point Option:- Using this option, you can break the object into two parts.

b. 1 Point Select Option: An object can be broken into two parts.

c. 2 Point Option:- Object can be broken down into two selected points.

d. 2 Point Select Option:- This option is similar to 2 point option; the only difference is that instead of making the selection point as the first break point, you are allowed to pick a new first point.

**Command: Break**

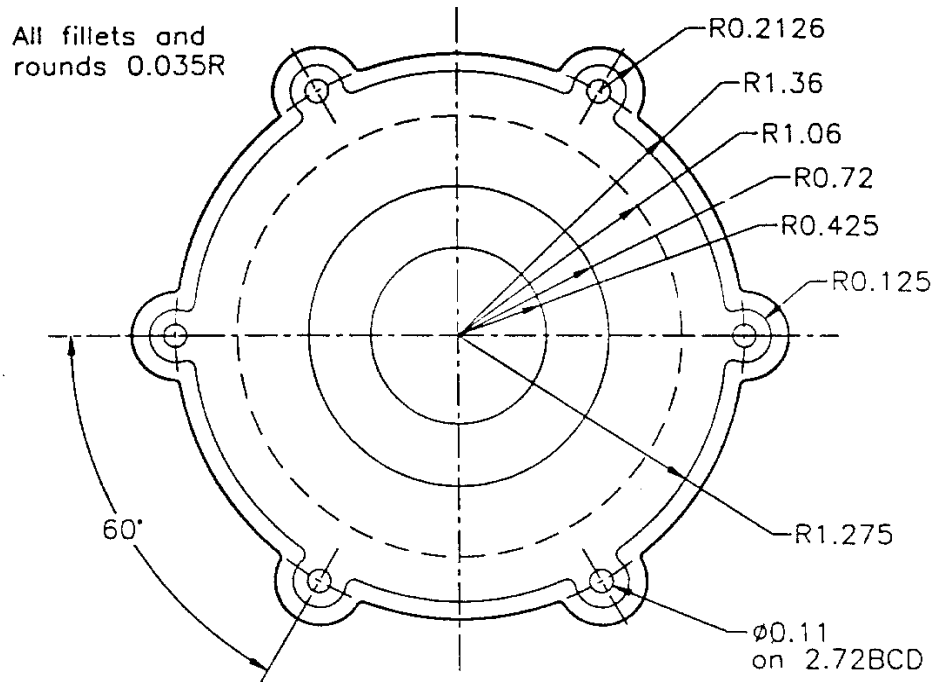
Select object: *Pick the object to be broken.*

Enter second point ( or F for first point): F

Enter first point: *Pick a new first break point.*

Enter second point. *Pick the second break point.*

**Exercise No.4 Draw the following drawing using Arrays, Offset, Circle commands.**



## DRAWING UNITS

### *Format ----UNITS*

The **UNITS** command is used to choose a format for the units of distance and angle measurement. From the **UNITS CONTROL** dialogue box select a desired format of units or angles by selecting the button to the left of the item. The precision of units and angles can also be selected by selecting the arrow in the Precision as shown in fig.

#### Specifying Units Format

1. *Scientific*:-  $1.75E+1, 3.5E+1$
2. *Decimal*:-  $1.75, 0.72$
3. *Fractional*:-  $10\text{-}3/8, 8\text{-}3/4$
4. *Engineering*:-  $0'10.375, 0'8.75$
5. *Architectural*:-  $1'1\text{-}3/4'', 3\text{-}1/4$

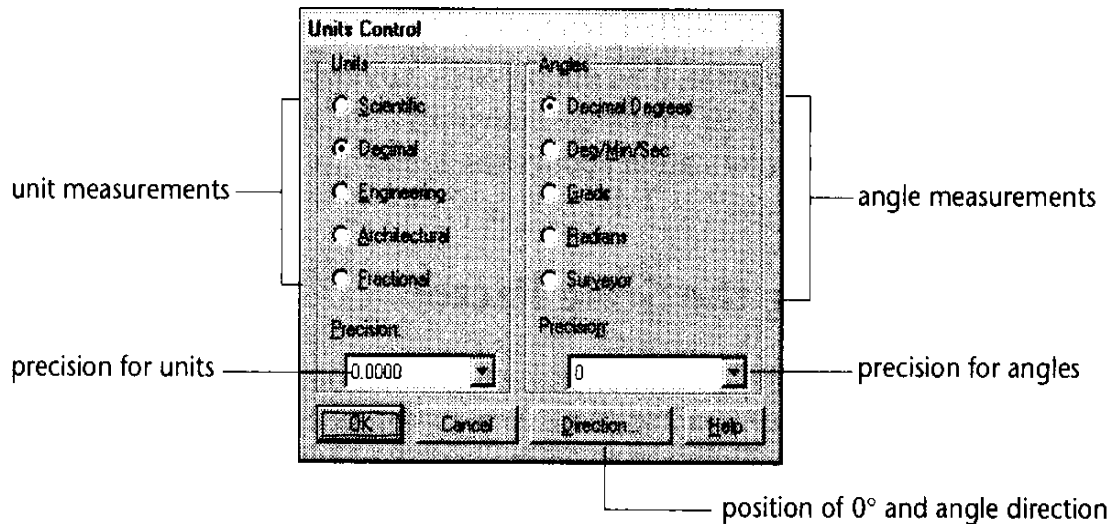


Fig. Courtesy from Autodesk Release-14 Manual

#### Specifying Angle Format

1. *Decimal Degrees*:-  $45.5$
2. *Degrees/minutes/seconds*:-  $90d30'45''$
3. *Grades*:-  $75g$
4. *Surveyor's units*:-  $N45d30'E$

**Radian** = Degrees x  $3.14159/180$

**Degrees** = Radians x  $180/3.14159$

**Grads** are generally used in land surveying. There are 400 grads or 360 degrees in a circle. A 90-degree angle is equal to 100 grads.

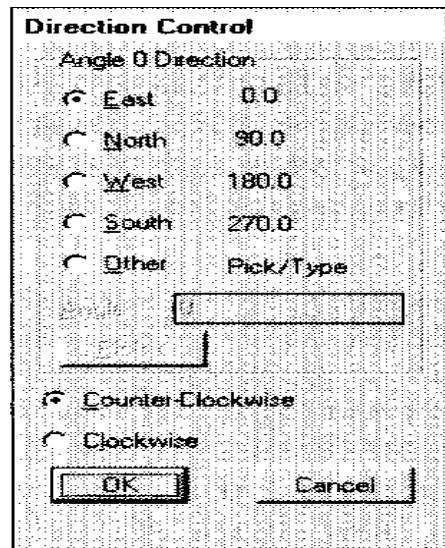


Fig. Courtesy from Autodesk Release-14 Manual

## DRAWING LIMITS

### *FORMAT---DRAWING LIMITS*

The use of LIMITS command is to set up new limits. The default limits are 12.00,9.00 when AutoCAD is started. Limits are needed to size up a drawing area. The limits of the drawing are usually determined by the following factors:

1. The actual size of the drawing.
2. Space needed for putting down the dimensions, notes, bill of materials, etc.
3. Space between different views so that the drawing does not look cluttered.
4. Space for the border and a title block, if any.

#### **Command: limits**

ON/OFF/<Lower left corner><current>:0,0

Upper right corner<current>: 30,18 ( you can specify values according to your drawing)

#### **Standard Sheet Sizes**

When you make a drawing, you might want to plot the drawing to get a hard copy. Several standard sheet sizes are available to plot your drawing. It is always recommended that you should select the work area based on the sheet size that you will be using to plot the drawing.



## Sheet sizes Metric system

Letter size	Sheet size
A4	210 x 297
A3	297 x 420
A2	420 x 594
A1	595 x 841
A0	841 x 1189

## LAYERS

### *Format-----Layers*

### *Object Properties----Layers*

In manual drafting, different details of the drawing can be drawn on different sheets of paper, or overlays. In AutoCAD, instead of using overlays use layers. Each layer is assigned a name. Colour & linetype can also be assigned to these layers.

### Advantages of Layers

1. Each layer can be assigned a different colour.
2. The layers are useful for some editing operations.
3. Layer can be off or freeze that is not desirable in plotting.
4. Layer can be locked to prevent the user from accidentally editing the objects that layer.
5. The colour is also helpful in distinguishing different groups of objects.

The Layer Control Dialogue Box is seen on your screen as shown in fig.

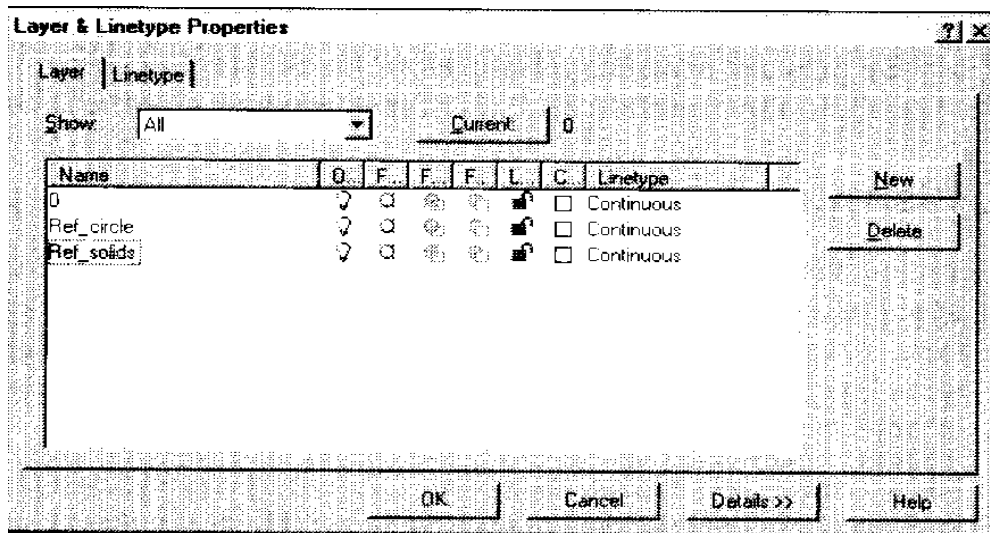


Fig. Courtesy from Autodesk Release-14 Manual

## **Creating New Layers**

The rectangular box just below the New, Current, Rename buttons is called the Layer edit box. Type the name of the layer, then pick the New button. The layer names will be displayed in the Layer Name list box.

## **Assigning a Linetype Or a Colour to a Layer**

To assign a linetype or a colour to a layer, select the layer or layers that to assign a linetype or colour. Pick the **SET LTYPE** button from the dialogue box for selection.

To assign a colour, select the layer and then pick the **SET COLOUR** button from the dialogue box.

## **Controlling Display of Layers**

Display of the layers can be controlled by selecting the layers and then picking the On, Off, Thaw, Freeze, Unlock, or Lock button.

With the **ON and OFF** option you can turn the layer on or off. The layers that are turned on are displayed on the screen and can be plotted. The layers that are turned off are not displayed on the screen and cannot be plotted.

While working on a drawing, if you do not want to see certain layers you can use the **FREEZE** option to freeze the layers. The frozen layers are invisible and cannot be plotted. The difference between the OFF and the FREEZE option is that the frozen layers are not calculated by computer while regenerating the drawing.

While working on a drawing, if you do not want to accidentally edit some objects on a particular layer but you still need to have them visible, you can use LOCK option to lock the layers.

## **DRAWING AID DIALOGUE BOX**

### ***Tools-----DRAWING AIDS***

Drawing Aids dialogue box can be used to set grid, snap, and drawing modes such as ortho, solid fill, blips, etc. as shown in fig.

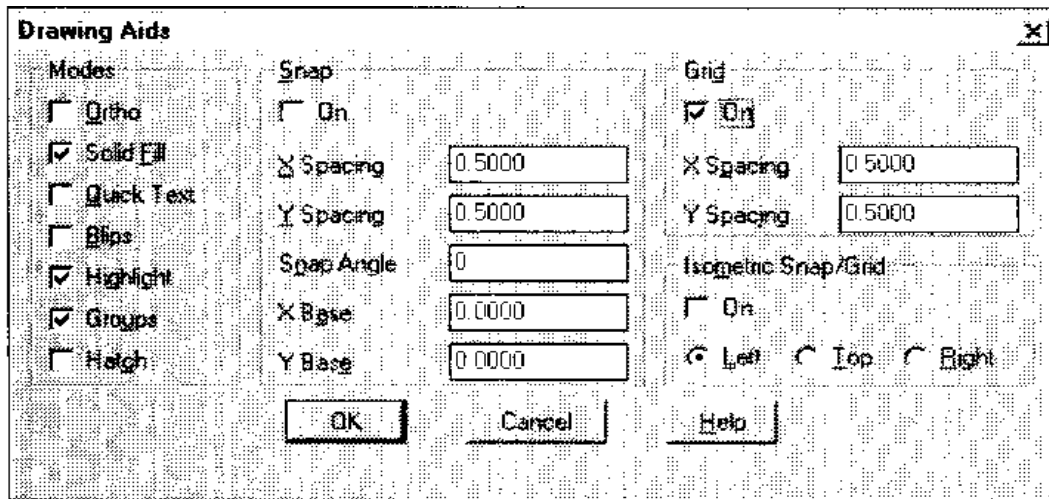


Fig. Courtesy from Autodesk Release-14 Manual

### Grid Command

The GRID command can be used to display dotted lines on the screen at predefined spacing. These dotted lines act as a graph that can be used as reference lines in a drawing. Distance between the grid lines can be changed according to the requirements. The grids can be turned ON & OFF by pressing the F7 function key.

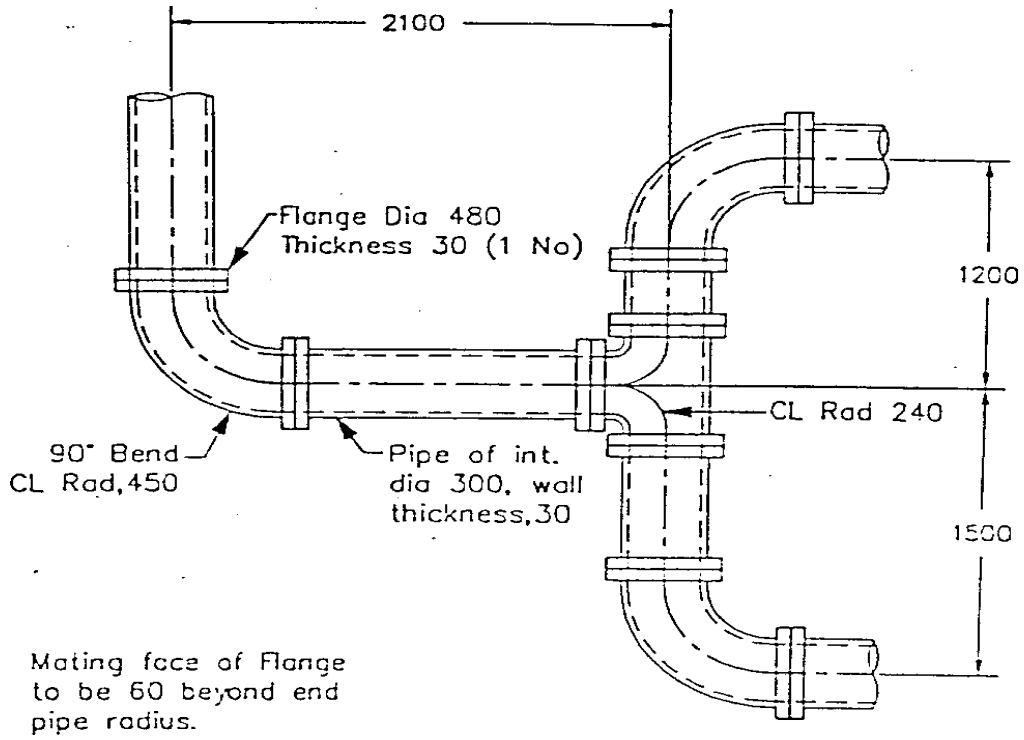
### Snap Command

The snap is used to set increment for cursor movement. The SNAP command allows to set up an invisible grid that allows the cursor to move in fixed increments from one snap point to another. SNAP command can be turned ON & OFF by using the function key F9.

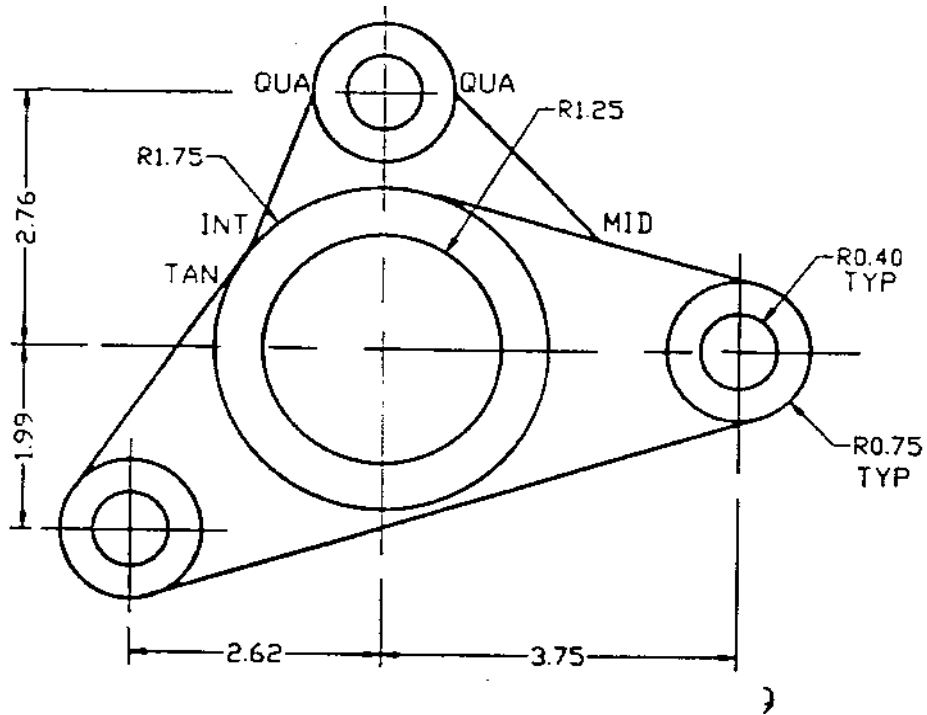
### Ortho Command

The ORTHO command allows you to draw lines at right angles only. ORTHO command can be turned ON & OFF by pressing F8 functional key.

**Exercise No.5- Draw the following drawing using layers & limits.**



**Exercise No. 6- Draw the following drawing using various options of layers.**



## OBJECT SNAPS

### TOOLS---OBJECT SNAP

Object Snap is one of the most powerful features of AutoCAD. It improves performance & accuracy of drawing and make drafting much simpler than it normally would.

These snaps can be selected from the Object Snap toolbar, from the Standard Toolbar by selecting the desired icon from the Object Snap flyout.

The following are the object snap modes available in AutoCAD:

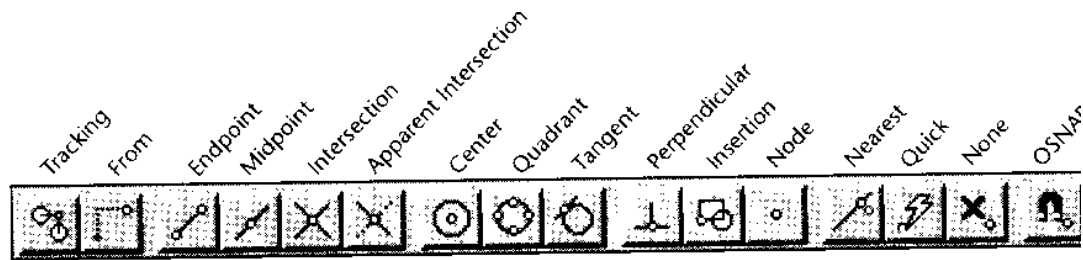


Fig. Courtesy from Autodesk Release-14 Manual

**NEArest:-** The NEArest object snap mode selects a point on an object ( line, arc, circle) that is visually closet to the graphics cursor.

**Command: line**

From point: **NEA**

to select a point near an existing object.

To point: *Enter*

**ENDpoint:-** The ENDpoint object snap mode snaps to the closest end point of a line or an arc.

**MIDpoint:-** The MIDpoint object snap mode snaps to the midpoint of a line or an arc.

**TANgent:-** The TANgent object snap allows you to draw a tangent to or from an existing circle or arc.

**CENter:-** The CENter object snap mode allows you to snap to the center point of an arc or circle.

**INTersect:-** The INTersection object snap mode is used if you need to snap to a point where two or more lines or arcs intersect

**QUAdrant:-** This option is used when you need to snap to a quadrant point of an arc or circle. A circle has four quadrants and each quadrant subtends an angle of 90 degrees. The quadrant points are located at 0,90,180,270-degrees positions.

**PERpendicular:-** The PERpendicular object snap mode is used to draw a perpendicular on a line. When you use this mode and select an object, AutoCAD calculates the point on the selected object so that the previously selected point is perpendicular to the line.

**QUICK:-** IF you are using an object snap when several objects cross the target box, AutoCAD normally searches all objects for the specified object snap mode and select the closest point. Depending on the hardware setup, this will cause delay in searching the specified point. The QUICK option stops searching as soon as AutoCAD finds a point on the object with the specified object snap.

**NODE:-** The NODE object snap can be used to snap to a point object.

Command: line  
From point: NOD  
Of select point P1  
To point: NOD  
Of select point P2  
To point: NOD  
Of select point P3

**INSert:-** The INSert object snap mode is used to snap to the insertion point of a text, shape, block, attribute, or attribute definition.

**NONE:-** The NONE object snap mode turns off any object snap and returns to the normal command prompt.

**Command: line**  
From point: MID  
To NONE  
Invalid point.  
From point:

**APParent Intersection:-** The APParent Intersection object snap mode is similar to the INTersection snap mode, except that this mode selects the visual intersection. The visual intersections are those intersections that are not present on the screen but are imaginary ones that can be formed if any two objects are extended.

## GIVING TEXT TO YOUR DRAWING

### *DRAW---TEXT*

The TEXT command lets you write text on a drawing. AutoCAD provides a number of different character patterns, known as FONTS.

### **Text Alignment**

Alignment refers to the layout of the text. The main text alignment modes are left, center, & right.

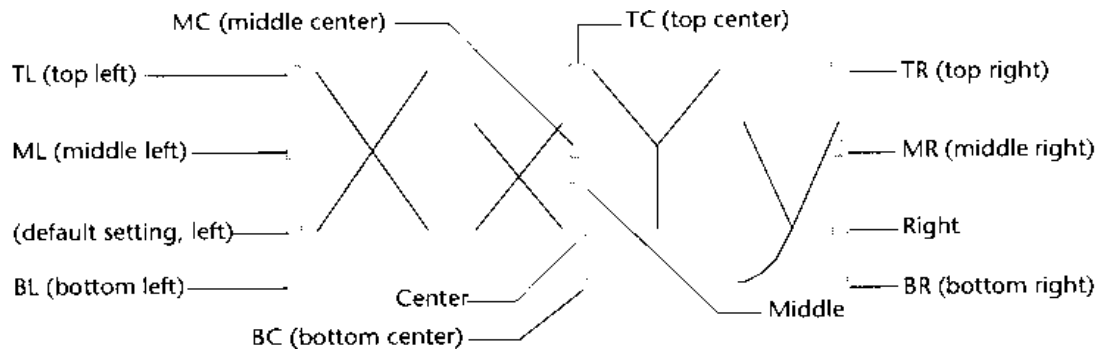


Fig. Courtesy from Autodesk Release-14 Manual

### **Command: Text**

Justify/Style/<Start point>:

**Start point option:-** By specifying a start point, the text is left-justified along its baseline starting from the location of the starting point. AutoCAD must know the text height, the rotation angle for the baseline, and the text string to be drawn.

Height<default>:

Rotation angle:

Text: Enter a text string

**Justify option:-** When the Justify option is invoked, the user can place text in one of the 14 various alignment types by selecting the desired alignment option as shown in fig..

### **Command: text**

Justify/Style/<Start point>: J

**Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR:** *select any of these option.*

**Align:** The text string is written between two points.

**Fit:** This is similar to the previous option. The only difference is that in this case select the text height and it does not vary according to the distance between the two points.

**Center:** You can use this option to select the midpoint of the baseline for the text.

**Middle:** Using this option you can enter a text not only horizontally, as in the previous case, but also vertically.

**Right:** This option is similar to the start point option. The only difference is that the text string is aligned with the lower right-corner i.e. the text is right justified.

**TL option:** In this option the text string is justified from the Top left.

**TC option:** Justify the text at the Top Center.

**TR option:** Justify the text at the Top Right.

### **Style option**

With this option you can load another text style from already existing ones. Different text styles have different text fonts, height, obliquing angles, and other features.

#### **Command: *text***

Justify/Style/<Start point>: S

Style name ( or ? ) <current>:

If you want to activate another text style, enter the name of the style at the last prompt. You can also choose from a list of variable text style, which can be displayed by entering ?. Next prompt will be

Text style(s) to list <\*>:

Press the enter key to display the available text styles.

### **Dtext Command**

Dtext stands for dynamic text. This command performs all the functions that the TEXT command does, and in addition, you can see the text on the screen as you type it. It also lets you enter multiple lines in one command.

### **Drawing Special Characters**

In all drafting applications, special symbols or characters are required like DEGREE symbol or the DIAMETER symbol. For each symbol the control sequence starts with a percent sign written twice (%%). The control sequences for some of the symbols are:

<b>Control sequence</b>	<b>Special character</b>
%%c	<i>Diameter symbol ( )</i>
%%d	<i>Degree symbol ( )</i>
%%p	<i>Plus/minus tolerance symbol ( )</i>
%%o	<i>Toggle for overscore mode on/off ( )</i>
%%u	<i>Toggle for underscore mode on/off ( )</i>
%%%	<i>Single percent sign ( % )</i>



## DIMENSIONING TECHNIQUES

To make design more informative and practical, the drawing must convey more than just the graphic picture of the product. To manufacture an object, the drawing must contain size descriptions such as the length, width, height, angle, radius, diameter, and location of features. All this information is added to the drawing with the help of dimensioning.

### FUNDAMENTAL DIMENSIONING TERMS

It is important to know and understand various dimensioning terms that are common to linear, angular, radial, diameter, and ordinate dimensioning as shown in fig.

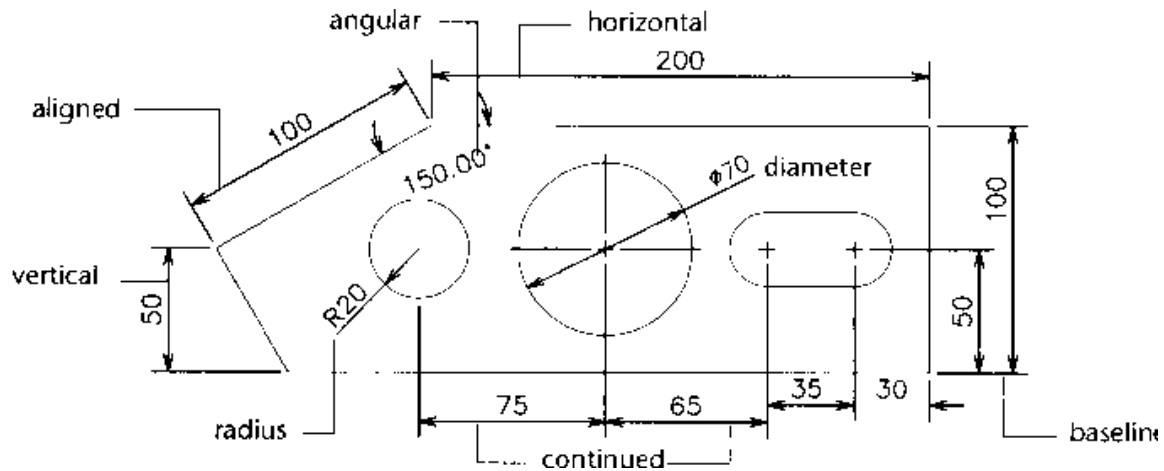


Fig. Courtesy from Autodesk Release-14 Manual

### Dimension Line

The dimension line indicates which distance or Angle is being measured.

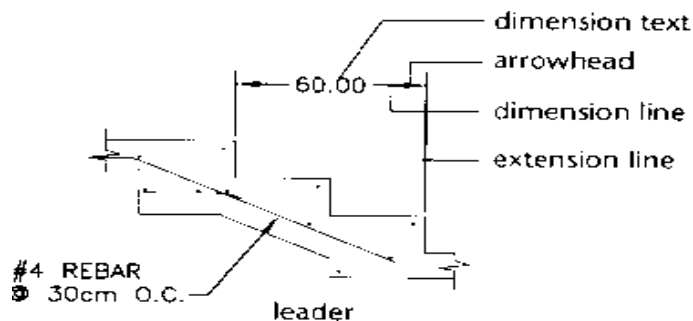


Fig. Courtesy from Autodesk Release-14 Manual

### **Dimension Text**

Dimension Text is a text string that reflects the actual measurement between selected points as calculated by AutoCAD.

### **Arrows**

An arrow is a symbol used at the end of a dimension line. Arrows are also called terminator because they signify the end of the dimension line.

### **Extension line**

Extension lines are drawn from the object measured to the dimension line. These lines are also called witness line.

### **Leader**

A leader is a line that stretches from the dimension text to the object being dimensioned. Sometimes the text for dimensioning and other annotations do not adjust properly near the object. In such cases use a leader and place the text at the end of the leader line.

### **Center Mark & Center Lines**

The center mark is defined as a cross mark that identifies the center point of a circle or an arc. Center lines are mutually perpendicular lines passing through the center of the circle/arc and intersecting the circumference of the circle/arc.

### **Alternate units**

With the help of alternate units you can generate dimensions for two systems of measurement at the same time. For exp., if the dimensions are in inches, you can use the alternate units dimensioning facility to append metric dimensions to the dimensions.

### **Tolerances**

Tolerance is defined as the amount by which the actual dimension can vary. AutoCAD can attach the plus/minus tolerances to the dimension text. This is also known as variance-style tolerance. You can give tolerance symbols, material condition, tolerance value, datum value. In material condition the symbol (M) stands for Maximum Material Condition, (L) stands for Least Material Condition & (S) stands for Regardless of Feature Size. Datum is a reference points, axis, or plane to make measurements and to verify dimensions.

### **Limits**

Instead of appending the tolerances to the dimension text, you can apply the tolerance to the measurement itself. AutoCAD will automatically calculate the upper and lower limits of the dimension.

### **Associative Dimensions**

Associative dimensioning is defined as a method of dimensioning in which the dimension is associated with the object that is dimensioned. The dimension is influenced by the changes in the size of the object. If the associative dimension is disabled, the dimension lines, arrows, leaders, extension lines, and the dimension text are drawn as separate objects. You can edit the dimension lines, extension lines, text, arrowheads as individual objects.

### **Selecting Dimensioning Commands In AutoCAD**

By using Toolbar, you can select dimension commands.

### Linear Dimensioning

Linear dimensioning applies to those dimensioning commands that measure the distance between two points. The points can be any two points in the space, the end points of an arc or line, or any set of points that can be identified. Linear dimensions include Horizontal & Vertical dimensioning.

### Aligned Dimensioning

If an object is inclined at an angle, not parallel to the X axis or Y axis, in such situation use aligned dimensioning. You can measure true distance between the two points as shown in fig.

### Baseline Dimensioning

Sometimes in manufacturing you may want to locate different points and features of a part with reference to a fixed point. This can be accomplished by using Baseline dimensioning. With this command you can continue a linear dimension from the first extension line origin of the first dimension. There must already be a linear, ordinate, or angular associative dimension to use the Baseline or Continue dimensions.

### Continue Dimensioning

With the continue command you can continue a linear dimension from the second extension line of the previous dimension. This is also referred to as chained incremental dimensioning. The Continue & Baseline commands are used in a similar manner as shown in fig.

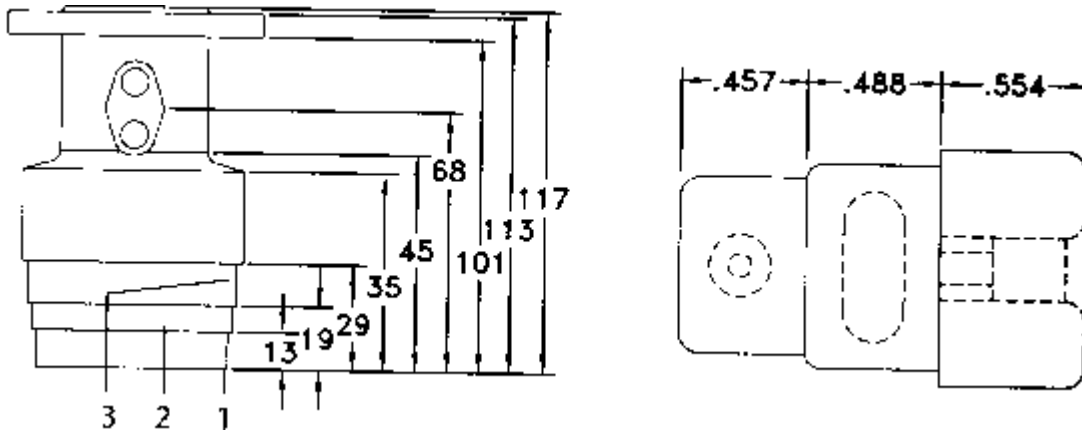


Fig. Courtesy from Autodesk Release-14 Manual

### Angular Dimensioning

Angular dimensioning is used when you want to dimension an angle. This command generates a dimension arc to indicate the angle between two non-parallel lines .

### Radial Dimensioning

Diameter dimensioning is used to dimension a circle, it can also be used to dimension an arc .

Radius Dimensioning is used to dimension an arc, it can also be used to dimension a circle.

### Generating Center Marks And Center Lines

When circles or arcs are dimensioned with the Radius or Diameter command, a small mark may be drawn in the center of the circle/arc. This mark is known as the center mark. This can be achieved by selecting the center mark icon from dimensioning toolbar.

### Ordinate Dimensioning

Ordinate dimensioning is also known as arrowless dimensioning because no arrowheads are drawn in this type of dimensioning. Ordinate dimensioning is also known as Datum dimensioning because all dimensions are related to a common base point. The current UCS origin becomes the reference or the base point for Ordinate dimensioning. With ordinate dimensioning you can determine the X or Y displacement of a selected point from the current UCS origin.

In Ordinate dimensioning, AutoCAD automatically places the dimension text ( X or Y coordinate value ) and the leader line along the X or Y axis. Since the ordinate dimensioning pertains to either the X coordinate or the Y coordinate, you should keep ORTHO on. When ORTHO is off, the leader line is automatically given bend when you pick the second leader line point that is offset from the first point as shown in fig.

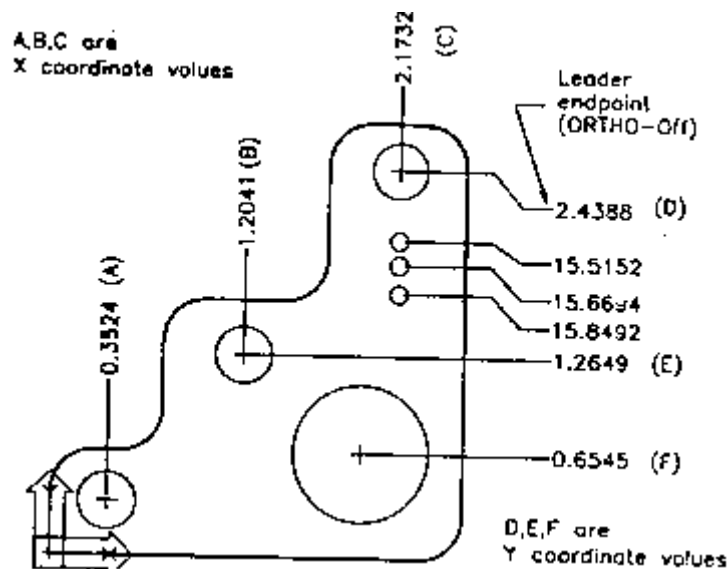


Fig. Courtesy from Autodesk Release-14 Manual

The leader line for an X coordinate value will be drawn perpendicular to the X axis and the leader line for a Y coordinate value will be drawn perpendicular to the Y axis.

### Leader

The LEADER command draws a line that extends from the object being dimensioned to the dimension text. The leader line is used to attach annotations to an object or when the user wants to show a dimension without using another dimensioning command .

#### Changing Dimension Settings

In Dimension Style dialogue box, there are three important parameters namely GEOMETRY, FORMAT, & ANNOTATION.

The GEOMETRY dialogue box is shown in fig. We will each part of it brief.

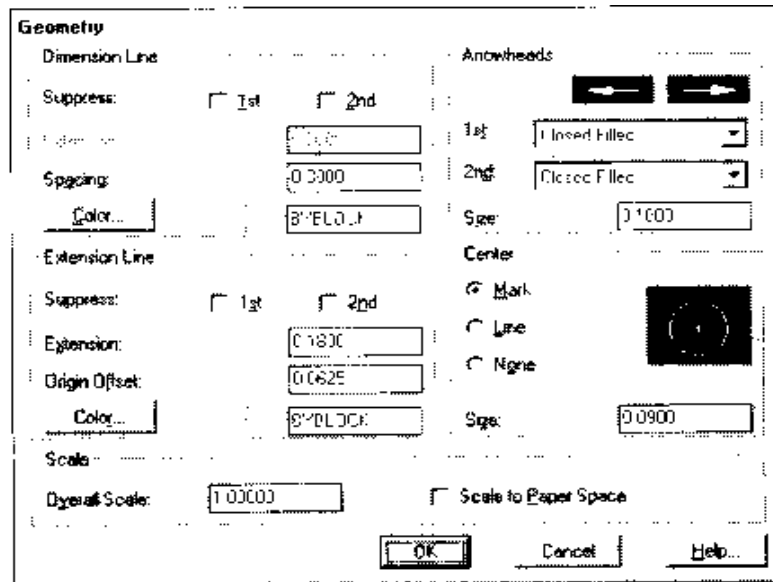


Fig. Courtesy from Autodesk Release-14 Manual

### Dimension Line

**Suppress:-** The suppress check boxes control the drawing of the first and second dimension line. By default both dimension lines will be drawn. You can suppress one or both dimension lines by selecting their corresponding check boxes.

**Extension:-** The EXTENSION edit box is used to specify the distance by which the dimension line will extend beyond the extension line. The Extension edit box can be used only when you have selected the oblique arrowhead in the Arrowhead pop-up list.

**Spacing:-** The Spacing ( Baseline Increment ) edit box is used to control the dimension line increment ( gap between successive dimension lines ) for the continuation of a linear dimension drawn with the BASELINE command.

### Extension Line

**Suppress:-** You can suppress one or both extension lines by selecting the corresponding check boxes.

**Extension:-** Extension is the distance the extension line should extend past the dimension line. You can change the extension line offset by entering the desired distance value in the Extension edit box.

**Origin Offset:-** You may have noticed a small space exists between the origin points you specify and the start of the extension lines. This space is due to specified offset.

## Arrowheads

*1<sup>st</sup>: and 2<sup>nd</sup>: pop-up boxes:-* AutoCAD has provided seven standard termination (arrowheads) symbols that you can apply at each end of the dimension line. In addition to these, you can create your own arrows at each end of the dimension line. If you select first arrowhead it is automatically applied to the second by default.

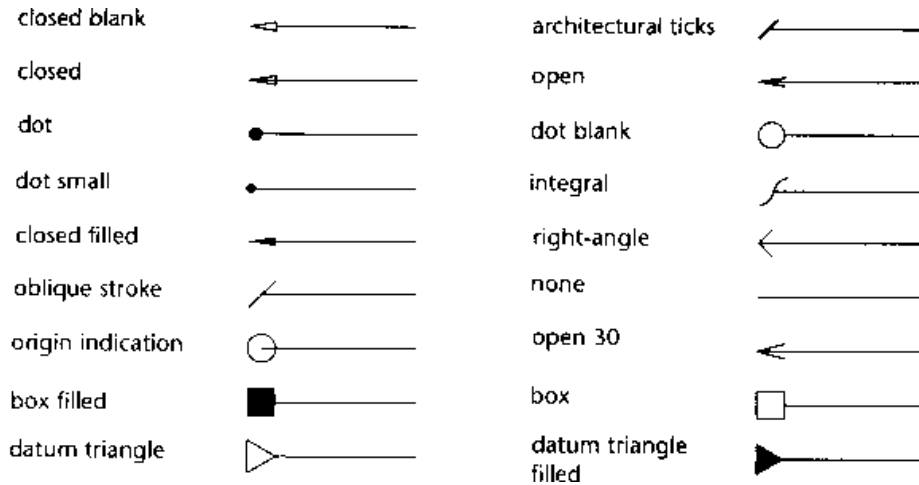


Fig. Courtesy from Autodesk Release-14 Manual

## Scale

**Overall scale:-** The current general scaling factor that pertains to all of the size-related dimension variable, like text size, center mark size, arrow size etc., is displayed in the Overall Scale edit box. You can alter the scaling factor of your requirement by entering the scaling factor of your choice in this box.

## Center

1. Mark:- if you want to have + mark on your circle or arc.
2. Line:- If you want to draw center lines for a circle or arc, pick LINE button.
3. None:- If you select none button, the center marks are not drawn.

## Controlling Dimension Format

You can control the dimension format through the FORMAT dialogue box as shown in the fig. The Format dialogue box has image tiles that update dynamically to display the text as the settings are changed. Individual items of the Format dialogue box are explained below.

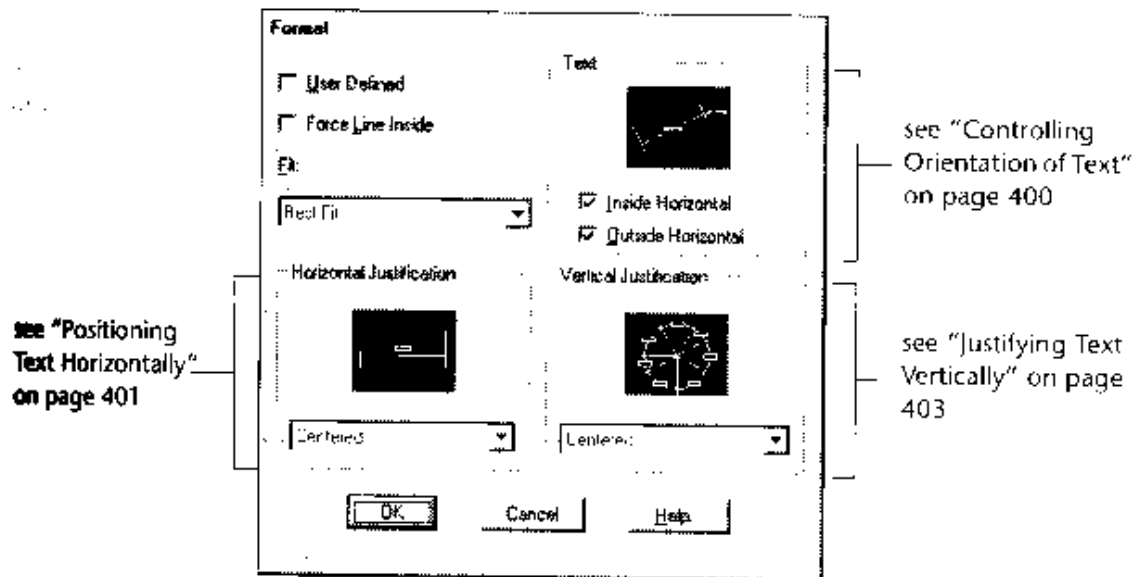


Fig. Courtesy from Autodesk Release-14 Manual

**User Defined:**- AutoCAD places the dimension text in the middle of the dimension line. By picking the User Defined, the dimension text can be place anywhere along the dimension line.

**Force Lines Inside:**- If the dimension text and the dimension lines are outside the extension lines and you want the dimension lines to be placed between the extension lines, Pick the Force Lines Inside check box.

**Fit:**- In the pop-up list, AutoCAD displays the available options for fitting the arrows and dimension text between the extension lines. The Fit options are:

1. Text and Arrows:- AutoCAD will place the arrow and dimension text between the extension lines if there is enough space available to fit both.
2. Text Only:- If space is not available for both arrows and text, the text is placed inside the extension lines and the arrows are placed outside the extension lines.
3. Arrows Only:- AutoCAD will place the arrow and dimension text between the extension lines if there is enough space available to fit both. If there is enough space to fit the arrows, the arrows will be placed inside the extension lines and the dimension text outside the extension lines.
4. Best Fit:- This is the default option. In this AutoCAD places the dimension where it fits best between the extension lines.
5. Leader:- AutoCAD creates leader lines if there is not enough space available to fit the dimension text between the extension lines. The horizontal justification determines whether the text is placed to the right or the left of the leader.

### ***Horizontal Justification***

Horizontal Justification controls the placement of the dimension text. The following is the available options

<b>Options</b>	<b>Description</b>
<i>Centered</i>	<i>Places text between extension lines</i>
<i>1<sup>st</sup> extension line</i>	<i>Places text next to first extension line</i>
<i>2<sup>nd</sup> extension line</i>	<i>Places text next to second extension line</i>
<i>Over 1<sup>st</sup> extension</i>	<i>Places text aligned with and above the 1<sup>st</sup> extension line</i>
<i>Over 2<sup>nd</sup> extension</i>	<i>Places text aligned with and above the 2<sup>nd</sup> extension line</i>

## **Text**

Use Text option to specify the alignment of the dimension text with the dimension line. These options can be used to control alignment of the dimension text for linear, radius, and diameter dimensions.

**Inside Horizontal:-** If you turn this check box off, the dimension text is aligned with the dimension line only when the dimension text is between the extension line.

**Outside Horizontal:-** If you turn this check box off, the dimension text is aligned with the dimension line only when the dimension text is outside the extension lines.

## **Vertical Justification**

The Vertical pop-up list controls the vertical placement of the dimension text. The current setting will be highlighted.

**Centered:-** The dimension text gets centered on the dimension line in such a way that the dimension line is split to allow for placement of the text.

**Above:-** The dimension text is placed above the dimension line, except when the dimension line is not horizontal and the dimension text inside the extension lines is horizontal.

**Outside:-** This option places the dimension text on the side of the dimension line.

**JIS:** This option lets you place the dimension text to conform to JIS representation.

## **Annotation Dialogue Box**

Use the Annotation dialogue box to control the dimension text format as shown in fig. AutoCAD lets you attach a user-defined prefix or suffix to the dimension text.

## **Primary Units**

When you select the Units... button, AutoCAD displays the Primary Units dialogue box as shown in fig. You can use this box to control Units, Dimension Precision, and Zero Suppression for dimension measurements.



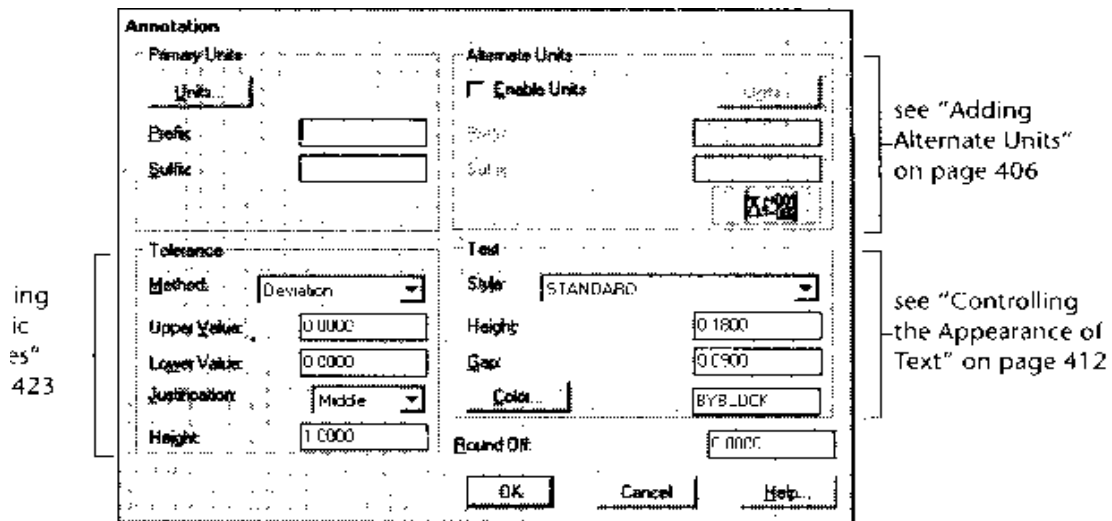


Fig. Courtesy from Autodesk Release-14 Manual

## Zero Suppression

If you want to suppress the leading Zeros in all of the distance measured in decimals, check the Leading check box. For exp., by picking this box, 0.0750 becomes .0750. If you want to suppress the trailing zeros in all the distances measured in decimals, check the Trailing check box. As in the above exp., 0.0750 becomes 0.075.

## Tolerance

The Tolerance area of the Annotation dialogue box lets you specify the tolerance method, tolerance value, justification of tolerance text, and the height of the tolerance text.

**Method:-** This allows you to select the tolerance method. The tolerance methods supported by AutoCAD are Symmetrical, Limits, Deviation, and Basic

**Symmetrical:-** In this the value specified in the Upper Value edit box is applied to both plus and minus tolerance. For exp., if the value specified in the upper value edit box is 0.005, the tolerance appended to the dimension is 0.005.

**Deviation:-** The value in the Upper Value and Lower Value edit boxes will be displayed as plus and minus dimension tolerance. For exp., if the upper value of the tolerance is 0.005 and the lower value of the tolerance is 0.002, the resulting dimension text generated will have a positive tolerance of 0.005 and a negative tolerance of 0.002.

**Limits:-** AutoCAD adds the upper value to the dimension text and subtracts the lower value from the dimension text.

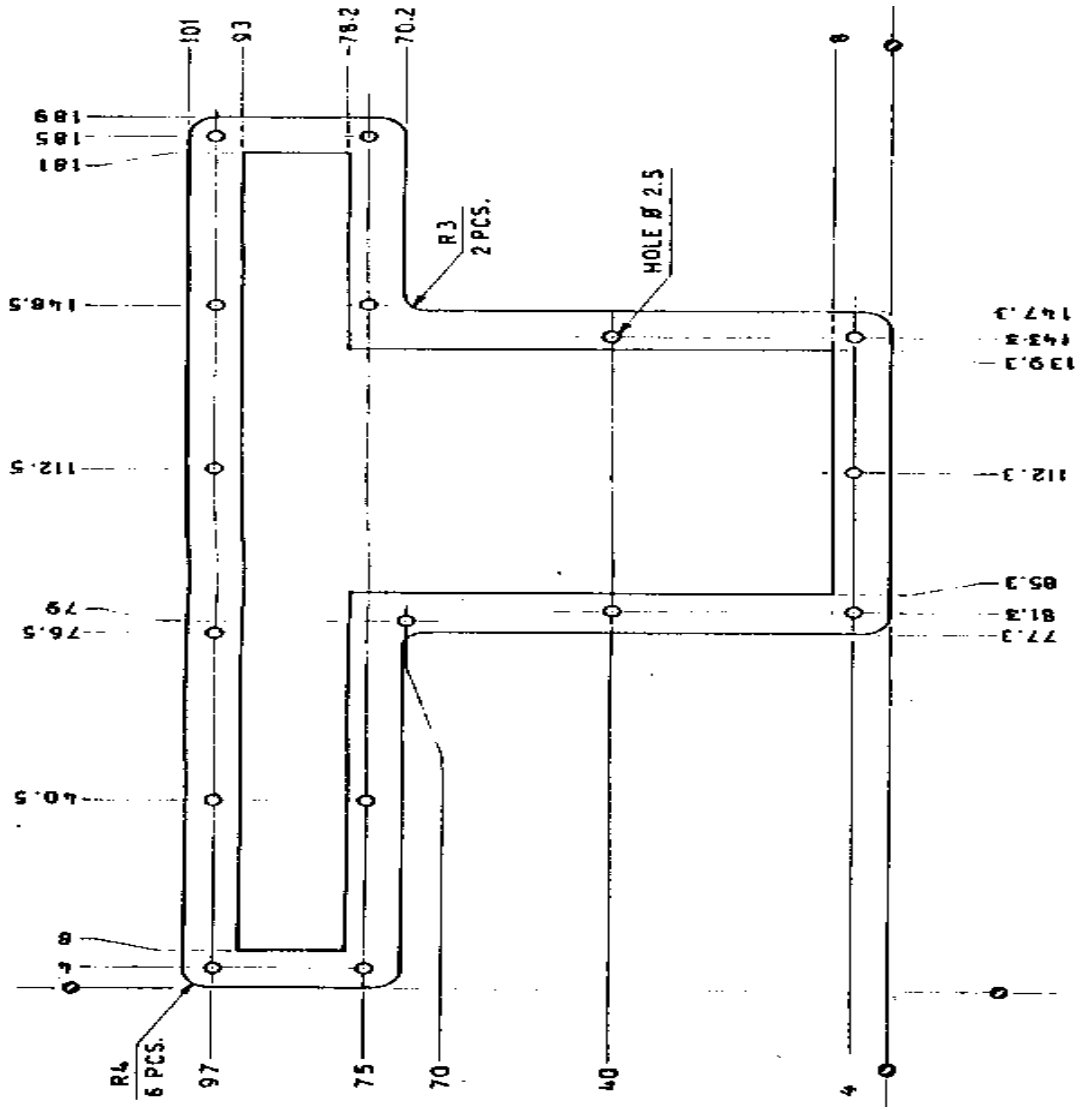
**Basic:-** a basic dimension text is dimension text with a box drawn around it. Reference dimensions are used primarily in geometric dimensioning and tolerances. The basic dimension can be realized by picking the basic tolerance method. The Basic dimension is also called a Reference dimension.

**Justification:-** Justification lets you justify the dimension tolerance text. Three justifications are possible Bottom, Middle, and Top.

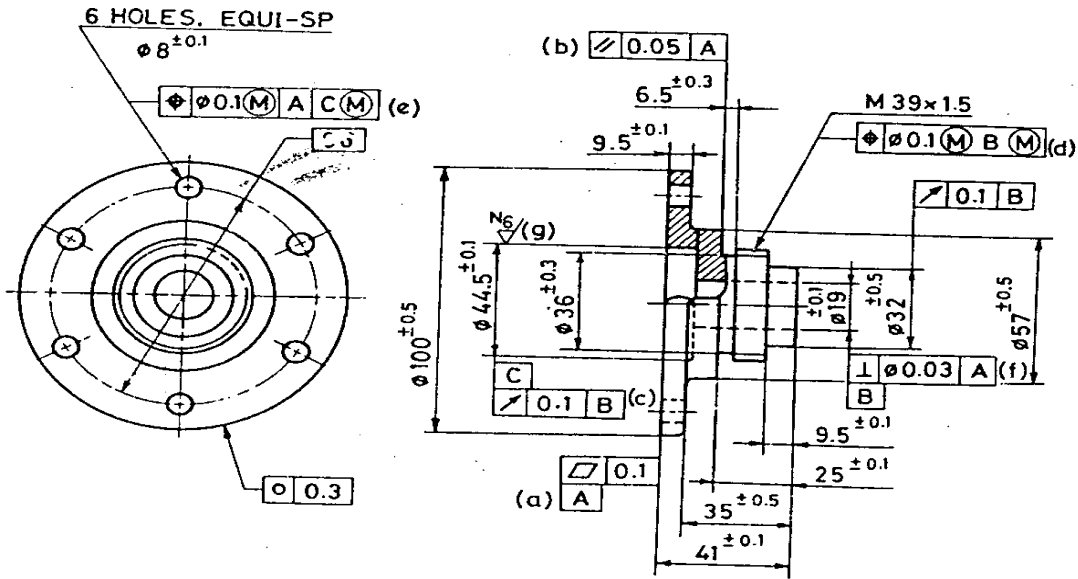
### **Text**

The TEXT area of the Annotation dialog box lets you specify the text style, text height, text gap, and the colour of the dimension text.

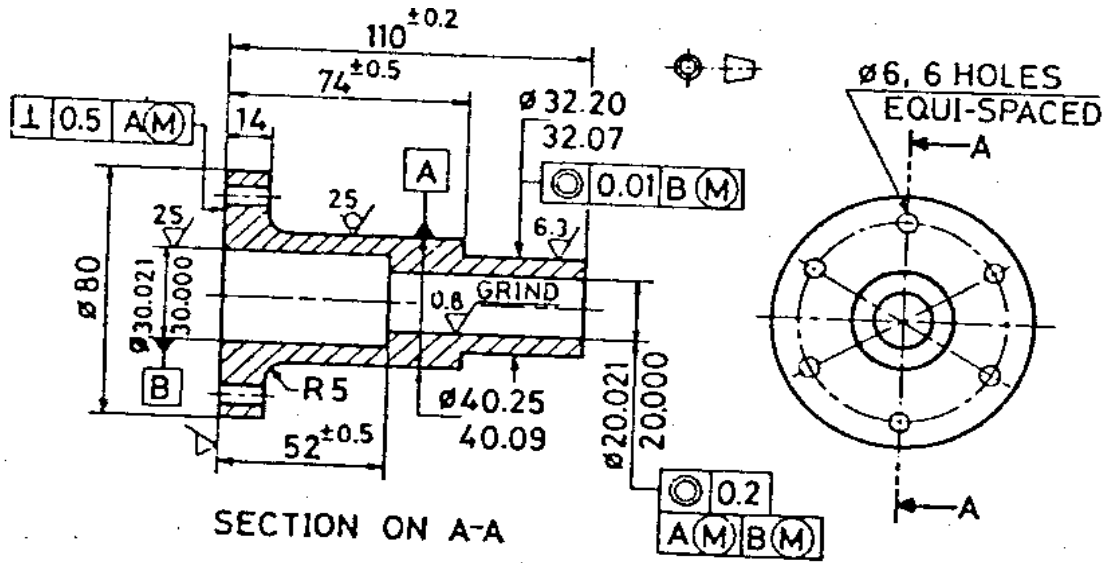
Exercise No.7-Draw the following diagram using draw tools and dimensioning.



Exercise No.8 Draw the following drawing and use proper dimensioning.



Exercise No.9 Draw the following drawing and use proper dimensioning.



## **HATCHING**

In many drawings, the area must be filled with some pattern. Different filling patterns make it possible to distinguish between parts or components of an object. Also, the material the object is made of can be indicated by the filling pattern. Filling the objects with a pattern is known as hatching.

### **The BHATCH Command**

The BHATCH or the boundary hatch command allows you to hatch a region enclosed within a boundary by picking a point inside the boundary or by selecting the object to be hatched. This command automatically designates a boundary and ignores any other objects that may not be a part of this boundary. The Boundary Hatch box can be called out from the DRAW Toolbar.

### **Boundary Hatch Options**

The Boundary Hatch dialogue box has several options that let you control various aspects of hatching, like pattern type, scale, boundary parameters, and associatively.

#### **Pattern Type**

This area lets you choose the type of hatch pattern you want. The default hatch pattern is predefined. AutoCAD displays a list of the available options, like User-defined, custom, and Predefined.

#### **User-Defined Hatch Pattern**

If you want to define a simple pattern, you can pick the User-Defined pattern in the pattern type list box. The boxes: the Spacing and the Double Hatch Check box will be enabled.

#### **Custom Pattern**

The second way to select a predefined hatch pattern is by selecting the custom pattern type, then entering the name of the stored hatch pattern in the Custom Pattern edit box. The custom pattern type can be selected by choosing the down arrow in the Pattern Type list box. You can do this if you know the name of the hatch pattern. For exp., you can enter Plastic or Steel as a hatch pattern. In AutoCAD all hatch patterns are stored in the file named ACAD.PAT. If AutoCAD doesn't locate the entered pattern in the ACAD.PAT file, it searches for it in a file with the same name as the pattern.

#### **Boundary Area**

The option available in the Boundary area of the boundary hatch box allow you to define the Hatch boundary by picking a point inside the area or by selecting the objects. The other options are for removing islands, viewing the selection, and setting advanced options.

### **Pick Points<**

This option makes AutoCAD automatically construct a boundary. To select this option, choose the Pick Points< button in the Boundary Hatch Dialogue box. The following prompts appears:

Select internal point: *Pick a point inside the object.*

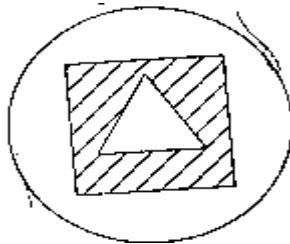
Selecting everything...

Selecting everything visible...

Analyzing the selected data.

Select internal point: *Pick a point or press the Enter key to end selection.*

1. Pick point by clicking Pick button
2. Pick point inside the object (Square in this case)
3. Pick another point inside the other object (Triangle in this case)
4. Then Apply.



### **Select objects<**

This option in the Boundary Hatch Dialogue box lets you select objects that form the boundary for hatching. AutoCAD prompts you to select objects. This option can also be used after a boundary has been defined. This way you can select any object that may be present within the boundary so that it is not hatched.

### **Remove Islands**

This option is used to remove any islands from the hatching area. For exp., if you have a rectangle and a circle and you use the Pick Points option to pick a point inside a rectangle, AutoCAD will select both circles and rectangle. To remove the circle you can use the Remove Islands option. When you select this option, AutoCAD will prompt you to select the islands to remove.

### **Inherit Properties<**

If you want to have the same hatching pattern and style as that of an existing hatch on the screen, pick the Inherit Properties button. The dialogue box is cleared from the screen and the following prompts appear:

Select hatch object: *select a hatch pattern.*

Select hatch object:

The Boundary Hatch dialogue box reappears on the screen with the name of the copied hatch pattern in the Pattern edit box. The inherited hatch pattern is also displayed in the hatch image box. The selected pattern now becomes the current hatch pattern

### **Advanced options**

Defining a boundary by specifying an internal point is quite simple in the case of small and less-complicated drawings. It may take more time in the case of large, complicated drawings because AutoCAD examines everything that is visible in the current viewport, so the larger and more complicated the drawing, the more time it takes to locate the boundary edged. When you select the Advanced button the Advanced Option dialogue box is displayed on the screen.

### **Object Type**

The Object Type option controls the type of object AutoCAD will create when you define a boundary. It has two options: Polyline and Region.

### **Define Boundary Set**

The boundary set comprises the objects that the BHATCH command uses when constructing the boundary. A boundary can be produced faster by specifying a boundary set because in this case AutoCAD does not have to examine everything on the screen.

### **Make New Boundary Set**

This option is used to create a new boundary set. All the dialogue boxes are cleared from the screen to allow selection of objects to be included in the new boundary set. While constructing the boundary set, AutoCAD uses only those objects that you select and are hatchable.

### **Style**

The hatching style can be specified by selecting a style from the style list box as shown in fig.s. There are three styles from which you can choose: Normal, Outer, and Ignore.

**Normal:-** This style hatches inward starting at the area boundary, at each end of the hatch line. In this alternate areas of the selected object are hatched starting with the outermost area.

**Outer:-** This particular option also lets you hatch inward from an area boundary but the hatching is turned off if an internal intersection is encountered. The hatching process in this case starts from both ends of each hatch line, so only the outermost level of the structure is hatched, hence the name Outer.

**Ignore:-** In this option all areas bounded by the outermost boundary are hatched, ignoring any hatch boundaries that are within the outer boundary.

## **BLOCKS**

### *Draw---Blocks*

The ability to store parts of a drawing, or the entire drawing, so that they need not be redrawn when needed again in the same drawing or another drawing, is extremely beneficial to the user. These parts of a drawing, entire drawings, or symbols (also known as blocks) can be placed (inserted) in a drawing at the location of your choice, in the desired orientation, with the desired scale factor.

The first step in creating blocks is to draw the object(s) to be converted into block. You can consider any symbol, shape, or view, that may be used more than once for conversion into a block. Even a drawing that is to be used more than once can be converted into a block.

## Converting the Objects into a Block (BLOCK command)

### *Draw---Block*

You can use **BLOCK** command to save any object as a block.

#### **Command: BLOCK**

Block name(or ?): *specify the block name.*

Insertion Base point: *Pick a point.*

Select objects: *Select all the objects comprising the block.*

Once you have selected the objects defining the block, AutoCAD acknowledges creation of the block by removing the objects defining the block from the screen. You can get back the objects at the same position by entering OOPS.

Within the BLOCK command, you can list all the blocks defined in that drawing.

Command: BLOCK

Block name (or ?): ?

Block(s) to list<\*>:

The lists of blocks of that particular drawing will be seen on your screen.

### **Inserting Blocks**

Insertion of a pre-defined block is possible with the **INSERT** command.

#### **Command: INSERT**

Block name (or ?): *Enter the name of the block to be inserted.*

Insertion Point: *Pick the point where you want the insertion base point for the blocks*

X scale factor<1>/Corner/XYZ: *Pick a point, enter a number, or press the Enter key*

Y scale factor (default = X): *Pick a point, enter a number, or press the Enter key*

Rotation angle<0>: *Pick a point or enter an angle.*

These scale factors allow you to stretch or compress a block along the X and Y axes according to the needs. The scaling of the blocks becomes easy if the exact dimensions of the drawing is known.



## Inserting Blocks Using the Insert Dialogue Box

Another method of inserting blocks is by specifying the different parameters of the block to be inserted in the Insert dialogue box., which can be invoked from the Insert Block flyout in DRAW toolbar.

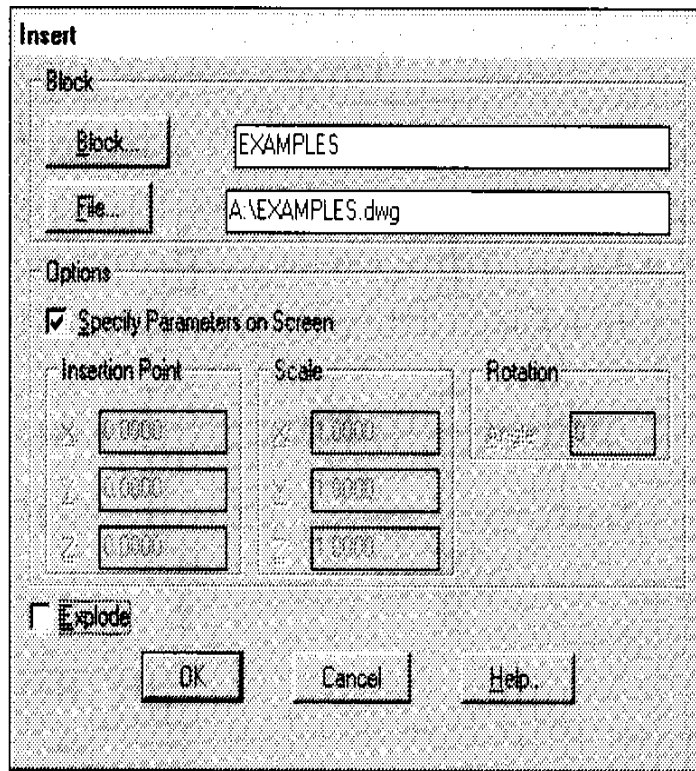


Fig. Courtesy From Autodesk Release-14 Manual

### Block... edit box:-

This edit box is used to specify the name of the block to be inserted. Pick the block.. button to generate the Defined Blocks dialogue box. Pick the desired block to be inserted.

### File... edit box:-

If you want to insert a drawing file as a block, enter the filename in the file...edit box. Pick the File...button to invoke the Select Drawing File dialogue box. Pick the drawing file from the listed drawing files existing in the current directory. Once a file has been selected, the drawing file name is displayed in the File...edit box. This name is also displayed in the Block...edit box so that you can create a block with a different name based on the selected drawing file.

## **Inserting Multiple Blocks (MINSERT command)**

### ***Insert--- Insert Multiple Blocks***

The **MINSERT** command is used for multiple insertion of a block. This command comprises features of the **INSERT** and **ARRAY** commands.

#### **Command: MINSERT**

Block name (or ?): *Enter block name.*

Insertion point: *Specify the insertion point.*

X scale factor<1>/Corner/XYZ:

Y scale factor<default = X>:

Rotation angle<0>:

Number of rows(---)<1>:

Number of columns(|||)<1>:

Unit cell distance between rows(---):

Unit cell distance between columns(|||):

In the array of blocks generated with the use of **MINSERT** command you cannot alter the number of rows/columns or the spacing between them. The whole **MINSERT** pattern is considered one object that cannot be exploded.

## **Creating Write Blocks (WBLOCKS command)**

This command is used to create symbols that can be inserted in any drawing. The blocks or symbols created by the **BLOCK** command can only be used in the drawing in which they were created. With the **WBLOCK** command you can formulate a drawing file of a specified blocks or selected objects in the current drawing, or the entire drawing. The **WBLOCK** command can be accessed, from the pull-down menu (select File, Export...), from the screen menu (select File, Export, Wblock, or by entering **WBLOCK**, at the command prompt.

## Changing Properties Of An Object

Use **CHPROP** (change properties) command to change the properties of an object.

Command: **CHPROP**

Select objects: *Pick the object whose property you want to change*

Select object:

Change what property (Color/LAyer/Ltype/lScale/Thickness)? *Specify property to be changed.*

**Color:-** By picking this button, Select Color dialogue box is displayed on the screen. Select the new layer to assign to the selected object from the list of layers in this dialogue box.

**Layer:-** Picking this button displays the Select Layer dialogue box on the screen. Select new layer to assign to the selected object from the list of layers in this dialogue box and pick the OK button.

**Linetype:-** Picking the linetype button displays the Select Linetype dialogue box on the screen. Select the new linetype to assign to the selected object from the list of linetype in this dialogue box.

**Linetype Scale:-** Enter the new linetype scale factor of the selected object in the Linetype Scale...edit box.

**Thickness:-** Enter the new thickness of the selected object in the Thickness...edit box. Once you have made the changes, pick the OK button to change the selected objects.

### **Polyline Editing (Using The PEDIT command)**

Toolbar---Modify---Edit Polyline

A polyline can assume various characteristics such as width, linetype, joined polyline, closed polyline, etc. You can edit polylines, polygons, or rectangles to attain the desired characteristics using the **PEDIT** command.

**Command: Pedit**

Select polyline: *Pick the polyline to be edited using any selection method*

If the selected line is an arc or line (not a polyline), AutoCAD issues the following prompts:

Object selected is not a polyline.

Do you want it to turn into one? <Y>: Y

The next prompt displayed is:

Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo/eXit <X>:

**Close:-** This option is used to close an open polyline.

**Open:-** If the selected polyline is closed, the Close option is replaced by the Open option. The Open option will work on a polyline in which the closing segment was created by the Close option.

**Join:-** This option appends lines, polylines or arcs that touch a selected polyline at any of its end points and adds them to it. This option can be used only if polyline is open.

**Width:-** This option allows you to define unvarying width for all segments of a polyline. It changes the width of a polyline with a constant or varying width. The following prompt sequence will change the width of the given figure,

**Command: Pedit**

Select polyline: *Pick the polyline*

Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo/eXit <X>: W

Enter new width for all segments: *Enter a value*

**Edit Vertex:-** This option lets you select a vertex of a polyline and perform different editing operations on the vertex and the segments following it. A polyline segment has two vertices. The first one is at the start point of the polyline segment, the other one is at end point of the segment.

**Fit:-** The Fit or Fit curve option generates a curve that passes through all the corners of the polyline, using the tangent directions of the vertices. The curve is composed of series of arcs passing through the corners of the polyline. This option is when you draw a polyline with sharp corners and need to convert it into a series of smooth curves.

**Spline:-** The Spline option also smooths the corners of a straight segment polyline as does the Fit option, but the curve passes only through the first and the last control points except in the case of a closed polyline.

**Decurve:-** The Decurve option straightens the curves generated after using the Fit or Spline options on a polyline to their original shape. Polyline segments are straightened using the Decurve option.

**Ltype gen:-** Use this option to control the linetype pattern generation for linetypes other than Continuous with respect to the vertices of the polyline. This option has two modes: ON and OFF. If turned on, this option generates the linetype in a continuous pattern with respect to the vertices of the polyline. If Turned to off, it generates the linetype with a dash at each vertex.

**Undo:-** The Undo option negate the effect of the most recent PEDIT operation.

**eXit:-** The eXit option helps you come out of the PEDIT command and return to the command: prompt.

## INQUIRY COMMANDS

When drawing in an AutoCAD, we will need to make inquiries about data pertaining to our drawing. The inquiries can be about the distance from one location on the drawing to another, the area of an object like a polygon or circle, coordinates of a location on the drawing. AutoCAD keeps track of all the details pertaining to a drawing. The following is the list of Inquiry commands:

**AREA          DIST          ID          LIST          MASSPROP**

### Calculating the Area of an object ( AREA command )

#### *Tools----Inquiry---Area*

In AutoCAD the AREA command is used to automatically calculate the area of an object in square units. Default option of area command can be used to calculate the area and perimeter/circumference of the space enclosed by the sequence of specified points. The prompt sequence in this case is:

#### **Command: AREA**

<First point>/Object/Add/Subtract: *Pick the first point*

Next point: *Pick the second point*

Next point: *Continue picking until all the points enclosing the area have been picked and all the objects selected.*

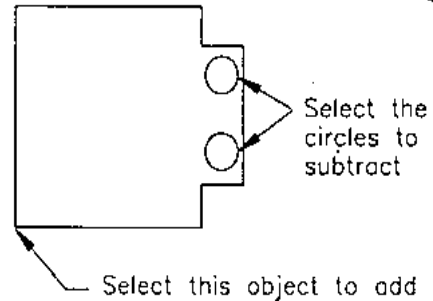
If you pick two points on the screen, the AREA command will display the value of the area as 0.00, while the perimeter value is the distance between the two points.

**Object option:-** Use the Object option of the area command to find the area of objects such as polygon, circles, closed polylines, regions, solids, and splines.

**Add option:-** Sometimes you want to add area of different objects to determine a total area. For exp., in a plan of a house, you need to add areas of all the rooms to get the total floor area. In such cases use Add option.

**Subtract Option:-** The action of the subtract option is the reverse of the Add option.

Command: Area  
<First point>/Object/Add/Subtract: A  
<First point>/Object/Add/Subtract: O  
(Add mode) Select objects: *Pick the polyline*  
Area = -----, Perimeter = -----  
Total Area = -----  
(Add mode) Select objects: *Enter*  
<First point>/Object/Add/Subtract: S



<First point>/Object/Add: O  
(Subtract mode) Select object: *Pick the circle*  
Area = -----, Circumference = -----

Fig. Boolean Operations

Total Area =-----  
(Subtract mode) Select object: *Pick the second circle*  
Area = -----, Circumference = -----  
Total Area =-----  
(Subtract mode) Select object: *Enter*  
<First point>/Object/Add: *Enter*

### Calculating the Distance Between Two Points (DIST command)

The **DIST** command is used to measure the distance between two selected points. The angles that the selected points make with the X axis and the X-Y plane are also displayed. The measurements are displayed in current units.

#### Command: **DIST**

First point: *Pick a point*  
Second point: *Pick a point*

Distance = *Calculated distance between the two points*  
Angle in XY plane = *Angle between the two points in XY plane*  
Angle from XY plane = *ANGLE THE SPECIFIED POINTS MAKE WITH xy PLANE*  
Delta X = Change in X, Delta Y = Change in Y, Delta Z = Change in Z

## Identifying a Position on the Screen (ID Command)

### *Tools---Inquiry---List---Locate Point*

This command identifies the position you specify and tells you its coordinates.

#### **Command: ID**

Point: *Pick the point to be identified*

X = <X coordinates> Y = <Y coordinates> Z = <Z coordinates>

## Listing Information about Objects (LIST command)

The list command displays all the data pertaining to the selected objects.

#### **Command: List**

Select objects: *select objects whose database you want to list*

Select objects:

The AutoCAD shifts from the graphics screen to the AutoCAD Text Window. The information displayed varies from object to object. For exp., for a line the following information os be displayed:

1. The coordinates of the end points of the line.
2. Its length (in 3D).
3. The angle made by the line with respect to the X axis of the current UCS.
4. The angle made by the line with respect to the XY plane of the current UCS.
5. Delta X, Delta Y, Delta Z:
6. Line weight.
7. The name of any dimension style; if it is unnamed, AutoCAD lists the dimension style as UNNAMED.

## Listing Information about Solids and Regions

### *Tools---Inquiry---List---Mass Properties*

With the **MASSPROP** command, you can determine volumetric information such as principle axes, center of gravity, and moment of inertia of 2D or 3D objects.

#### **Command: MASSPROP**

Select objects: *Pick an object (solid object)*

The explanation of various terms displayed on the screen as a result of invoking MASSPROP command is as follows:

#### **For Solids**

**Mass:-** This property provides the measure of mass of a solid.

**Volume:-** This tells you the measure of the space occupied by the solid.

**Bounding Box:-** If the solid were to be enclosed in a 3D box, the coordinates of the diagonally opposite corners are provided by this property.

**Centroid:-** This provides the coordinates of the center of mass for the selected solid.

**Moment of Inertia:-**This property provides the mass moment of inertia as a solid about the three axes. The equation used to calculate this value is:

$$\text{Mass\_moments\_of\_inertia} = \text{object\_mass} * (\text{radius of axes})^2$$

**Products of Inertia:-** The value obtained with this property helps to determine the forces resulting in the motion of the object. The equation used to calculate this value is:

$$\text{Product\_of\_inertia YX,XZ} = \text{mass} * \text{dist centroid\_to\_YZ} * \text{dist centroid\_to\_XZ}$$

**Radii of Gyration:-** The equation used to calculate this value is:

$$\text{Gyration\_radii} = (\text{moments\_of\_inertia/body\_mass})^{1/2}$$

**Principal Moments and X-Y-Z Direction about Centroid:-** This property provides you with the highest, lowest, and middle value for the moment of inertia about an axis passing through the centroid of the object.



## **Orthographic Projections**

An engineer always describe the shape of an object by drawing it on two dimensional object i.e. on paper. The graphical representation is known as Engineering Drawing.

Projection is the image of the object formed by rays of lines, by taking in particular direction, onto the picture plane, as it appears to the observer standing at that point.

If we take infinite distance ,at which observer is standing, from the object such that the projections are parallel to each other but perpendicular to the picture plane, the projection obtained is called the orthographic projection.

There are two systems of projections

1. First Angle Projection
2. Third Angle Projection

Indian Standards Institution recommended, in the edition of IS:696-1972 to use first angle projections after 1991.

### **First Angle Projection**

In this, the top view comes below the front view, the left view is placed to the right side of the front view and right side view will be placed on the left side of the front view.

### **Third Angle Projections**

In this, The top view comes above the front view, the left side view comes left of the front view and right side view comes right of the front view.

### **Model space & Paper space**

You can work with a drawing in two separate spaces, Model space and Paper space. Paper space and model space let you draw at a 1:1 scale and apply scaling factor to views before you plot. By default, the space that you draw in is model space, the drawing editor always begins in model space.

If you are creating drawings with multiple views and different scales in those views, paper space is essential.

Paper Space is used for plotting different viewports on single sheet.

When Paper Space is used first time, the graphic area displays as a blank space. Floating View ports are created to get different views of the model. Floating viewports are treated as an object that can be moved and resized for creating suitable layout. Title blocks can be inserted without affecting the model space.

There is a restriction in paper space, nothing can be edited in paper space. For editing one has to move to model space.

## **VIEWPORTS**

A viewport is a rectangular area that shows a view of a model.

### **Tiled And Floating Viewports**

Tiled Viewports appears on your screen like tiles; they completely fill the graphics area and cannot overlap. Tiled viewports can only be plotted one at a time.

Floating viewports are like rectangular boxes that you can place and move where you like on your screen. They can overlap one another and be plotted at the same time.

The type of viewport you use is set by the TILEMODE system variable. If you set TILEMODE to 1 (on), the viewports are tiled. If you set TILEMODE to 0 (off), the viewports are floating.

### **The MVSETUP command**

The **MVSETUP** command consists of several routines designed to automate the processing of working in paper space. It also lets you define the number of viewports, their positions, and even sets up default positions. MVSETUP also inserts predefined title blocks of different sizes or lets you insert your own in the drawing.

Command: MVSETUP

Enable paper space? (No/<Yes>): n

Units type (Scientific/Decimal/Engineering/Architectural/Metric): d

Scientific scales

(4.0) 4 TIMES

(2.0) 2 TIMES

(1.0) FULL

(0.5) HALF

(0.25) QUARTER

Enter scale factor =1

Enter paper width = any paper width from A0 to A4

Enter paper height = According to sheet size.

## **3D Coordinate System**

1. Cylindrical Coordinate system
2. Spherical Coordinate System

### **Cylindrical Coordinate System**

In this system the location of any point is described by three things:

1. Distance from the present UCS origin.
2. Angle in the XY plane.
3. Z dimension.

This coordinate system is mostly used for location of points on a **CYLINDRICAL** shape. For exp. A point 5 units from the UCS origin, at an angle of 60 degrees from the X axis, and having a Z dimension of 6 units would be presented in the following format:

5<60,6.0

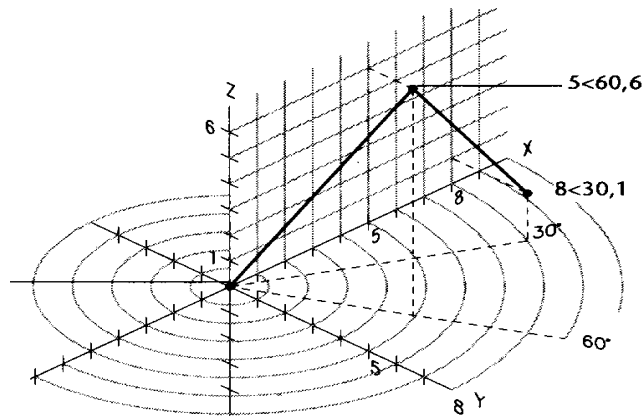


Fig. Courtesy From Autodesk Release-14 Manual

## Spherical Coordinate System

In the spherical coordinate system, the location of any point is described by three things:

1. Distance from the present UCS origin.
2. Angle in the XY plane.
3. Angle from the XY plane.

This coordinate system is mostly used in locating points on a spherical surface. For exp. A point 4 units from the UCS origin, at an angle of 45 degrees from the X axis, and at an angle of 5 degrees from XY plane would be represented as:

4<45<5

## USER COORDINATE SYSTEM

### UCSICON Command

#### *Tools---UCS*

The orientation of the origin of the current UCS is represented by the UCS icon. This icon is a graphic reminder of the direction of the UCS axes and the location of the origin. This icon also tells you what the viewing direction is, relative to the UCS XY plane. The + sign on the icon reflects that the UCS icon is placed at the origin of the current UCS. Absence of the W on the Y axes indicates that the current coordinate system is not WCS.

### The User Coordinate System (UCS)

The user coordinate system (UCS) helps you to establish your own coordinate system. The UCS is mostly used in 3D drawings where you may need to specify points that vary from each other along the X, Y, and Z axes.

#### **Command: UCS**

Origin/Zaxis/3point/Object/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>:

**W (World):** Using this option you can set the current UCS to be identical to the world coordinate system.

**O (Origin):** With this option you can define a new UCS by changing the origin of the current UCS. The direction of the X,Y, and Z axes remain unaltered.

**ZA (ZAxis):** With this option of the UCS command, you can change the coordinate system by selecting the origin point of the coordinate system and a point on the positive Z axis. After specifying a point on the Z axis, AutoCAD determines the X and Y axes of the new coordinate system accordingly.

**3 (3point):** In this option you can establish a new coordinate system by specifying the new origin point, a point on the positive X axis, and a point on the positive Y axes. The direction of Z axes is determined by right hand rule.

**OB (Object):** With the OB (Object) option of the UCS command, you can establish a new coordinate system by pointing to any object in an AutoCAD drawing except a 3D polyline, 3D solid, 3D mesh, Viewport object, mtext, xline, ray, leader, ellipse, region, or spline.

**V (View):** The V (View) option of the UCS command lets you define a new UCS whose XY plane is perpendicular to your current viewing direction. In other words, the new XY plane is parallel to the screen. The origin of the UCS defined in this option remain unaltered. This option is mostly used to view drawing from a oblique viewing direction.

**X/Y/Z:** With this option you can rotate the current UCS around a desired axis.

**P (Previous):** The P (Previous) option restore the previous UCS.

**R (Restore):** With this option of the UCS command you can restore a previously saved UCS. Once a saved UCS is restored, it become4s the current UCS. The viewing direction of the saved UCS is not restored.

**S (Save):** With this option you can name and save the current UCS.

**D (Delete):** The D (Delete) option is used to delete the selected UCS from the list of saved coordinate systems.

**? option:** By invoking this option, you can list the name of the specified UCS. This option gives you the name, origin,, and XYZ axes of all the coordinate systems relative to the existing UCS.

## DRAWING PRE-DEFINED 3D OBJECTS

### *Draw---Surfaces*

AutoCAD supports a library of 3D objects. To draw these pre-defined objects, you simply specify the values of various parameters pertaining to the selected 3D object. Once you have done entering the information, the object is automatically drawn on the screen. In this manner you can create a number of 3D objects with very little effort. You can create surfaced wireframe and solid primitive types of predefined 3D shapes as shown in **fig.**

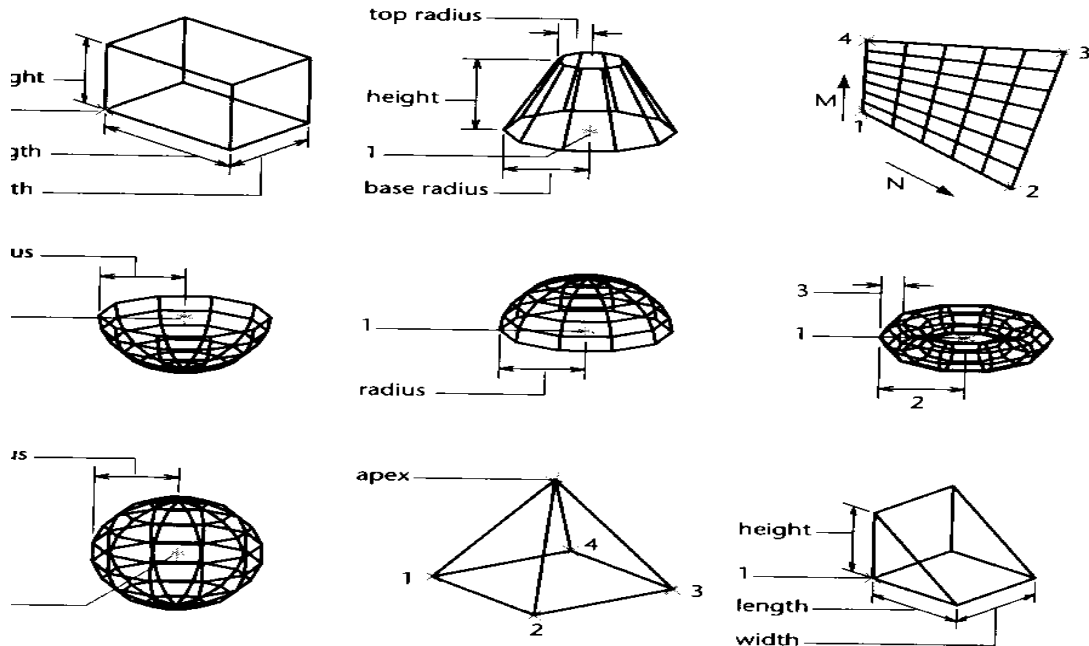


Fig. Courtesy From Autodesk Release-14 Manual

### Viewing Objects in 3D Space

AutoCAD has provided the VPOINT command to view the objects in 3D space.

#### **View---3D Viewpoint---Vector**

Once you set a viewpoint, all the objects in the current viewport are regenerated and the objects are displayed as if you were viewing them from the newly defined viewpoint.

#### **Command: VPOINT**

Rotate/<View point> <0,0,1>:

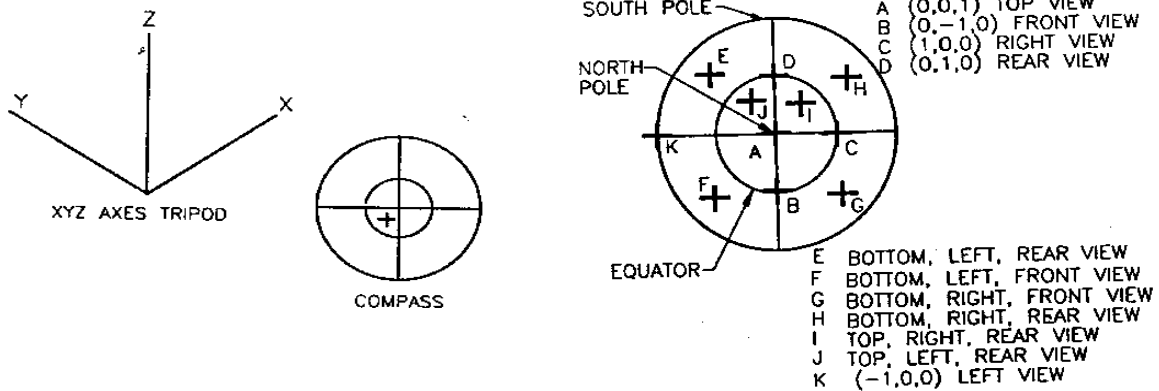
The prevailing viewpoint is the default. At the default viewpoint (0,0,1), you are looking at the drawing from a point on the positive Z axis at a height of 1 unit from the origin. The following are some of the viewpoint settings and the sort of view obtained.

Vpoint Value	View Displayed
0,0,1	Top View
0,0,-1	Bottom View
0,-1,0	Front View
0,1,0	Rear View
1,0,0	Right Side View
-1,0,0	Left Side View
1,-1,-1	Bottom,Front,Rightside view
-1,-1,-1	Bottom,Front,Left side view
1,1,-1	Bottom,Rear,Right side view
-1,1,-1	Bottom,Rear,Left side view
1,-1,1	Top,Front,Right side view
-1,-1,1	Top,Front,Left side view
1,1,1	Top,Rear,Right side view
-1,1,1	Top,Rear,Left side view

## Using Axes Tripod and Compass to Set Viewpoint

### View---3D Viewpoint---Tripod

You can specify the viewpoint by moving the pointing device as shown in fig. As you move the pointing device, the XYZ axes tripod also moves and so does the small cross-hair in the compass. The center point of the circle is the north pole (0,0,1) and by placing the cross-hairs on the north pole you can see the top view of the object. The outer circle represents the south pole (0,0,-1). By picking cross-hair location on the screen in this manner, you can get any viewpoint as shown in fig.



## CREATING SURFACES

### *Draw---Surfaces---Ruled Surface*

The **RULESURF** command can be used to generate a polygon mesh between two objects such as lines, arcs, circles, 2D polylines, 3D polylines, splines, ellipses, or elliptical arcs. The two specified objects acts as the edge of the rule's surface. The only condition for this command is that if one of the two objects is open, the other object should also be open, or if one of the object is closed, the other object should also be closed.

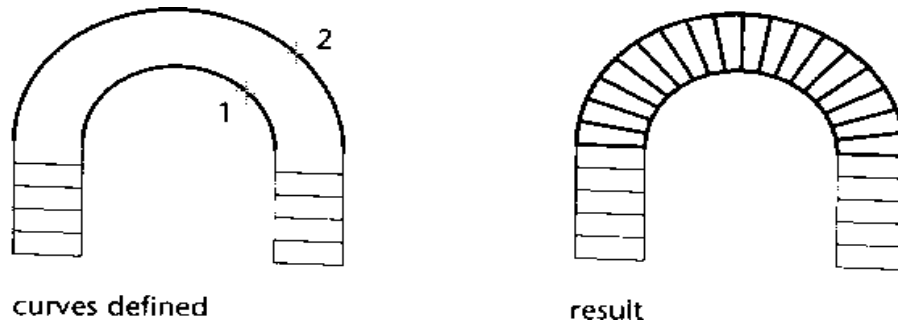


Fig. Courtesy From Autodesk Release-14 Manual

### **Command: Rulesurf**

Select first defining curve: Select 1

Select second defining curve: Select 2

This command is frequently used to generate flat surfaces of different shapes, surfacing holes, curves on the exterior of an object, curves on the interior of an object, etc.

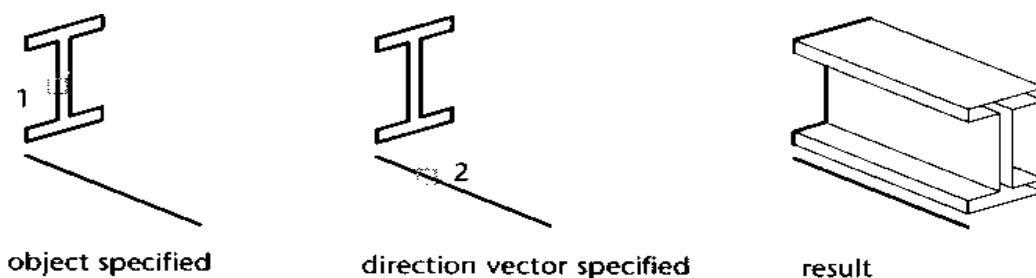
### **TABSURF Command**

The **TABSURF** command can be used to generate a surface extrusion from an object. Only one object is needed in the TABSURF. This object is also known as path curve. The path curve can be a line, arc, elliptical arc, circle, 2D polyline, or 3D polyline. The direction and length of extrusion are governed by length and direction of a second line or a polyline, known as the direction vector.

### **Command: Tabsurf**

Select path curve: *Pick a point on the path curve*

Select direction vector: *Pick a point on the direction vector.*



## REVSURF Command

With the **REVSURF** command , you can create a 3D mesh that trails along the path curve (profile) and rotate about the specified axis. The path curve or the profile can be an arc, line, 2D polyline, ellipse, elliptical arc, or circle. The axis of revolution can be an open 2D or 3D polyline or a line

### Command: *Revsurf*

Select path curve: *Pick a point on the path curve.*

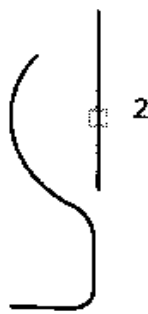
Select axis of revolution: *Pick a point on the axis of revolution line.*

Start angle <0>: *Specify the start angle*

Included angle (+ = ccw, - = cw) <full circle>: *specify the included angle.*



profile specified



axis of revolution specified



result

Fig. Courtesy From Autodesk Release-14 Manual

## EDGESURF Command

With the **EDGESURF** command you can create a 3D mesh with four connected sides as the edges (boundaries). This mesh is a matrix of 3D faces. The only condition to create a mesh with the EDGESURF command is that there should be four points. The edges can be selected in any order.

### Command: *Edgesurf*

Select edge 1: *Pick a point on the first of the boundary*

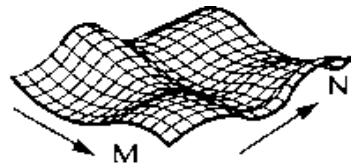
Select edge 2: *Pick a point on the second edge of the boundary*

Select edge 3: *Pick a point on the third edge of the boundary*

Select edge 4: *Pick a point on the fourth edge of the boundary.*



four edges selected



result



### 3DMESH Command

The **3DMESH** command lets you create a 3D polyline mesh. The mesh is defined in terms of rows (M) and columns (N). The mesh is open in both M and N directions. Use the Close option of the PEDIT command to close the mesh. The total number of vertices in a mesh is the product of M and N. Depending on the coordinate value of the vertices, the vertices can be at any distance from each other.

#### Command: 3DMESH

Mesh Size: 6

Mesh Size: 6

Vertex (0,0): Give Vertex Coordinate (0,0,0)

Vertex (0,1): Give Vertex Coordinate (0,1,0)

Vertex (0,2): Give Vertex Coordinate (0,2,0)

Vertex (0,3): Give Vertex Coordinate (0,3,0)

Vertex (0,4): Give Vertex Coordinate (0,4,0)

Vertex (0,5): Give Vertex Coordinate (0,5,0)

Vertex (1,0): Give Vertex Coordinate (1,0,0)

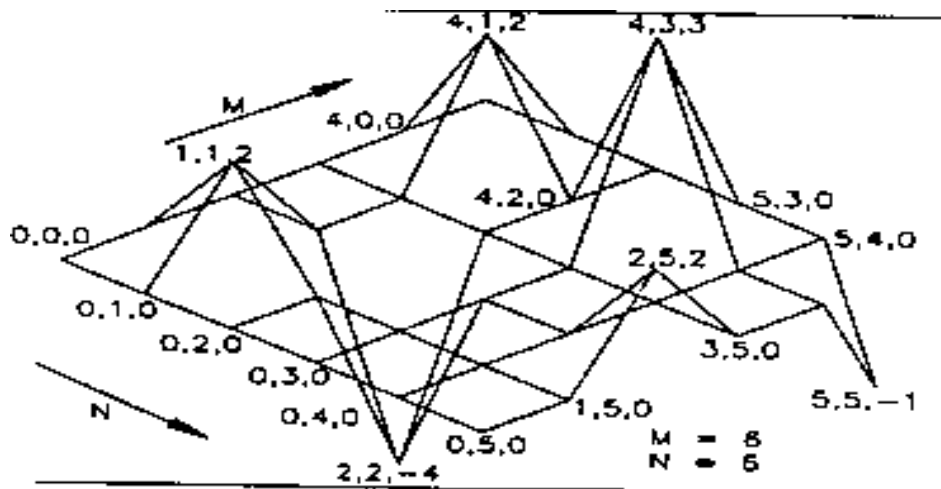
Vertex (1,1): Give Vertex Coordinate (1,1,2)

Vertex (1,2): Give Vertex Coordinate (1,2,0)

Vertex (1,3): Give Vertex Coordinate (1,3,0)

Vertex (1,4): Give Vertex Coordinate (1,4,0)

Vertex (1,5): Give Vertex Coordinate (1,5,0)



## 3DFACE Command

### *Draw---Surfaces---3Dface*

3D models often need solid surfaces for shading, hiding, etc. Three- or four-sided surfaces are created with the help of the **3DFACE** command. With 3DFACE command the surfaces can be created anywhere in the 3D space. In this command you can specify different Z coordinates for the different corners of a face, thus forming a plane in the 3D space.

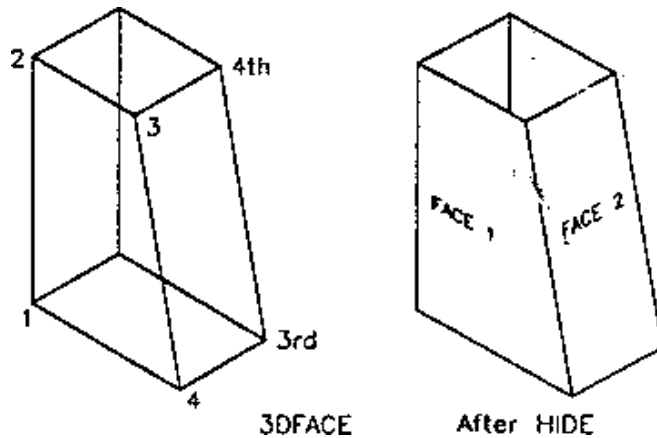
#### **Command: 3DFACE**

First point: *Select first point of the first face.*

Second point: *Select second point of the first face.*

Third point: *Select third point of the first face.*

Fourth point: *Select fourth point of the first face.*



Once you specify the 1<sup>st</sup>, 2<sup>nd</sup>, 3<sup>rd</sup>, & 4<sup>th</sup> points, AutoCAD closes the face from the fourth point to the first point.

If you want to make some edges invisible, use **INVISIBLE** option of the **3DFACE** command. Select the invisible option from the screen menu or enter I before picking the first point of the invisible edge. All the edges of the 3D solid can be made invisible.

## **DRAWING 3D POLYLINES (3D POLY Command)**

### *Draw---Polyline---3Dpolyline*

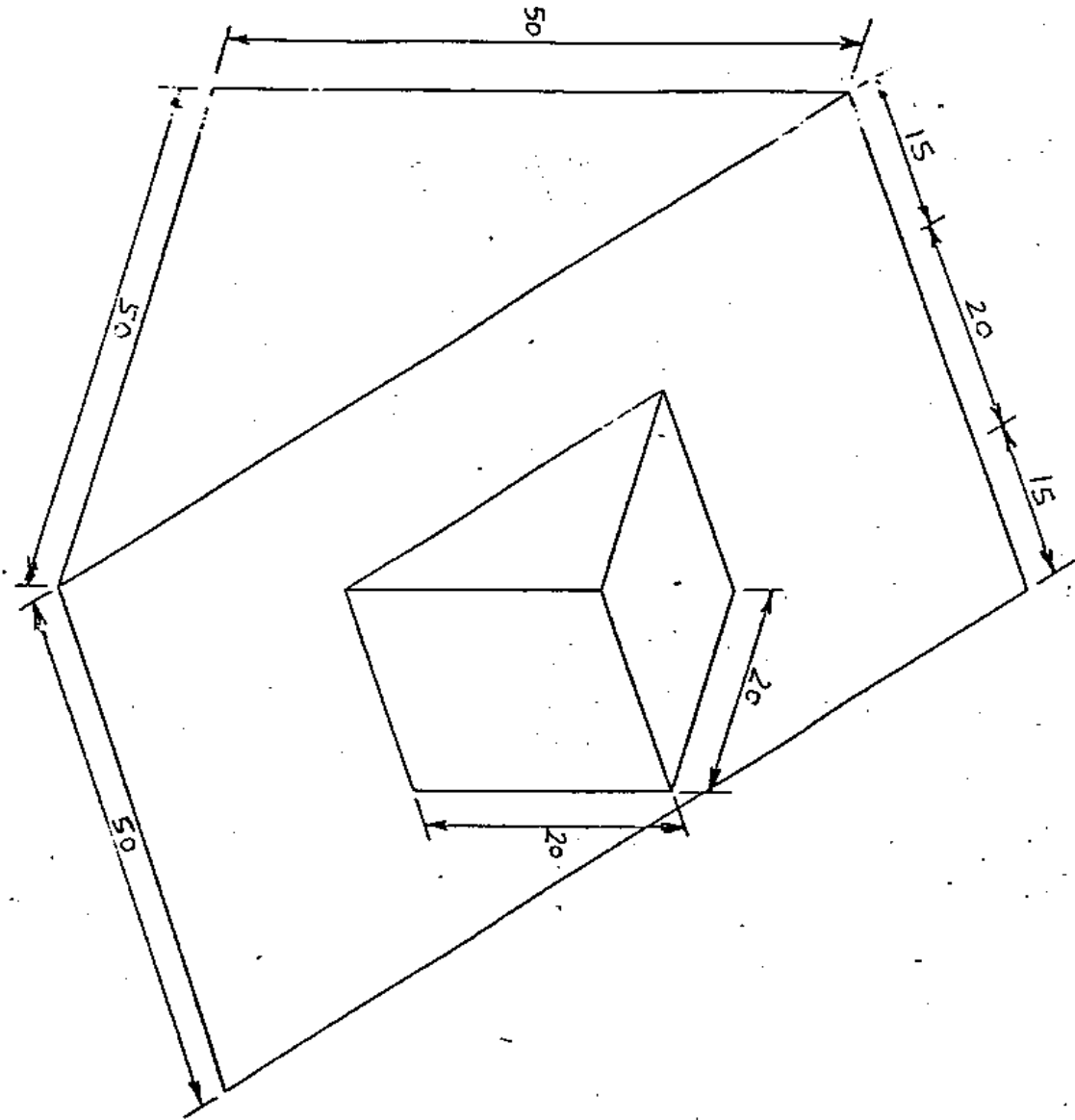
The working of the **3DPOLY** and **PLINE** commands is similar. The exceptions are that a third dimension (Z) is added to the 3Dpolyline and with polyline only straight lines can be drawn with variable widths.

Command: *3Dpoly*

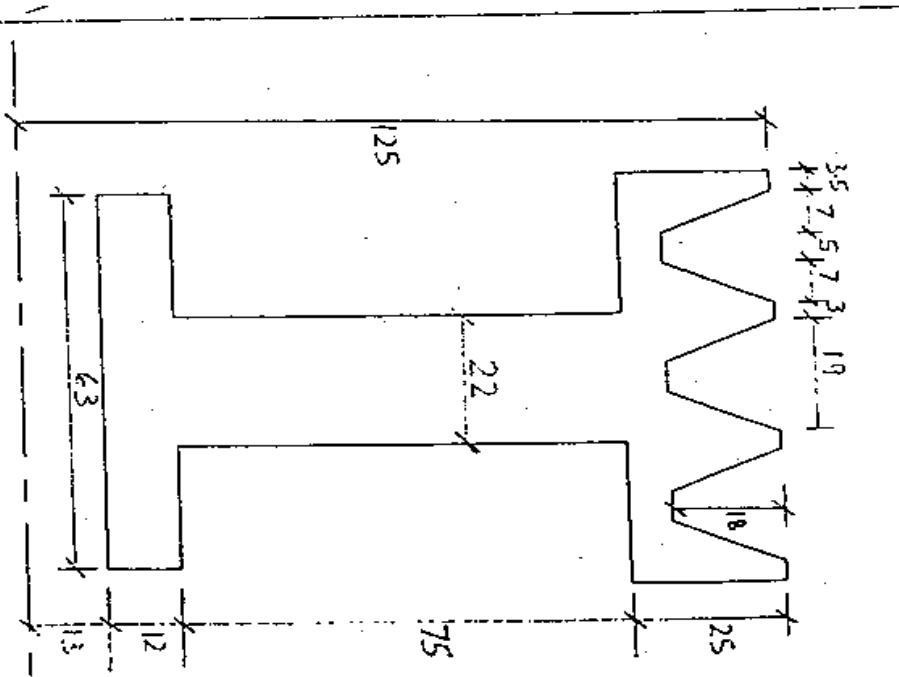
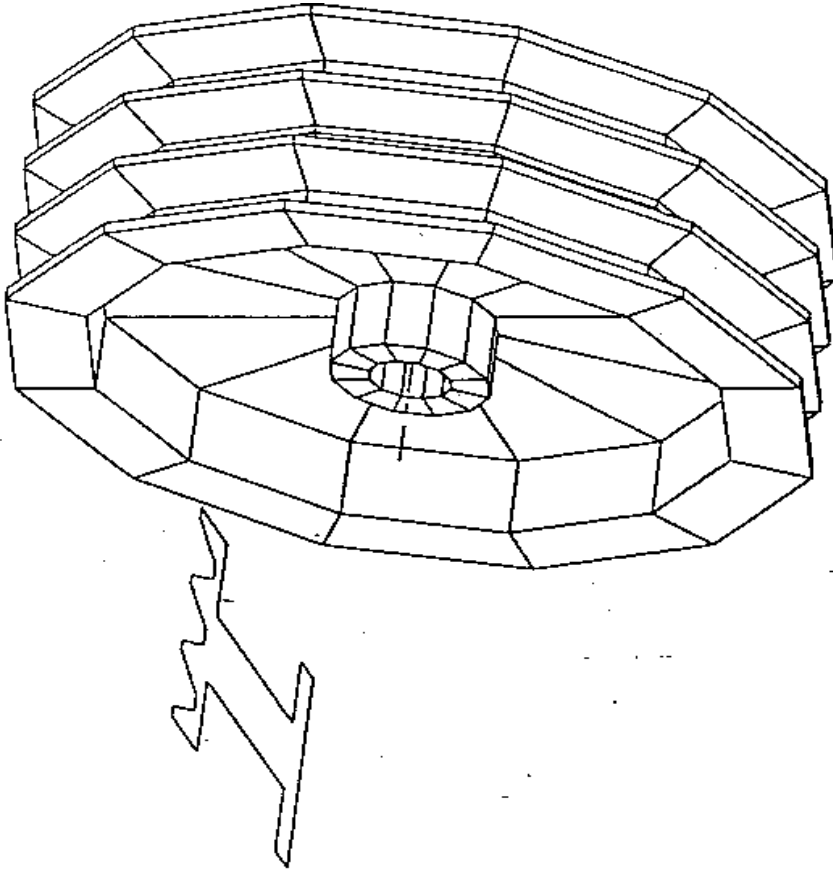
From point: *specify a point where you want to start the 3D polyline.*

Close/Undo/<Endpoint of line>: *Specify the end point on the screen.*

Exercise No. 10 Draw the following drawing With 3D FACE command.



Exercise No. 11 Draw the following drawing with Revolve command



## SOLID MODELING

Solid modeling is the process of building objects that have all the attributes of an actual solid objects. Solid models make easy to visualize faster as we think. AutoCAD's solid modeling is based on ACIS solid modeler, which is a part of core technology.

### Solid Primitives

#### *Draw---Solids*

The solid primitives form the basic building blocks for a complex solid. AME has six predefined solid primitives (box, wedge, cone, cylinder, sphere, and torus) that can be used to construct a solid model. When you enter commands for creating solid primitives, AutoCAD solids will prompt you to enter information about the part geometry. The height of the primitive is always along the positive Z axis, perpendicular to the construction plane.

### Creating a BOX

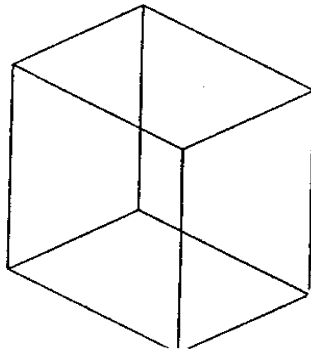
The following is the command prompt sequence for the BOX command:

**Command:** *box*

Center/<Corner of box><0,0,0>: 1,1,0

Cube/Length/<Other corner>: 2,3,0

Height: 4



### Creating a Solid Cone

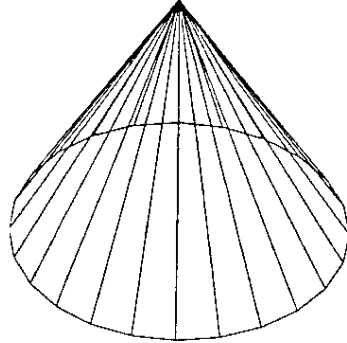
The CONE command creates a solid cone with an elliptical or circular base.

**Command: Cone**

Elliptical/<Center point> <0,0,0>: 6,2

Diameter/<Radius>: 2

Apex/<Height>: 4



**Creating a Solid Cylinder**

It has two options: Elliptical and Center point.

**Command: Cylinder**

Elliptical/<Center point> <0,0,0>: 6,2

Diameter?<Radius>: 3

Center point of other end/<Height>: 4

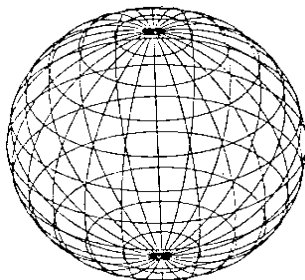
**Creating a Solid Sphere**

The command has only one option: Center of sphere. The center of the sphere automatically aligns with the Z axis of the current UCS.

**Command: Sphere**

<Center of sphere> <0,0,0>: 2,5

Diameter/<Radius> of sphere: 3



**Creating a Solid Torus**

After selecting the command, AutoCAD prompt you to enter Diameter/<Radius> of torus and Diameter/<Radius of tube>.

**Command: Torus**

<Center of torus> <0,0,0>: 6,4

Diameter/<Radius> of torus: 4

Diameter/<Radius> of tube: .5

**Creating a solid Wedge**

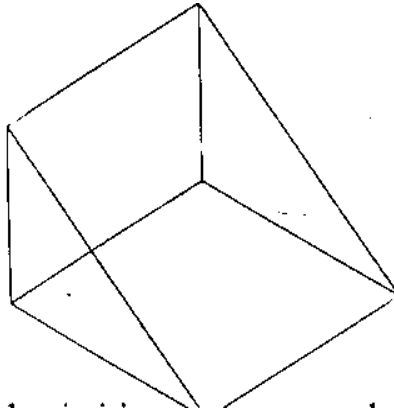
The **WEDGE** command can be selected just as any command to create any solid primitive object.

**Command: Wedge**

Center/<Corner of wedge><0,0,0>: 1,1,0

Cube/Length/<Other corner>: 4,3,0

Height: 5

**Creating A Composite Solid**

A composite solid consists of two or more solid primitives. The example given below shows you the common solid commands used.

**Creating 2D Region**

Use the **ARC** and **LINE** commands to draw the outer profile as shown in fig.

**Command: ARC**

Center/<Start point>: C

Center: 5,4

Start point: 7,5.5

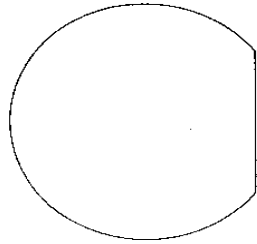
Angle/Length of cord/<End point>: 7,2.5

**Command: Line**

From point: 7,5.5

To point: 7,2.5

To point: *Enter*



Outer profile of the base plate

Use **PEDIT** command to change the line into a polyline, then use Join option.

### Creating a 2D Region

Use the **REGION** command to create a region.

Use **BHATCH** command to hatch the region.

Draw 0.6 diameter circles located on the 4.0 diameter bolt circle then use **ARRAY** command to create the remaining circles

#### Command: *Circle*

3P/2P/TTR/<Center point>: @2<60

Diameter/<Radius>:D

Diameter: 0.6

#### Command: *Array*

Select objects: *select 0.6 diameter circle*

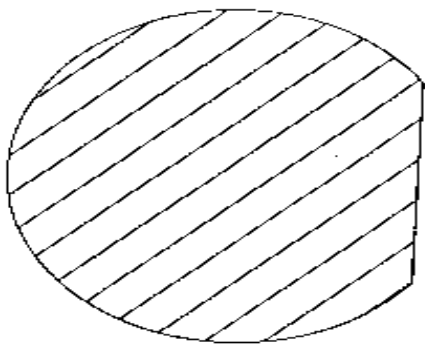
Rectangular or Polar array (R/P): P

Center point of array: 0,0

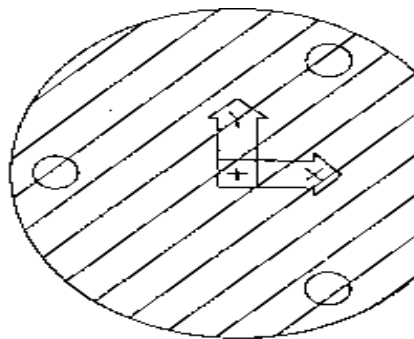
Number of items: 3

Angle to fill (+ =CCW, - =CW) <360>: 240

Rotate objects as they are copied? <Y>:



Creating a 2D Region



Creating a circular array

### Subtracting Regions

Use the **SUBTRACT** command to subtract regions or solids.



**Command: *Subtract***

Source objects...

Select objects: select the region

Select objects: *Enter*

Objects to subtract from them...

Select objects: *select three 0.6 dia. Circles.*

Select objects: *Enter*

**Creating an Extruded Solid**

***Draw---Solid---Extrude***

Extrude command gives the height to an object.

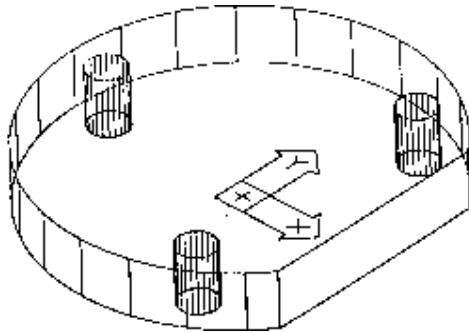
**Command: *Extrude***

Select objects: select region

Select object: *Enter*

Path/<Height of extrusion>: 0.70

Extrusion taper angle<0>: *Enter*



Use **VPOINT** command to get a 3D view of the extruded region.

**Command: *Vpoint***

Rotate/<View point><0,0,1>: 1,-1,-1

## Creating a REVOLVED Solid

### *Draw---Solid---Revolve*

The **REVOLVE** command creates a solid by sweeping a polyline or circle around an axis. The selected objects can be rotated through any angle from 0 to 360 degrees. Before using the REVOLVE command, you must draw the profile of the surface that you want to revolve. Use the UCS command to rotate the UCS icon by +90 degrees and then draw the profile

#### **Command: UCS**

Origin/Zaxis/3point/Object/View/X/Y/Z/Pre/Restore/Save/Del/?/<World>: X  
Rotation angle about X axis<0>: 90

#### **Command: PLINE**

From point: 0,0

Current line-width is 0.00

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: @0.7,0

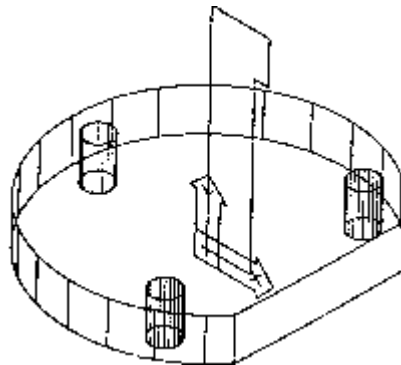
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: @0,3

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: @0.25,0

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: @0,0.70

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: @-1,0

Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: C



Use the **REVOLVE** command to revolve the profile through through 360 degrees.

#### **Command: Revolve**

Select region, polyline or circle for revolution...

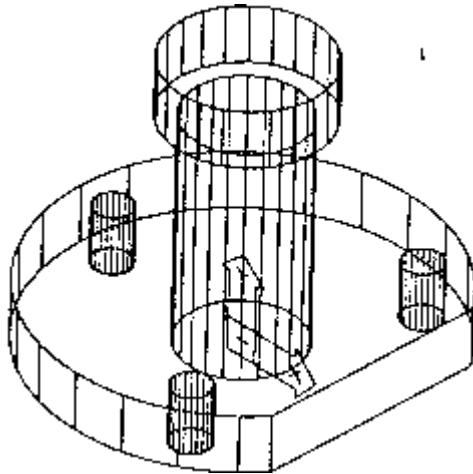
Select objects: *Select polyline*

Select object: Enter

Axis of revolution – Object/X/Y/<Start point of axis>: 0,0

End point of axis: 0,3

Angle of revolution<full Circle>: *Enter*



### Creating a cylinder

The **CYLINDER** command creates a solid cylinder. Use the following to draw the solid cylinder:

**Command:** *UCS*

Origin/Zaxis/3point/Object/View/X/Y/Z/Pre/Restore/Save/Del/?/<World>: X

Rotation angle about X axis<0>: -90

Origin/Zaxis/3point/Object/View/X/Y/Z/Pre/Restore/Save/Del/?/<World>: O

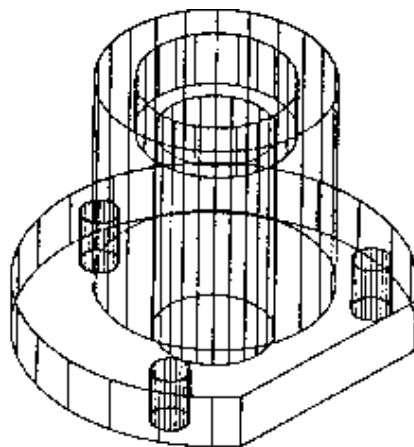
Origin point (0,0,0): @0,0,0.70

**Command:** *Cylinder*

Elliptical/<center point><0,0,0>: *Enter*

Diameter/<Radius>: 1.25

Center of other end/<Height>: 4



## USE OF UNION COMMAND

### *Modify---Bollean---Union*

The UNION command can be used to create composite solids or regions. Several solids or regions can be combined in the same command.

#### **Command: Union**

Select objects: *select baseplate*

Select objects: *Select cylinder.*

Select objects: *Enter*

#### **Command: Subtract**

Select solids and regions to subtract from...

Select objects: *select the unioned solid. 1 found.*

Select objects: *Enter*

Select solids and region to subtract from...

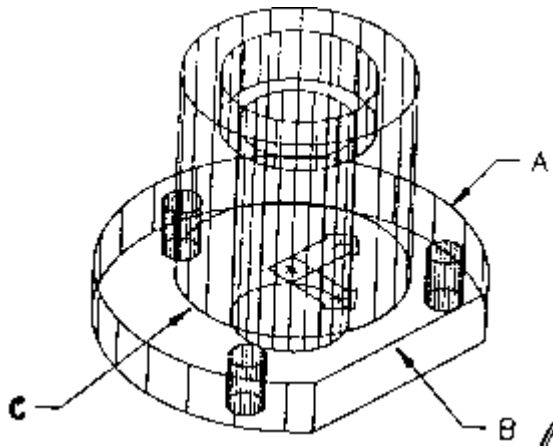
Select objects: *select the inner core. 1 found.*

Select objects: *Enter*

## Creating Fillets and Rounds

### *Modify---Chamfer---Fillet*

The FILLET command is used to create fillets and rounds. The FILLET command does not have any special fillet or round option for solids. Solids is intelligent enough to determine, depending on the geometry of the solid, whether to create a fillet or a round at the selected edges as shown in fig.



**Command: *Fillet***

(Trim mode) Current fillet radius = 0.125

Polyline/radius/trim/<Select first object>: *Select edge A*

Chain/radius/<Select edge>:

Enter radius<0.125>: 0.15

Chain/radius/<Select edge>: *Select edges B and C*

**Creating Chamfer**

The **CHAMFER** command can be used to create chamfer; that is, bevel the edges of a solid by a specified distance. The CHAMFER command requires you to select the surface you want to chamfer, select the edges where the chamfer is needed, and indicate the chamfer distance of the base and the adjacent surfaces.

**Command: *Chamfer***

Polyline/Distance/Angle/trim/Method/<Select first line>: *select surface E*

Select base line:

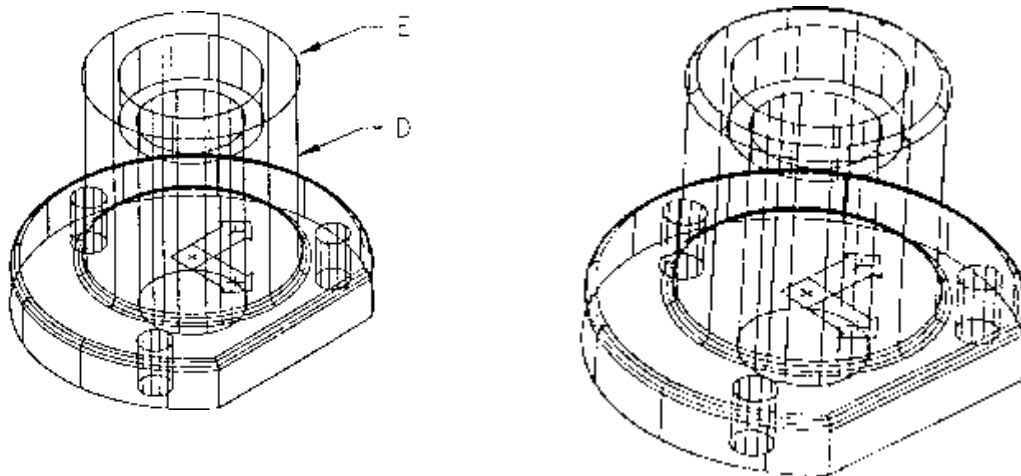
Next/OK: *enter Next if needed until the cylinder edges are highlighted.*

Enter base surface distance: 0.15

Enter other surface distance<0.1250>: 0.15

Loop/<Select edge>: *select edge E*

Loop/<select edge>: *Enter*



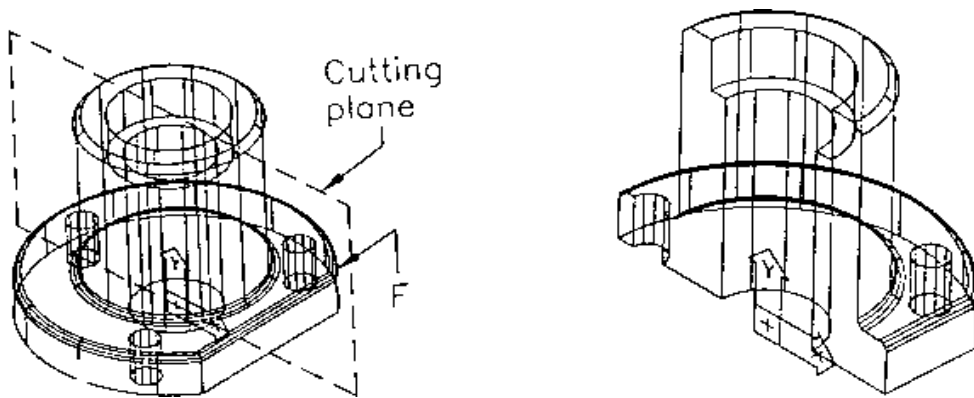
## Slicing Solids (SLICE)

### *Solids---Slice*

The SLICE command cuts a solid along the specified plane. This command cannot be used with a region. AutoCAD will ask you to select the object you want to cut and the side of the solid you want to retain. You can specify the part that you want to retain by selecting a point on that part. Use the following commands to cut the solid as shown in the fig.

#### **Command: UCS**

Origin/Zaxis/3point/Object/View/X/Y/Z/Pre/Restore/Save/Del/?/<World>: X  
Rotation angle about X axis<0>: 90



#### **Command: Slice**

Select object: select the solid

Select objects: *Enter*

0 solid created

Slicing plane by Object/Zaxis/View/XY/YZ/ZX/<3point>: XY

Point on XY plane<0,0,0>: *Enter*

Both sides/<Point on desired side of the plane>: *select point F using ENDPPOINT O'Snap.*

## Create a Cross Section (SECTION)

The SECTION command creates the cross section of a solid through the specified plane. In addition, the cross section blocks or regions are created in that current layer, not on the layer of the solid that is sectioned.

## 3D ARRAY

*Modify---3D Operation----3D Array*

### 3D Rectangular Array

This command is similar to 2D array, the only difference being the 2D and 3D. The 3D Rectangular array can be drawn in the same manner as 2D array.

### 3D Polar Array

The 3D Polar Array is also same as 3D array.

### 3D MIRROR

#### **Command: *Mirror3D***

Select objects: *select the objects to be mirrored.*

Select objects: *press enter*

Plane by Entity/Last/Zaxis/View/XY/YZ/ZX/<3Point>:

**3point:** In this option specify the mirroring plane with the help of three points. The mirroring plane is the plane that passes through the three points.

1<sup>st</sup> point on plane:

2<sup>nd</sup> point on plane:

3<sup>rd</sup> point on plane:

Delete old object?<N>: *Enter Y or N*

**Plane by Entity:** In this option select an entity and the mirroring plane is aligned with the plane of the selected entity.

Pick a circle, arc, or 2D-polyline segment: *Select the object.*

Delete old object?<N>:

**Last:** The Last option of the **MIRROR3D** command uses the last defined mirror plane to mirror objects.

**Zaxis:** This option prompts you to specify two points. The mirroring plane is defined by a point on the plane and a point on the Z axis of that plane.

Point on plane:

Point on Z-axis (normal) of the plane:

**View:** The mirroring plane passes through the specified point and is perpendicular to the view direction.

Point on view plane<0,0,0>:

**XY/YZ/ZX:** The mirroring plane is aligned with the **XY/YZ/ZX** plane and passes through the specified point.

Point on XY/YZ/ZX plane<0,0,0>:

### Command: MIRROR3D

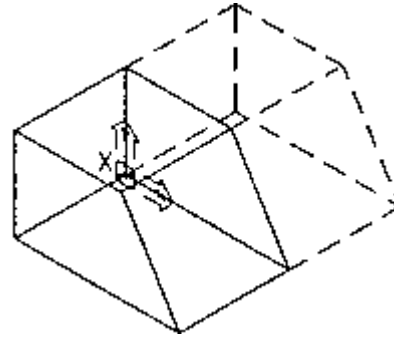
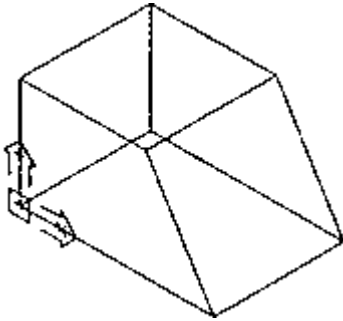
Select object: Select the Object

Select object: Press enter

Plane by Entity/Last/Zaxis/View/XY/YZ/ZX <3 point>: XY

Point on XY plane<0,0,0>: Specify X as the point on XY plane

Delete Old object:<N>: Enter



### ROTATING 3D OBJECTS

With the help of the **ROTATED3D** command, you can rotate an object in 3D space about any axis irrespective of the orientation of the current UCS.

#### Command: *Rotated3d*

Select objects: *Specify the objects you want to rotate.*

Select objects:

Axis by Entity/Last/View/Xaxis/Yaxis/Zaxis/<2points>:

**2points:** In this option specify the two points. The axis of rotation is defined by the line between the two points.

Axis by Entity/Last/View/Xaxis/Yaxis/Zaxis/<2points>:

1<sup>st</sup> point on axis: *Specify the first point.*

2<sup>nd</sup> point on axis: *Specify the second point.*

<rotation angle>/Reference: *Specify the angle of rotation.*

**Axis by Entity:** In this case objects like lines, polylines, circles, and arcs are used to define the axis of rotation. The axis of the circle/arc is perpendicular to its plane and passes through its center.

Pick a line, circle, arc or 2D-polyline segment: *Select an object.*

<rotation angle>/Reference: *Specify the angle of rotation.*

**Last:** In this option the most recently used axis is selected.

**View:** This option allows you to specify a point in the space and the viewing direction is aligned with this point.

Axis by Entity/Last/View/Xaxis/Yaxis/Zaxis/<2points>:

Point on view direction axis<0,0,0>: *Specify a point.*



<Rotation angle>/Reference: *Specify the angle of rotation.*

**Xaxis/Yaxis/Zaxis:** *With option, the axis of rotation is aligned with the X/Y/Z axis and the specified point.*

Axis by Entity/Last/View/Xaxis/Yaxis/Zaxis/<2points>: *X/Y/Z axis*

Point on view X/Y/Z axis <0,0,0>: *Specify a point.*

<Rotation angle>/Reference: *Specify the angle of rotation.*

## **INTERSECTION COMMAND**

The **INTERSECT** command creates a composite solid or region from objects that intersect. If the objects do not intersect, they do not have any common overlaying area or volume, therefore INTERSECT cannot be used with them.

## **INTERFREE COMMAND**

The **INTERFREE** command can be used to find interference of two or more solids and creates a composite solid from the common volume shared by the objects. If the objects don't intersect, they cannot be Interfree.

## **SOLVIEW Command**

Solview command helps to lay out paper space viewports of the model. Once the viewports are created, SOLDRAW command is used to create orthographic projections of the model. Follow these sequence :

Command: SOLVIEW

Ucs/Ortho/Auxiliary/Section/<eXit>: U

Named/World/?/<Current>: Enter

Enter view scale<1.00>: Enter

View Center: Pick lower left corner of the screen.

View center: Enter

Clip first corner: Pick point above the first view and to the left of the first view

Clip other corner: Pick point below the first view and to the right of the first view.

View name: give any name

Ucs/Ortho/Auxiliary/Section/<eXit>: X

Now draw the other orthographic projections from the above drawn view as:

Command: SOLVIEW

Ucs/Ortho/Auxiliary/Section/<eXit>: O

Pick side of viewport to project: Select top edge of the drawn viewport

View Center: Select above of the first view

View center: Enter

Clip first corner: Pick point above the first view and to the left of the first view

Clip other corner: Pick point below the first view and to the right of the first view.

View name: give any name

Ucs/Ortho/Auxiliary/Section/<eXit>: O

Pick side of viewport to project: Select side edge (right) of the first drawn viewport

View Center: Select side (right) of the first view

View center: Enter

Clip first corner: Pick point above the first view and to the left of the first view

Clip other corner: Pick point below the first view and to the right of the first view.  
View name: give any name  
Ucs/Ortho/Auxiliary/Section/<eXit>: X

In order to create the section view follow:

Command: SOLVIEW

Ucs/Ortho/Auxiliary/Section/<eXit>: S

Cutting plane's 1<sup>st</sup> point: Pick a point on the center of the drawn first viewport projection on the left side

Cutting plane 2<sup>nd</sup> point: Pick a point on the center using the ortho mode on the right side

Side to view from: Pick a point on the side from where section view to be seen.

View Center: Select a point below of the first view

View center: Enter

Clip first corner: Pick point above the first view and to the left of the first view

Clip other corner: Pick point below the first view and to the right of the first view.

View name: give any name

Ucs/Ortho/Auxiliary/Section/<eXit>: X

### **SOLDRAW Command**

The SOLDRAW command uses the view created by the SOLVIEW command and it generates the proper lines of the view which includes section hatching & hidden lines.

Command: SOLDRAW

Select viewports to draw: Select all the viewports

Select objects: Other corner : four viewports found

Select objects: enter

### **SOLPROF Command**

It is similar to SOLDRAW command but it does not requires the view generated from the SOLVIEW command. This command is used to generate isometric view and can be used in paper space. Using SOLPROF command use the following commands:

Command: \_mview

ON/OFF/Hideplot/Fit/2/3/4/Restore/<First Point>: Pick first corner at the top right of the screen

Other corner: Pick the second corner

Command: MS

MSPACE

Command: DDVPOINT

Change the vpoint to 245 degrees from X axis and 45 degrees from XY plane.

Command: SOLPROF

Initializing...

Select Objects: Select the solid object in the isometric viewport

Select Object: Enter

Display hidden profile lines on separate layer? Enter

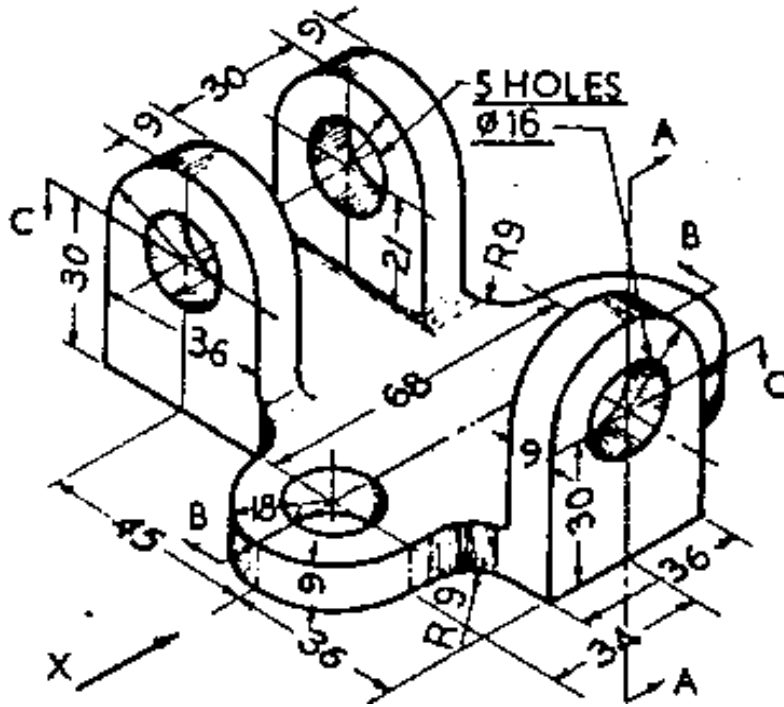
Project profile lines onto a plane?<Y>: Enter

Delete tangential edges?<Y>: Enter

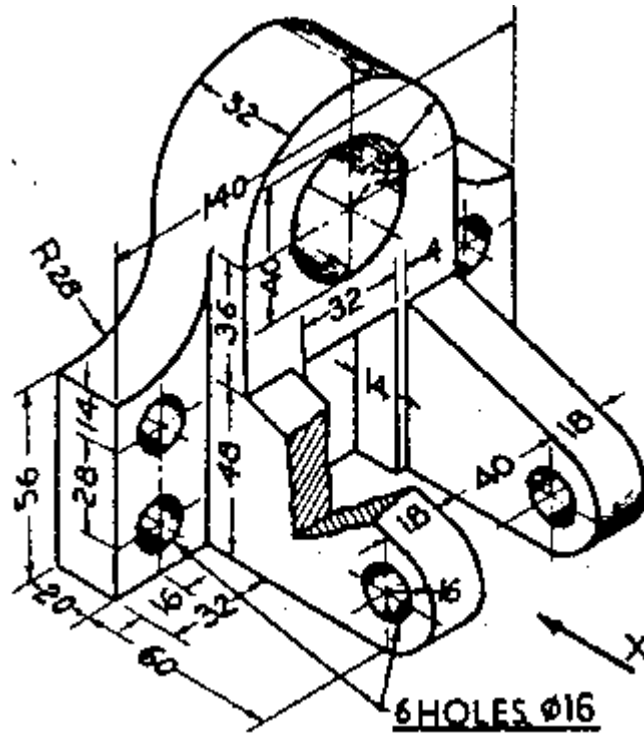
One solid selected.

By this way 2D drawings can be generated from 3D model.

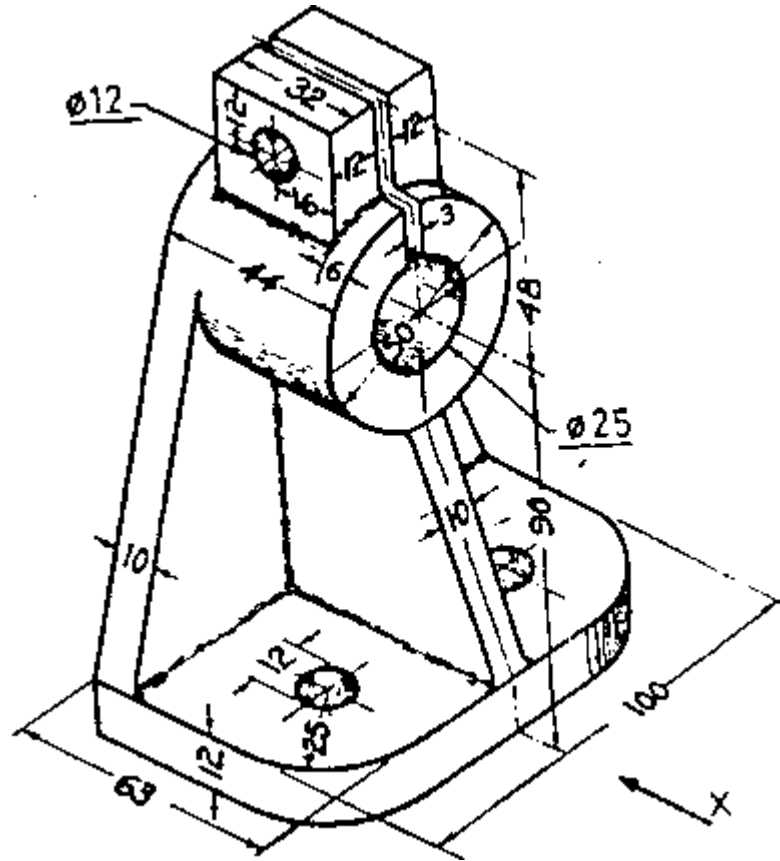
Exercise No. 12 Draw the Following diagram using 3D commands.



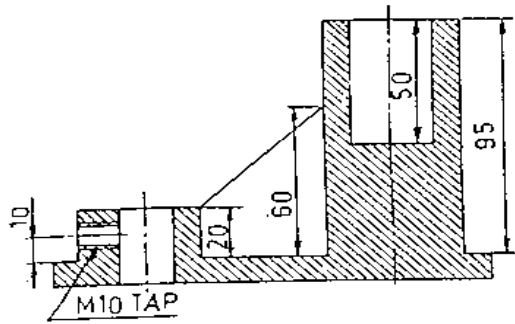
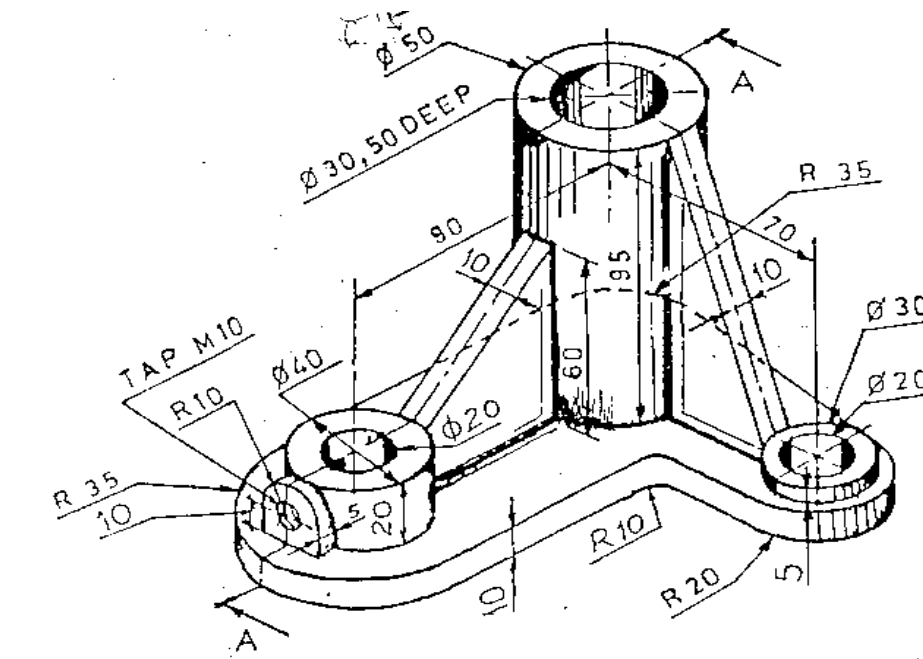
Exercise No.13 Draw the Following diagram using 3D commands.



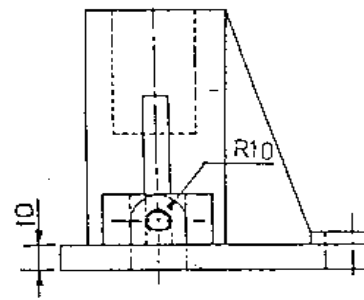
Exercise No.14 Draw the Following diagram using 3D commands.



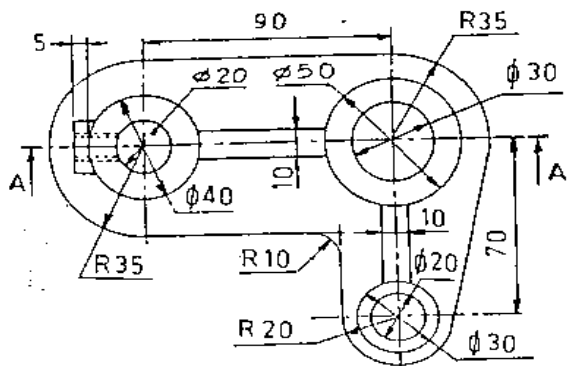
Exercise No.13 Draw the Following diagram using 3D commands.



SECT. ELE. A-A



END-VIEW



Rendering refers to the translation of 3D geometry into one or many fixed coloured 2D images.

### **Lighting the Model**

Lighting is the single most difficult aspect of rendering. Effective lighting produces good results, poor lighting makes the model unattractive.

### **Types Of Light Source**

#### **Ambient Light:**

It is a background light which provides constant illumination to all the surface of a model. It does not come from any source and has no specific direction.

It's intensity can be controlled from AutoCAD Render. The amount of ambient light can be increased or reduced and can be turn off completely. It does not produce any realistic images.

#### **Distant Light:**

A distance light source produces uniform parallel light rays in one direction only. The intensity of distant light does not diminish over distances. The location of a distant light source in a drawing is critical only in terms of its direction not in terms of its distance from any surface.

Distant lights are useful for lighting objects or a backdrop uniformly and for simulating sunlight. A single distant light refers the sun.

#### **Point Light:**

A point light radiates light in all directions from its location. The intensity diminishes over distant according to its rate of attenuation . A light bulb is a good example of point light.

#### **Spot Lights:**

A spot light emits light in the form a cone. The size and direction of the light and the size of the cone can be controlled. The intensity of point light diminishes over distance. When light from a spotlight falls on a surface , the area of maximum illumination is surrounded by an area of lesser intensity.

### **STEPS**

1. Display the Render toolbar by choosing Tools---Toolbar---Render on the menu
2. Click the Render icon to start the render command. The render dialogue box appears on the screen.
3. Click Render scene.

### **Adding a light using the Light command**

#### **STEPS**

1. Click the Light icon. The dialogue box appears on the screen.
2. Choose Point light and click NEW
3. Another box appears and requires you to name the light. Type in the name P! and click Modify. This allows you to place the light source in the model.
4. Follow these prompts:

#### **Command: light**

Enter light location : .xy

Of (need Z): -10

The light is placed to the left of the model so that light from it will strike the model at approximately a right angle from you observation point. After that you will return to the dialogue box.

5. Click Use colour Wheel, and the dialogue box appears.

6. Choose a magenta colour from the outer area of the wheel
7. Change the colour system to HLS (short for Hue, Luminance, and Saturation).
8. Increase the Lightness slider to .90.
9. Click OK in the dialogue boxes until you are returned to the drawing.

### **Adding materials to the Model**

To make the images more realistic, materials can be applied such as steel, copper, brass, etc to the images. The steps involved are:

#### **STEPS**

1. Access the material dialogue box, either through the sixth icon in the render toolbar or by typing **rmat** at the command prompt. The material dialogue box allows you to create or to modify materials or to choose from the material library.
2. Click Material library and the dialogue box appears. This box allows you to access and modify a material library that is kept external to the current drawing file.
3. Choose the Stainless-steel material from the list by using scroll bars.
4. Click Preview to get a sample rendering of the material.
5. Choose the Yellow material from the list and notice the Preview button is grayed out. Choose Stainless-steel again to remove it from the materials selected to allow you to preview the Yellow material.
6. Make sure Stainless-steel, Yellow, and Cyan are selected in the library list and then click Import. The materials appears in the Material list on the left-hand side of the dialogue box. These materials are now available for you to use in the model.
7. Click OK to return to main Material dialogue box.
8. Choose the material Yellow from the list on the left, then click By Layer. The Attach by layer dialogue box appears.
9. With the Yellow material highlighted, choose the layer To which you want to attach colour and click Attach.
10. Click OK to exit the Attach by layer box and again to exit Materials dialogue box.
- 11.** Render your isometric view.

## **PLOTTING**

### ***The Plot Dialogue Box***

Star setting up plot by entering the PLOT command, choosing the Print icon, or Print from the FILE menu. He dialogue box appears is shown in fig.



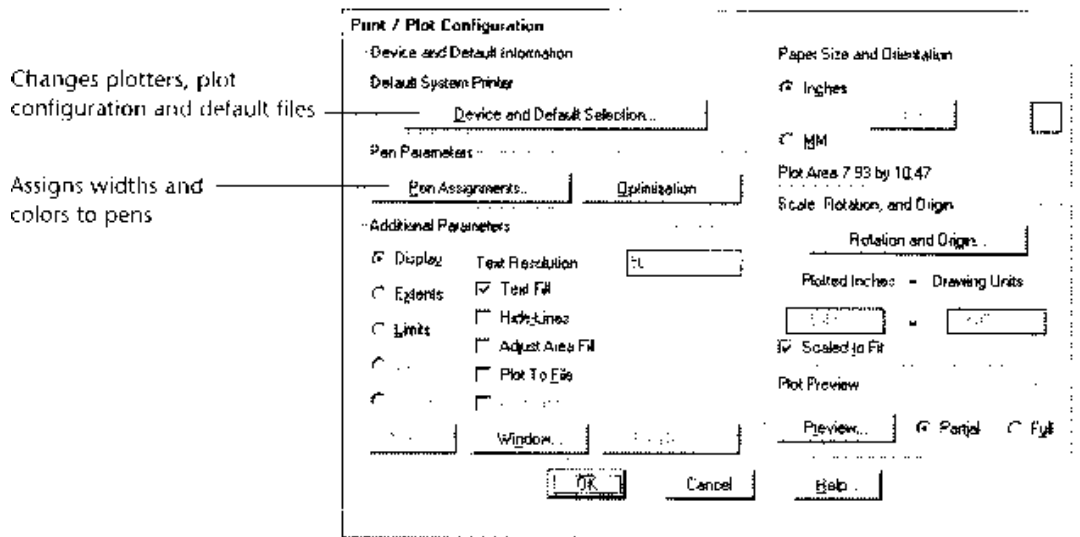


Fig. Courtesy From Autodesk Release-14 Manual

### Device and Default information

AutoCAD displays the description of the currently selected device in this area of the Plot configuration dialogue box. You can view or change information about configured plot devices by choosing Device and Default selection.

### Pen Parameters

Many plotters lets you plot different AutoCAD colours with different pen, line, width, line type, and speed.

### Paper size and orientation

Use this section of the dialogue box to set the units and alignment for the plot. When you choose size, you see a list of paper sizes available for the selected plot device.

### Scale, rotation, and origin

If you need to rotate the image or modify the origin point of the image, you use the plot rotation and origin option.

### Plot preview

You can confirm the paper size, origin, and image area settings on screen, before you waste a sheet of Vellum or Mylar.

### Window

This allows you to place drawing fully on paper. Then click OK to send your drawing to Printer.

## QUICK GLANCE AT COMMANDS

3D	Creates three-dimensional polygon mesh objects
3DARRAY	Creates a three-dimensional array
3DFACE	Creates a three-dimensional face

3DMESH	Creates a free-form polygon mesh
3DPOLY	Creates a polyline of straight line segments in three-dimensional space
3DSIN	Imports a 3D Studio File
3DSOUT	Exports to a 3D Studio file
'ABOUT	Displays information about AutoCAD
ACISIN	Imports an ACIS file
ACISOUT	Exports AutoCAD solid objects to an ACIS file
ALIGN	Moves and rotates objects to align with other objects
AMECONVERT	Converts AME solid models to AutoCAD solid objects
'APERTURE	Controls the size of the object snap target box
'APLOAD	Loads AutoLISP, ADS, and ARX applications
ARC	Creates an arc
AREA	Calculates the area and perimeter of objects or of defined areas
ARRAY	Creates multiple copies of objects in a pattern
ARX	Loads, unloads, and provides information about ARX applications
ASEADMIN	Performs administrative functions for external database commands
ASEEXPORT	Exports link information for selected objects
ASELINKS	Manages links between objects and an external database
ASEROWS	Displays and edits table data and creates links and selection sets
ASESELECT	Creates a selection set from rows linked to textual selection sets and graphic selection sets
ASESQLED	Executes Structured Query Language (SQL) statements
ATTDEF	Creates an attribute definition
'ATTDISP	Globally controls attribute visibility
ATTEDIT	Changes attribute information independent of its block definition
ATTEXT	Extracts attribute data
ATTREDEF	Redefines a block and updates associated attributes
AUDIT	Evaluates the integrity of a drawing
BACKGROUND	Sets up the background for your scene
'BASE	Sets the insertion base point for the current drawing
BHATCH	Fills an enclosed area with a hatch pattern
'BLIPMODE	Controls the display of marker blips
BLOCK	Creates a block definition from selected objects
BMAKE	Defines a block using a dialog box
BMPOUT	Saves selected objects to a file in device-independent bitmap format
BOUNDARY	Creates a region or a polyline from an enclosed area
BOX	Creates a three-dimensional solid box
BREAK	Erases parts of objects or splits an object in two
BROWSER	Launches the default Web browser defined in your system's registry
'CAL	Evaluates mathematical and geometric expressions
CHAMFER	Bevels the edges of objects
CHANGE	Changes the properties of existing objects
CHPROP	Changes the color, layer, linetype, linetype scale factor, and thickness of an object

CIRCLE	Creates a circle
'COLOR	Sets the color for new objects
COMPILE	Compiles shape files and PostScript font files
CONE	Creates a three-dimensional solid cone
CONVERT	Converts 2D polylines and associative hatches to the optimized Release 14 format
COPY	Duplicates objects
COPYCLIP	Copies objects to the Clipboard
COPYHIST	Copies the text in the command line history to the Clipboard
COPYLINK	Copies the current view to the Clipboard for linking to other OLE applications
CUTCLIP	Copies objects to the Clipboard and erases the objects from the drawing
CYLINDER	Creates a three-dimensional solid cylinder
DBLIST	Lists database information for each object in the drawing
DDATTDEF	Creates an attribute definition
DDATTE	Edits the variable attributes of a block
DDATTEXT	Extracts attribute data
DDCHPROP	Changes the color, layer, linetype, and thickness of an object
DDCOLOR	Sets the color for new objects
DDEDIT	Edits text and attribute definitions
'DDGRIPS	Turns on grips and sets their color
DDIM	Creates and modifies dimension styles
DDINSERT	Inserts a block or another drawing
DDMODIFY	Controls properties of existing objects
'DDPTYPE	Specifies the display mode and size of point objects
DDRENAME	Changes the names of named objects
'DDRMODES	Sets drawing aids
'DDSELECT	Sets object selection modes
DDUCS	Manages defined user coordinate systems
DDUCSP	Selects a preset user coordinate system
'DDUNITS	Controls coordinate and angle display formats and determines precision
DDVIEW	Creates and restores views
DDVPOINT	Sets the three-dimensional viewing direction
'DELAY	Provides a timed pause within a script
DIM	Accesses Dimensioning mode
DIMALIGNED	Creates an aligned linear dimension
DIMANGULAR	Creates an angular dimension
DIMBASELINE	Continues a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension
DIMCENTER	Creates the center mark or the center lines of circles and arcs
DIMCONTINUE	Continues a linear, angular, or ordinate dimension from the second extension line of the previous or a selected dimension
DIMDIAMETER	Creates diameter dimensions for circles and arcs
DIMEDIT	Edits dimensions
DIMLINEAR	Creates linear dimensions
DIMORDINATE	Creates ordinate point dimensions

DIMOVERRIDE	Overrides dimension system variables
DIMRADIUS	Creates radial dimensions for circles and arcs
DIMSTYLE	Creates and modifies dimension styles on the command line
DIMTEDIT	Moves and rotates dimension text
'DIST	Measures the distance and angle between two points
DIVIDE	Places evenly spaced point objects or blocks along the length or perimeter of an object
DONUT	Draws filled circles and rings
'DRAGMODE	Controls the way dragged objects are displayed
DRAWORDER	Changes the display order of images and other objects
DSVIEWER	Opens the Aerial View window
DTEXT	Displays text on screen as it is entered
DVIEW	Defines parallel projection or perspective views
DWFOUT	Exports a drawing web format file
DXBIN	Imports specially coded binary files
DXFIN	Imports a drawing interchange file
DXFOUT	Creates a drawing interchange file of the current drawing
EDGE	Changes the visibility of three-dimensional face edges
EDGESURF	Creates a three-dimensional polygon mesh
'ELEV	Sets elevation and extrusion thickness properties of new objects
ELLIPSE	Creates an ellipse or an elliptical arc
ERASE	Removes objects from a drawing
EXPLODE	Breaks a compound object into its component objects
EXPORT	Saves objects to other file formats
EXTEND	Extends an object to meet another object
EXTRUDE	Creates unique solid primitives by extruding existing two-dimensional objects
'FILL	Controls the filling of multilines, traces, solids, solid-fill hatches, and wide polylines
FILLET	Rounds and fillets the edges of objects
'FILTER	Creates lists to select objects based on properties
FOG	Provides visual cues for the apparent distance of objects
'GRAPHSCR	Switches from the text window to the graphics area
'GRID	Displays a dot grid
GROUP	Creates a named selection set of objects
HATCH	Fills a specified boundary with a pattern
HATCHEDIT	Modifies an existing hatch object
'HELP ()	Displays online help
HIDE	Regenerates a three-dimensional model with hidden lines suppressed

'ID	Displays the coordinate values of a location
IMAGE	Inserts images in many formats into an AutoCAD drawing file
IMAGEADJUST	Controls the brightness, contrast and fade values of the selected image
IMAGEATTACH	Attaches a new image object and definition
IMAGECLIP	Creates new clipping boundaries for single image objects
IMAGEFRAME from view	Controls whether the image frame is displayed on the screen or hidden
IMAGEQUALITY	Controls the display quality of images
IMPORT	Imports various file formats into AutoCAD
INSERT	Places a named block or drawing into the current drawing
INSERTOBJ	Inserts a linked or embedded object
INTERFERE	Finds the interference of two or more three-dimensional solids and creates a composite 3D solid from their common volume
INTERSECT	Creates composite solids or regions from the intersection of two or more solids or regions
'ISOPLANE	Specifies the current isometric plane
'LAYER	Manages layers
LEADER	Creates a line that connects annotation to a feature
LENGTHEN	Lengthens an object
LIGHT	Manages lights and lighting effects
'LIMITS	Sets and controls the drawing boundaries
LINE	Creates straight line segments
'LINETYPE	Creates, loads, and sets linetypes
LIST	Displays database information for selected objects
LOAD	Makes shapes available for use by the SHAPE command
LOGFILEOFF	Closes the log file opened by LOGFILEON
LOGFILEON	Writes the text window contents to a file
LSEDIT	Lets you edit a landscape object
LSLIB	Lets you maintain libraries of landscape objects
LSNEW	Lets you add realistic landscape items, such as trees and bushes, to your drawings
'LTSCALE	Sets the linetype scale factor
MASSPROP	Calculates and displays the mass properties of regions or solids
'MATCHPROP	Copies the properties from one object to one or more objects
MATLIB	Imports and exports materials to and from a library of materials
MEASURE	Places point objects or blocks at measured intervals on an object
MENU	Loads a menu file
MENULOAD	Loads partial menu files
MENUUNLOAD	Unloads partial menu files
MINSERT	Inserts multiple instances of a block in a rectangular array
MIRROR	Creates a mirror image copy of objects
MIRROR3D	Creates a mirror image of objects about a plane
MLEDIT	Edits multiple parallel lines
MLINE	Creates multiple parallel lines
MLSTYLE	Defines a style for multiple parallel lines
MOVE	Displaces objects a specified distance in a specified direction

MSLIDE	Creates a slide file of the current viewport
MSPACE	Switches from paper space to a model space viewport
MTEXT	Creates multiline text
MULTIPLE	Repeats the next command until canceled
MVIEW	Creates floating viewports and turns on existing floating viewports
MVSETUP	Sets up the specifications of a drawing
NEW	Creates a new drawing file
OFFSET	Creates concentric circles, parallel lines, and parallel curves
OLELINKS	Updates, changes, and cancels existing OLE links
OOPS	Restores erased objects
OPEN	Opens an existing drawing file
'ORTHO	Constrains cursor movement
'OSNAP	Sets running object snap modes and changes the target box size
'PAN	Moves the drawing display in the current viewport
PASTECLIP	Inserts data from the Clipboard
PASTESPEC	Inserts data from the Clipboard and controls the format of the data
PEDIT	Edits polylines and three-dimensional polygon meshes
PFACE	Creates a three-dimensional polyface mesh vertex by vertex
PLAN	Displays the plan view of a user coordinate system
PLINE	Creates two-dimensional polylines
PLOT	Plots a drawing to a plotter, printer, or file
POINT	Creates a point object
POLYGON	Creates an equilateral closed polyline
PREFERENCES	Customizes the AutoCAD settings
PREVIEW	Shows how the drawing will look when it is printed or plotted
PSDRAG	Controls the appearance of a PostScript image as it is dragged into position with PSIN
PSFILL	Fills a two-dimensional polyline outline with a PostScript pattern
PSIN	Imports a PostScript file
PSOUT	Creates an Encapsulated PostScript file
PSPACE	Switches from a model space viewport to paper space
PURGE	Removes unused named objects, such as blocks or layers, from the database
QSAVE	Saves the current drawing
'QTEXT	Controls the display and plotting of text and attribute objects
QUIT	Exits AutoCAD
RAY	Creates a semi-infinite line
RECOVER	Repairs a damaged drawing
RECTANG	Draws a rectangular polyline
REDEFINE	Restores AutoCAD internal commands overridden by UNDEFINE
REDO	Reverses the effects of the previous UNDO or U command
'REDRAW	Refreshes the display of the current viewport

'REDRAWALL	Refreshes the display of all viewports
REGEN	Regenerates the drawing and refreshes the current viewport
REGENALL	Regenerates the drawing and refreshes all viewports
'REGENAUTO	Controls automatic regeneration of a drawing
REGION	Creates a region object from a selection set of existing objects
REINIT	Reinitializes the digitizer, digitizer input/output port, and program parameters file
RENAME	Changes the names of objects
RENDER	Creates a realistically shaded image of a three-dimensional wireframe or solid model
REPLAY	Displays a BMP, TGA, or TIFF image
'RESUME	Continues an interrupted script
REVOLVE	Creates a solid by revolving a two-dimensional object about an axis
REVSURF	Creates a rotated surface about a selected axis
RMAT	Manages rendering materials
ROTATE	Moves objects about a base point
ROTATE3D	Moves objects about a three-dimensional axis
RPREF	Sets rendering preferences
RSCRIPT	Creates a script that repeats continuously
RULESURF	Creates a ruled surface between two curves
SAVE	Saves the drawing under the current file name or a specified name
SAVEAS	Saves an unnamed drawing with a file name or renames the current drawing
SAVEIMG	Saves a rendered image to a file
SCALE	Enlarges or reduces selected objects equally in the X, Y, and Z directions
SCENE	Manages scenes in model space
'SCRIPT	Executes a sequence of commands from a script
SECTION	Uses the intersection of a plane and solids to create a region
SELECT	Places selected objects in the Previous selection set
SETUV	Lets you map materials onto geometry
'SETVAR	Lists or changes the values of system variables
SHADE	Displays a flat-shaded image of the drawing in the current viewport
SHAPE	Inserts a shape
SHELL	Accesses operating system commands
SHOWMAT	Lists the material type and attachment method for a selected object
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
SOLPROF	Creates profile images of three-dimensional solids
SOLVIEW	Creates floating viewports using orthographic projection to lay out multi- and sectional view drawings of 3D solid and body objects
'SPELL	Checks spelling in a drawing
SPHERE	Creates a three-dimensional solid sphere

SPLINE	Creates a quadratic or cubic spline (NURBS) curve
SPLINEDIT	Edits a spline object
STATS	Displays rendering statistics
'STATUS	Displays drawing statistics, modes, and extents
STLOUT	Stores a solid in an ASCII or binary file
STRETCH	Moves or stretches objects
'STYLE	Creates named styles
SUBTRACT	Creates a composite region or solid by subtraction
SYSWINDOWS	Arranges windows
TABLET	Calibrates, configures, and turns on and off an attached digitizing tablet
TABSURF	Creates a tabulated surface from a path curve and a direction vector
TEXT	Creates a single line of text
'TEXTSCR	Opens the AutoCAD text window
'TIME	Displays the date and time statistics of a drawing
TOLERANCE	Creates geometric tolerances
TOOLBAR	Displays, hides, and customizes toolbars
TORUS	Creates a donut-shaped solid
TRACE	Creates solid lines
TRANSPARENCY	Controls whether background pixels in an image are transparent or opaque
'TREESTAT	Displays information about the drawing's current spatial index
TRIM	Trims objects at a cutting edge defined by other objects
U	Reverses the most recent operation
UCS	Manages user coordinate systems
UCSICON	Controls the visibility and placement of the UCS icon
UNDEFINE	Allows an application-defined command to override an internal AutoCAD command
UNDO	Reverses the effect of commands
UNION	Creates a composite region or solid
'UNITS	Sets coordinate and angle display formats and precision
'VIEW	Saves and restores named views
VIEWRES	Sets the resolution for object generation in the current viewport
VPLAYER	Sets layer visibility within viewports
VPOINT	Sets the viewing direction for a three-dimensional visualization of the drawing
VPORTS	Divides the graphics area into multiple tiled viewports
VSLIDE	Displays a raster image slide file in the current viewport
WBLOCK	Writes objects to a new drawing file
WEDGE	Creates a 3D solid with a sloped face tapering along the X axis
WMFIN	Imports a Windows metafile
WMFOPTS	Sets options for WMFIN
WMFOUT	Saves objects to a Windows metafile



XATTACH	Attaches an external reference to the current drawing
XBIND	Binds dependent symbols of an xref to a drawing
XCLIP	Defines an xref clipping boundary and sets the front or back clipping planes
XLINE	Creates an infinite line
XPLODE	Breaks a compound object into its component objects
XREF	Controls external references to drawing files
ZOOM	Increases or decreases the apparent size of objects in the current viewport