

Computer Graphics

19.1 INTRODUCTION

The previous chapters of the book have described the fundamental concepts of manual drawing using various drawing instruments and producing a hardcopy of the drawing. This traditional drawing however still remains the foundation for engineering communication, as the subject of computer-aided drafting (CAD) is developed on the fundamentals of traditional drawing. So the advent of computers aided drafting (CAD) does not necessarily eliminate manual engineering graphics.

There are several reasons for implementing a computer aided design system, a few of them are as follows :

- To increase the productivity of the designer.
- To improve the quality of design.
- To improve the communication.
- To create a database for manufacturing.

19.2 COMPUTER GRAPHICS

The term 'computer graphics' is very generic in the sense that it has different meaning for different disciplines of people. In the operation of the graphics system by user, a variety of activities take place, which can be divided into three categories. Interact with graphics terminal to create and alter to images on the screen, construct a model of something physical and of the image on the screen and enter the model into computer memory. In working with the graphics system, the user performs these various activities in combination rather than sequentially.

19.3 REQUIREMENTS OF COMPUTER GRAPHICS

It comprises of CAD workstation and requisite software. CAD workstation comprises of common PC of minimum of pentium configuration, a video display, a mouse or pen, a plotter or printer. The graphics software can be divided into three modules : the graphic package, the application program and the application database. Various requirements of graphics software are :

- It should be easy to use.
- The package should operate in a consistent and predictable way to the user.
- Graphics programs should be efficient.
- It should not be too expensive to make their use prohibitive.

This chapter explores the AutoCAD package for its basic drafting capabilities.

19.4 GETTING STARTED WITH AUTO CAD

Start AutoCAD by clicking on the windows start button placed at the bottom left. then move the mouse to programs, click on AutoCAD as shown in Fig. 19.1 or can directly be opened by double clicking on the desktop icon by AutoCAD.

The AutoCAD starts to load. In the process of start up a dialog box, as shown in Fig. 19.2, giving various start-up options will be displayed. Select the option 'Start from scratch'; click on metric radio button and OK. Fig. 19.3 shows a typical initial AutoCAD window screen. The AutoCAD window screen has a number of important features :

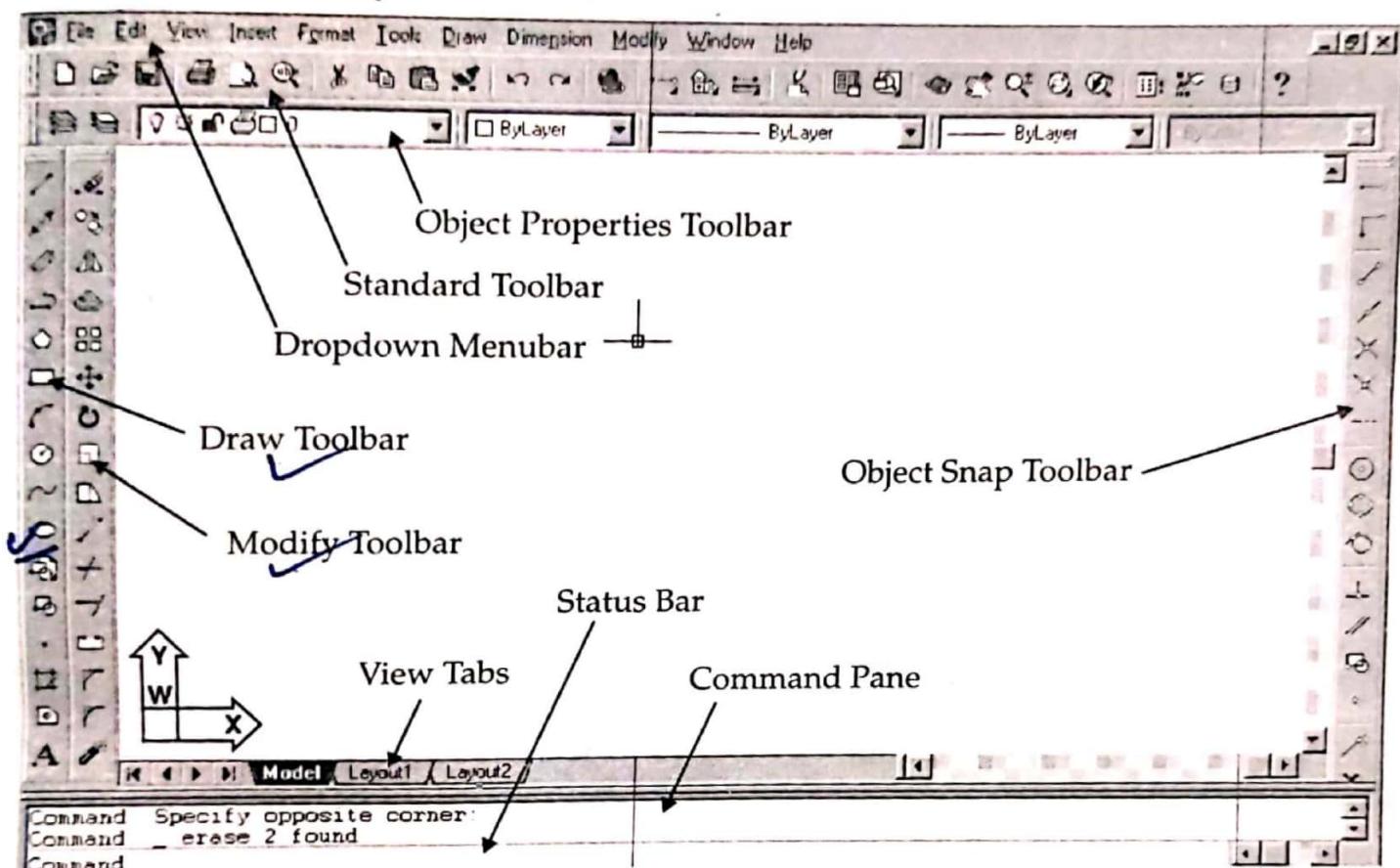


Fig. 19.3 Typical initial AutoCAD screen

- The top line displays the windows pull-down menus for exiting a program and changing a program.
- The second line is the standard toolbar and contains a group of the most commonly used commands.
- The third line contains some command icons and an area that shows the current, clocked and object properties that are active.
- The line just above the drawing portion of the screen displays the name of the current drawing. Once a drawing name has been defined, it will appear at the top of the screen.
- The bottom left corner of the screen shows the coordinate display position.
- View tabs give access to different views of the current drawing. The model tab should be selected generally.
- Command pane is the place to type commands.
- The commands listed on the status line are displayed in light gray when they are off and in black when they are on.
- The large open area in the centre of the screen is called the drawing area.
- The two rectangular boxes of command icons, located along the left edge of the drawing screen are the draw and modify toolbars.

19.5 SAVING A DRAWING

There are three ways to save a drawing.

- Go to the top left corner of the screen and click on 'File' option and then select 'Save' as shown in Fig. 19.4.

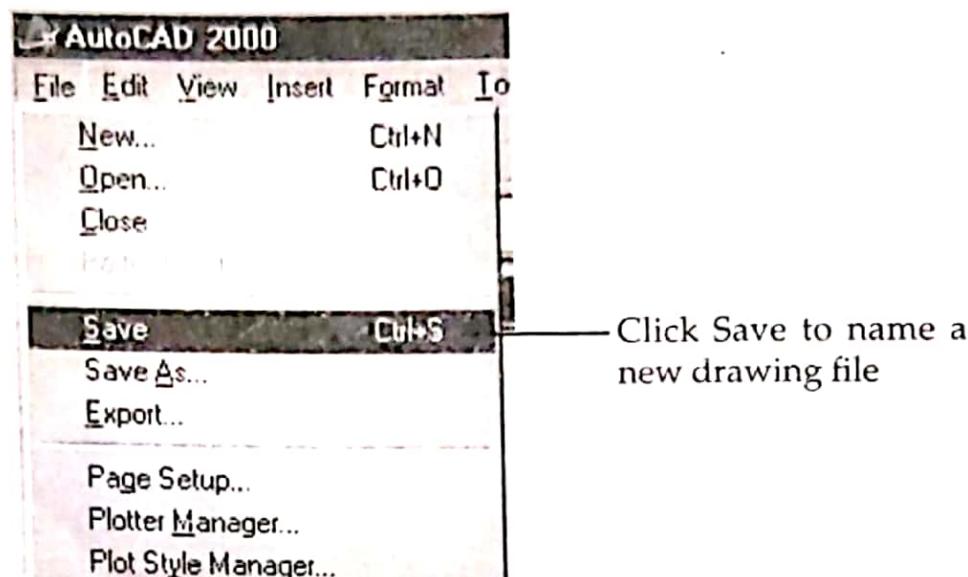


Fig. 19.4 How to save a drawing

- The quickest way is to save a drawing by pressing **Ctrl + S**.
- The another way is to select the save icon from the standard toolbar.

19.6 COMMAND ENTRY

There are four methods of giving a command. There are listed as below :

- First is on the command pane.
- Second is from the side screen.
- Third is from the toolbar.
- Fourth is from dropdown menus.

19.7 DRAWING LIMITS ✓

This is the first command, which is to be given in AutoCAD. The lower left corner is normally kept at 0,0 and the upper right corner depends upon the size of the figure to be drawn. By default, the size of the screen is 12×9 . If the size of the figure is 200×200 units. Then the size of the upper right corner should be more than that, in order to fit the figure on the screen.

To set the drawing limits

Command : _limits.

19.8 UNITS

This command will help in selecting appropriate units for the given figure i.e., mm, cm, m, inches etc.

Command : _units ; enter

See Figs. 19.5 and 19.6

Select the appropriate units and also the precision i.e., how many number of zero after decimal points. Set the units and precision for angles too.

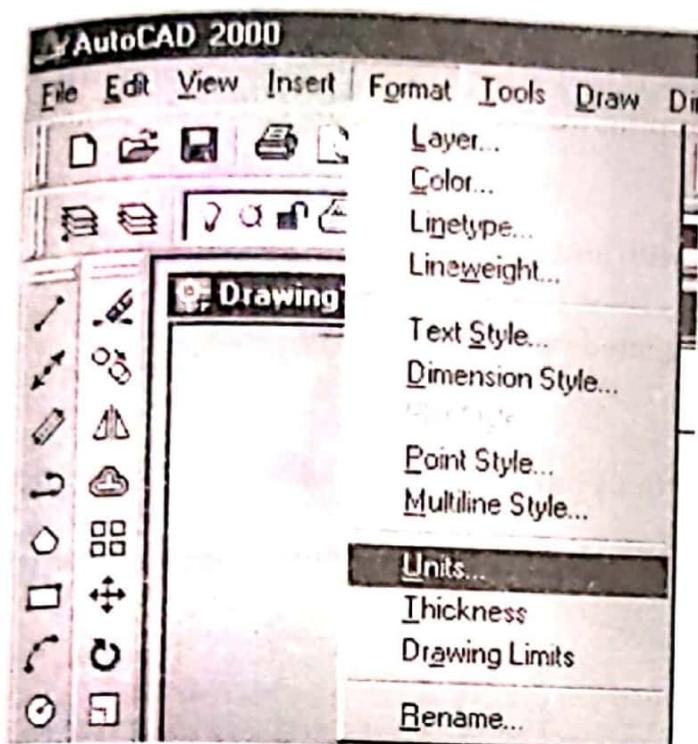


Fig. 19.5 Selecting of drawing units from drop down menu bar

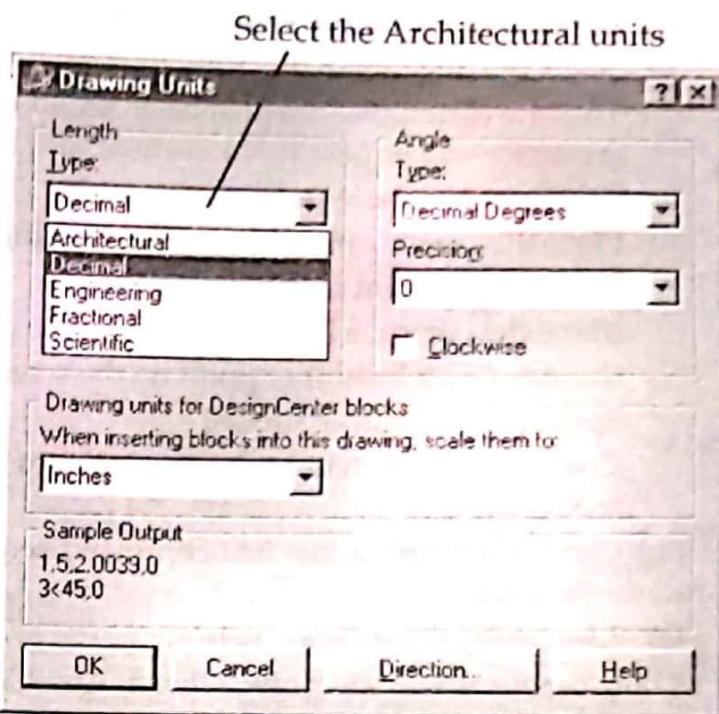


Fig. 19.6 How to select the drawing units

19.9 DRAW COMMANDS

These commands are arranged in draw drop down menu bar or standard draw toolbar as shown in Fig. 19.7. Some of these commands will be discussed in subsequent paragraphs.

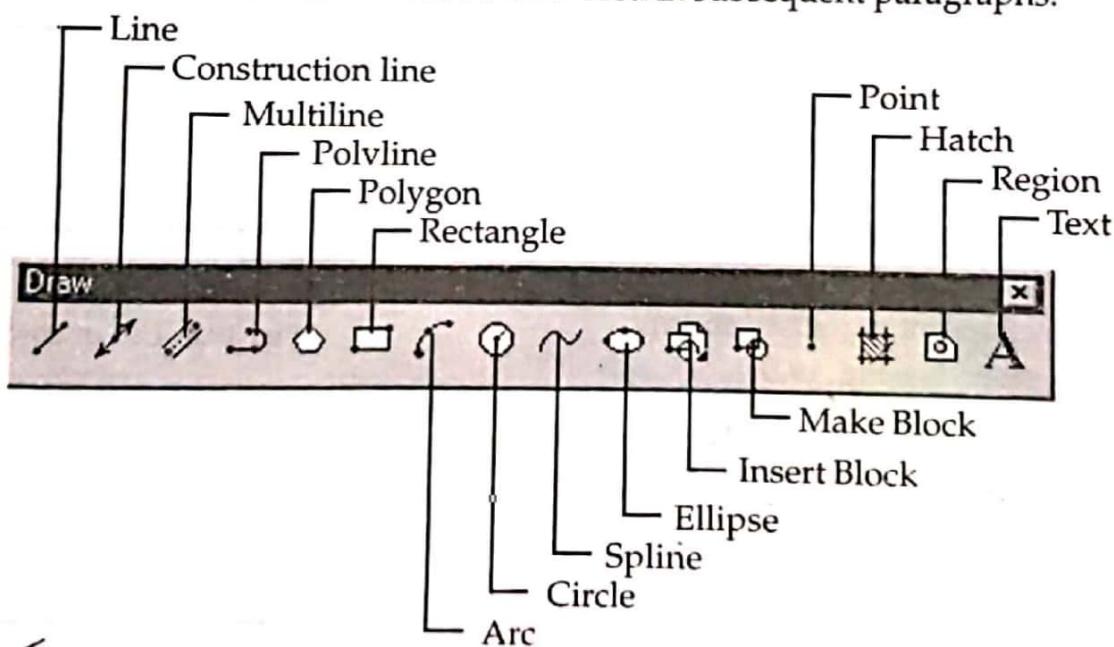


Fig. 19.7 Draw Toolbar

Line
There are different ways to define the length and locations of a line ; these are randomly selected points, enter the coordinate values for the start and end points or specify the first point, the length and the direction of the line.

(a) Line using randomly select points

- (i) Select the line tool in the draw toolbar.

The line command can also be accessed by typing the word line. The following command sequence will appear in the command.

Command : _line; enter

- (ii) Place the cursor randomly on the drawing screen and press the left mouse button.

Specify next point or [undo] :

Move the cursor, a line extends from the designated point to the cursor.

- (iii) Randomly pick another point on the screen.

Specify next point or [undo] :

Command pane will keep on asking for another point until press, either enter or the right mouse button.

(iv) If enter is pressed, the line command sequence will end. Pressing enter a second time will restart the line command.

(b) Line using Coordinate values

There are three ways to draw a line in AutoCAD

 **Absolute Mode**

Draw 50 mm × 50 mm rectangle starting at the 10, 10 coordinate point by using absolute, incremental and polar mode.

- (i) Select the line tool from the draw toolbar

Command : _line ; enter

Specify the first point :

- (ii) Type 10, 10 ; enter

Specify next point or [undo] :

- (iii) Type 60, 10 ; enter

Specify next point or [clear/undo] :

- (iv) Type 60, 40 ; enter

Specify next point or [close/undo] :

- (v) Type 10, 40 ; enter

Specify next point or [close/undo] :

- (vi) Type C, enter. See Fig. 19.8

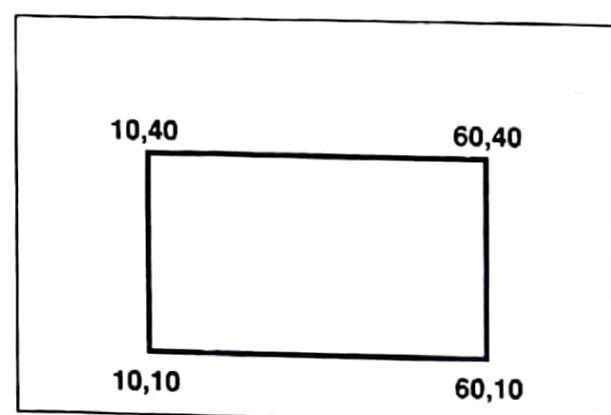


Fig. 19.8 Use of absolute mode

 **Incremental Mode / Relative**

- (i) Select the line tool from the draw toolbar

Command : _line ; enter

Specify first point.

- (ii) Type 10, 10 ; enter

Specify next point or [undo] :

- (iii) Type @ 50, 0 ; enter

Specify next point or [close/undo] :

- (iv) Type @ 0, 30 ; enter

Specify next point or [close/undo] :

- (v) Type @ -50, 0 ; enter

Specify next point or [close/undo] :

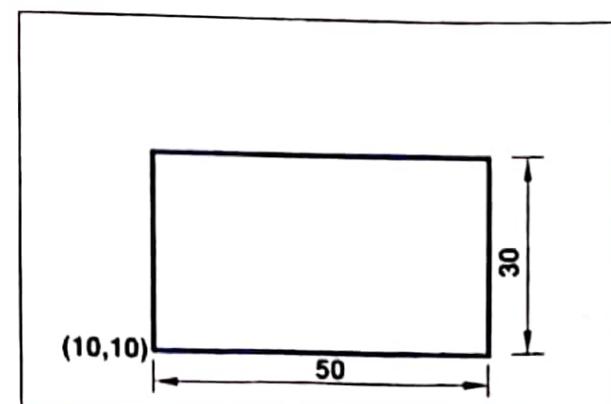


Fig. 19.9. Use of incremental mode

(vi) Type @ 0, -30, (or C) ; enter. See Fig. 19.9

Absolute coordinate measures distance from the origin of AutoCAD system. Whereas, the relative coordinates assign the distances with respect to the current position.

Polar Mode

(i) Select the line tool from the draw toolbar

Command : _line ; enter

Specify first point.

(ii) Type 10, 10 ; enter

Specify next point or [undo] :

(iii) Type @ 50 < 0 ; enter

Specify next point or [close/undo] :

(iv) Type @ 30 < 90 ; enter

Specify next point or [close/undo] :

(v) Type @ 50 < 180 ; enter

Specify next point or [close/undo] :

(vi) Type @ 30 < 270 (or @ 30 < -90 or C) ; enter. See Fig. 19.10

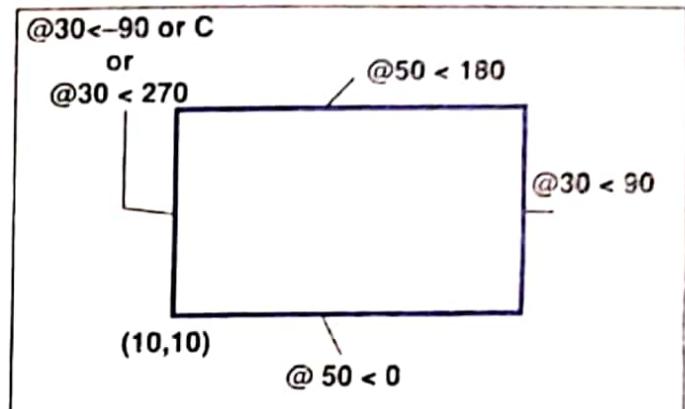


Fig. 19.10 Use of polar mode

Polyline

A polyline is a line made from a series of individual, connected line segments that act as a single entity. Polyline are used to generate curves, splines and polygon etc. These are also used in three-dimensional to produce solid objects.

To draw a polyline.

(i) Select a polyline tool from the draw toolbar

Command : _pline ; enter

Specify start point.

(ii) Select a start point.

Specify next point or [arc/close/halfwidth/length/undo/width]

(iii) Select a second point.

Specify next point.

(iv) Select several more points.

Specify next point.

(v) Press the right mouse button ; then enter. See Fig. 19.11

To draw a polyline – arc.

(i) Select a polyline tool from the draw toolbar

Command : _pline

Specify start point.

(ii) Select a start point

Specify next point or [arc/close/halfwidth/length/undo/width] :

(iii) Type A ; then enter

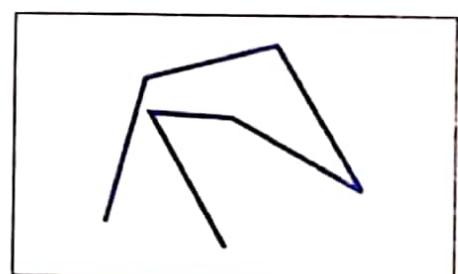


Fig. 19.11 Use of polyline

Specify end point of arc or [angle/center/close/direction/halfwidth/line/radius/second point/undo/width].

- Select the point.

Specify endpoint of arc or [angle/center/close/direction/halfwidth/line/radius/second point/undo/width].

- Select another point.

- Press the right mouse button, then enter. See Fig. 19.12.

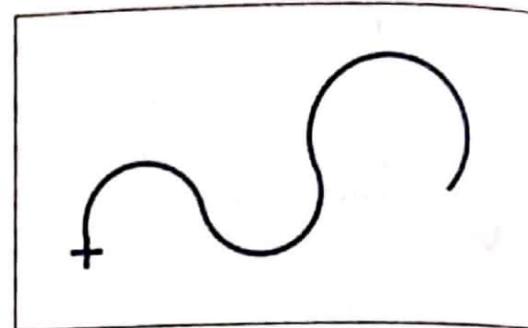


Fig. 19.12 Polyline drawn using the arc option

Similarly, other options of a polyline may be used depending upon the requirement.

Circle

A circle can be drawn by any of the option as shown in Fig. 19.13. Depending on the requirement of the drawing any of these options may be selected, some of them are as follows :

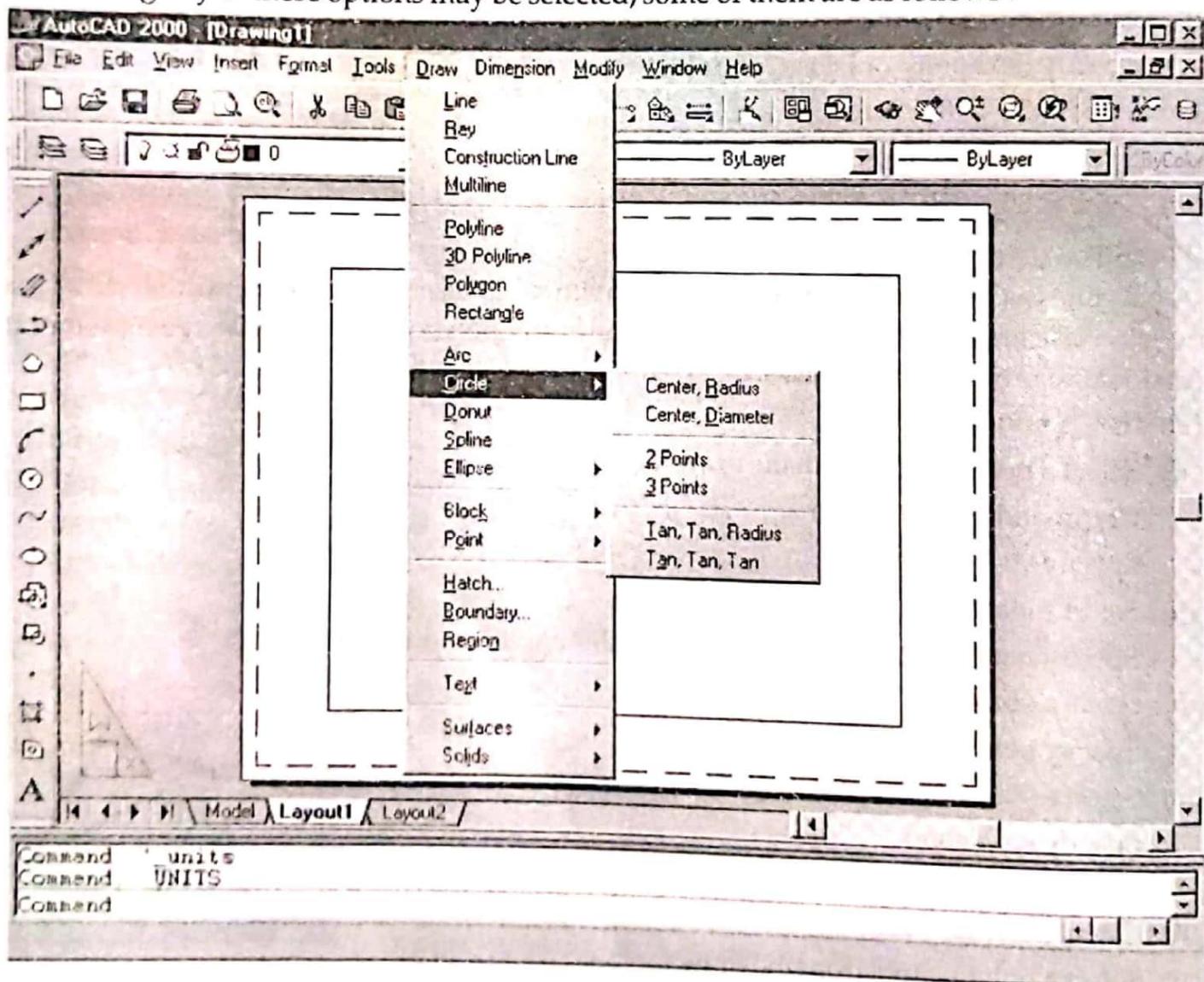


Fig. 19.13 Various options available to draw a circle

To draw a circle - **radius**

- Select the circle tool from the draw toolbar

Command : _circle ; enter

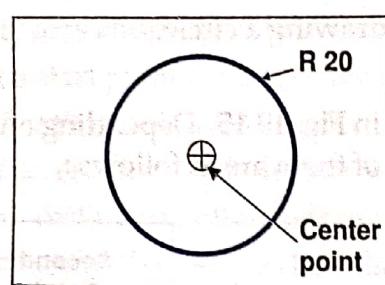
Specify a centre point for circle or [3P/2P/Ttr (tan tan radius)]

- Select a center point.

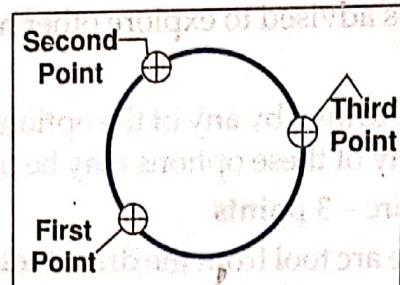
COMPUTER GRAPHICS

Specify radius of circle or [diameter] :

- (iii) Give value, then enter. See Fig. 19.14.



(a)



(b)

Fig. 19.14 Use of circle command specifying various option.

To draw a circle – 3 points

- (i) Select the circle tool from the draw toolbar

Command : _circle ; enter

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

- (ii) Type 3 P ; enter

Specify first point of a circle :

- (iii) Select a first point.

Specify second point on circle :

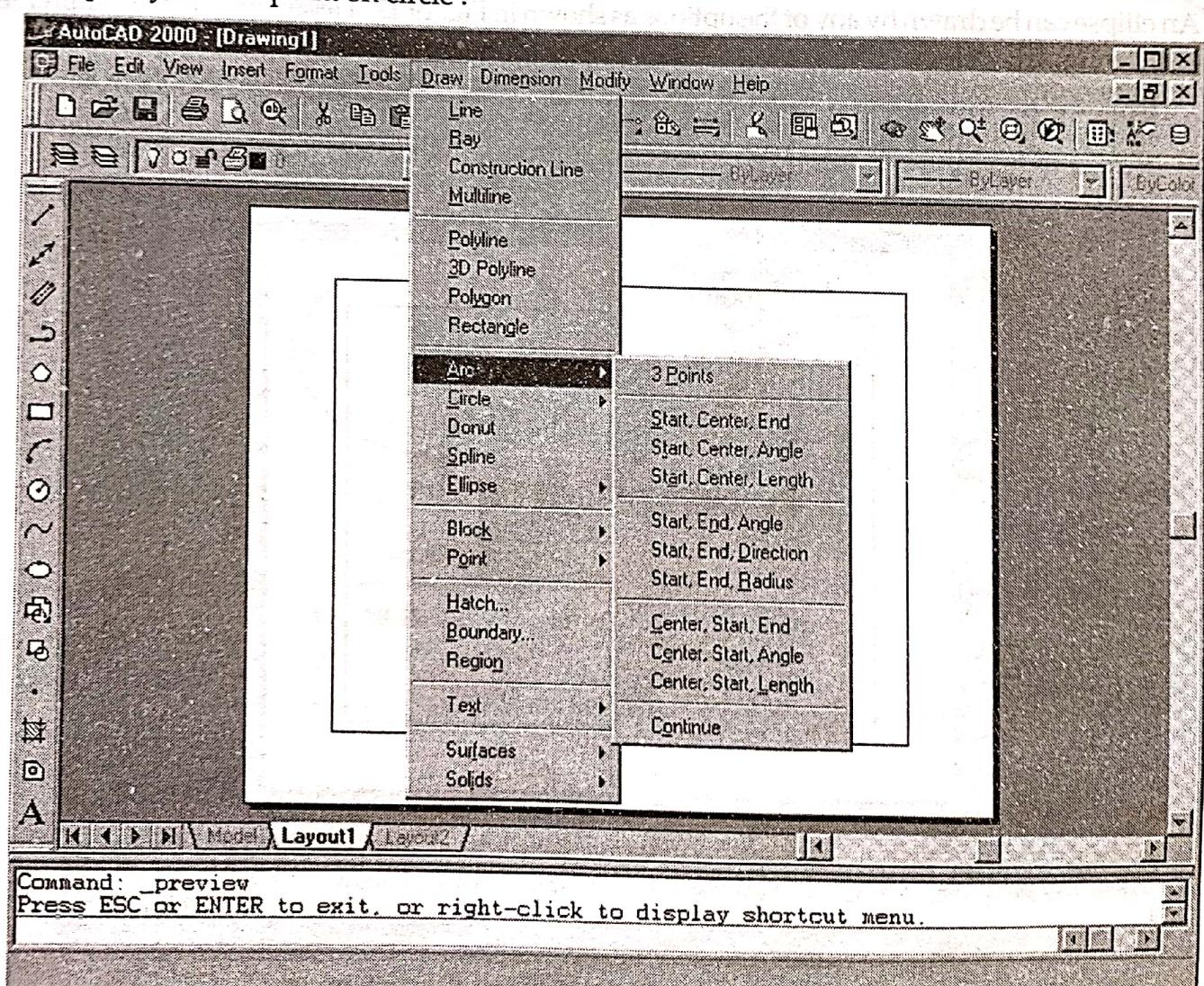


Fig. 19.15 Various options available to draw an arc

- (iv) Select a second point.
Specify third point on circle.

- (v) Select a third point.

The student is advised to explore other methods of drawing a circle.

Arc

An arc can be drawn by any of the options as shown in Fig. 19.15. Depending on the requirement of the drawing any of these options may be used. Some of them are as follows :

To draw an arc – 3 points

- (i) Select the arc tool from the draw toolbar.

Command : _arc

Specify start point of the arc or [enter] :

- (ii) Select a start point.

Specify second point of the arc or [center/end] :

- (iii) Select a second point.

Specify end point of the arc :

- (iv) Select an end point. See Fig. 19.16.

The student is advised to explore other methods of drawing an arc.

Ellipse

An ellipse can be drawn by any of the options as shown in Fig. 19.17. Depending on the requirement of the drawing, any of these options may be used. Some of them are as follows :

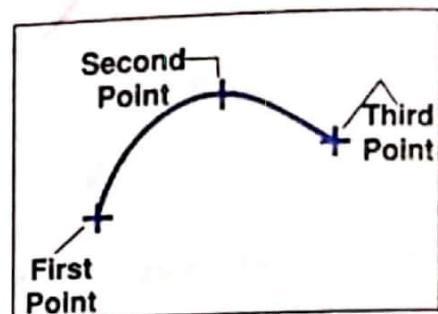


Fig. 19.16 An arc drawn through three points using arc command

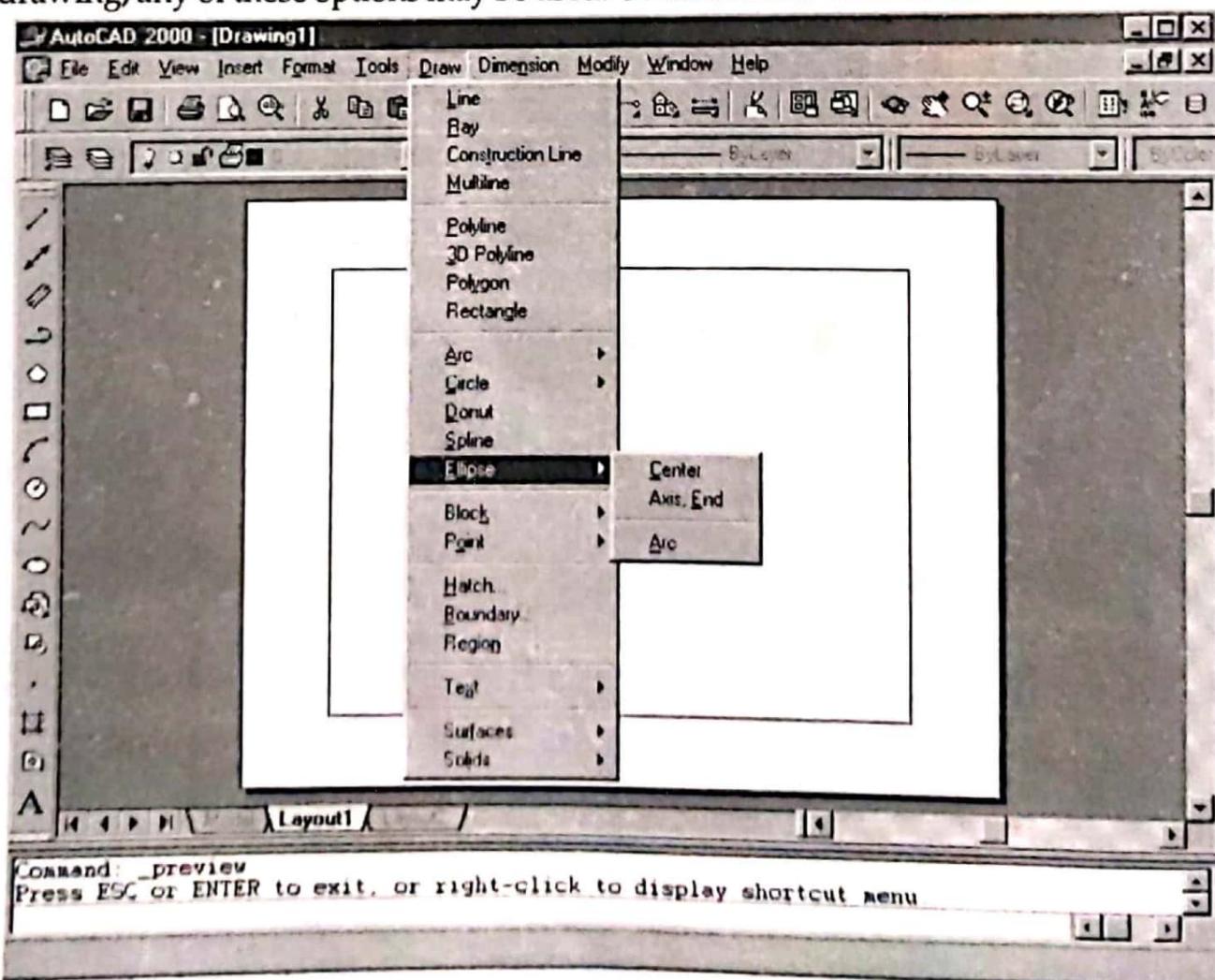


Fig. 19.17 Various options available to draw an ellipse

To draw an ellipse – axis, end

- Select the ellipse tool from the draw toolbar.

Command : _ellipse ; enter

Specify axis endpoint of ellipse or [arc/center] :

- Select a start point for one of the axes

Specify other end point of axis :

- Select an endpoint that defines the length of the axis.

Specify distance to other axis or (rotation) :

- Select a point that defines half the length of the other axis

Points 1 and 2 define the major axis

Points 3 and 4 define the minor axis

To draw an ellipse – center

- Select the ellipse tool from the draw toolbar.

Command : _ellipse ; enter

Specify axis endpoint of ellipse or [arc/center] :

- Type C ; press enter

Specify the center point of the ellipse :

- Select the center point of the ellipse.

Specify axis end point.

- Select one of the endpoints of one of the axes.

Specify distance to the other axis or (rotation) :

- Select a point that defines half the length of the other axis. See Fig. 19.18.

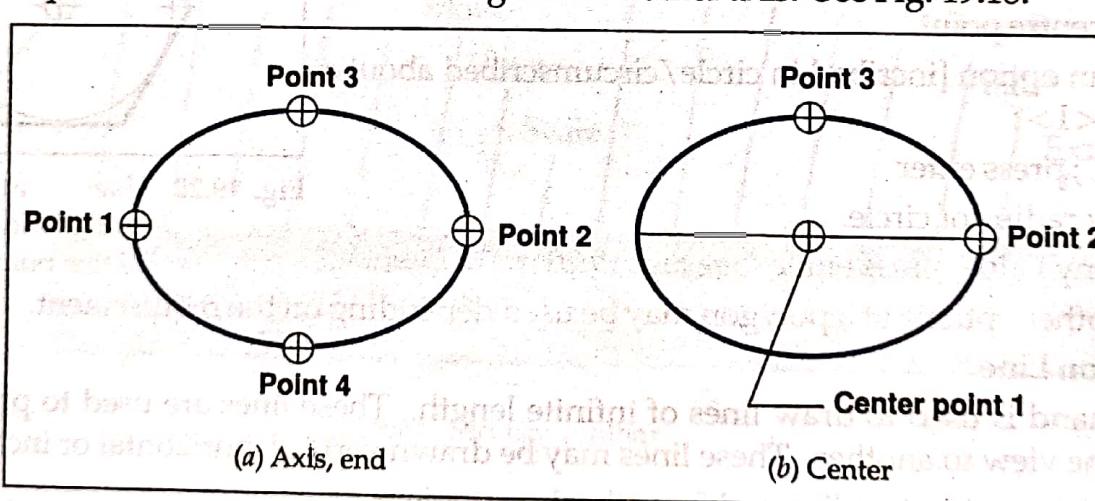


Fig. 19.18 Drawing an ellipse - specifying various options

Rectangle

This command is used to draw rectangles by specifying two diagonal points.

To draw a rectangle.

- Select the rectangle tool from the draw toolbar.

Command : _rectangle ; enter

Specify first corner point or (chamfer/elevation/ fillet/thickness/width) :

- Select a point.

- Specify other corner point :**
 (iii) Select a point. See Fig. 19.19.

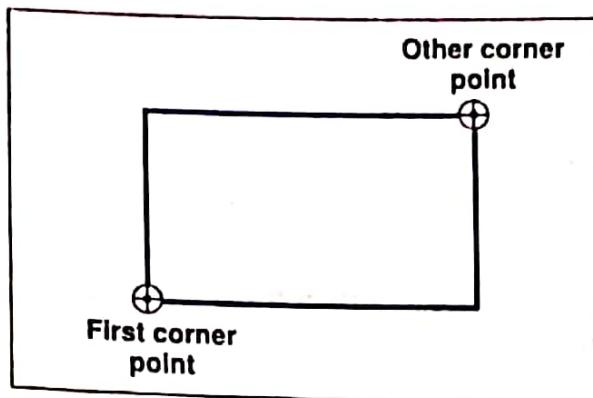


Fig. 19.19 Drawing a rectangle

The student is advised to work with different options mentioned above.

Polygon

A polygon is a closed figure bounded by straight lines. Only regular polygons will be drawn here. A regular polygon with six equal sides is called a regular hexagon.

To draw a polygon - center point

- (i) Select the polygon tool from the draw toolbar

Command : _polygon

Enter number of sides < 4 > :

- (ii) Type 6 ; press enter

Edge 1 < center of polygon > :

- (iii) Select centre point.

Enter an option [inscribed in circle/circumscribed about a circle] < I > :

- (iv) Type C ; press enter

Specify radius of circle.

- (v) Type any value, press enter. See Fig. 19.20.

Similarly, other options of a polygon may be used depending on the requirement.

Construction Line

This command is used to draw lines of infinite length. These lines are used to project various entities from one view to another. These lines may be drawn vertical, horizontal or inclined.

- (i) Select the construction line tool from the draw toolbar.

Command : _xline ; enter

Specify a point or [Hor./Ver/Ang/Bisect/Offset] :

- (ii) Select a starting point.

Specify through point :

A line will pivot about the designated starting point and extend an infinite length in a direction through the cursor.

- (iii) Select a through point.

A line of infinite length is drawn through the two designated points.

- (iv) Specify through point :

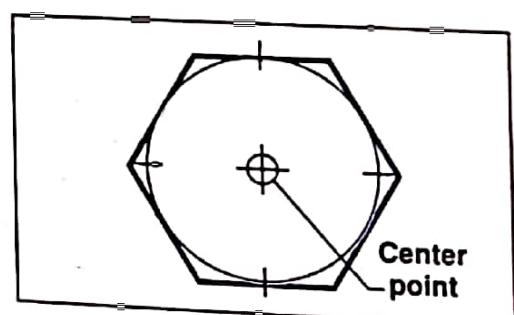


Fig. 19.20 Use of polygon command specifying center point

Another infinite line will appear through the starting point through the cursor.

- Press enter. See Fig. 19.21

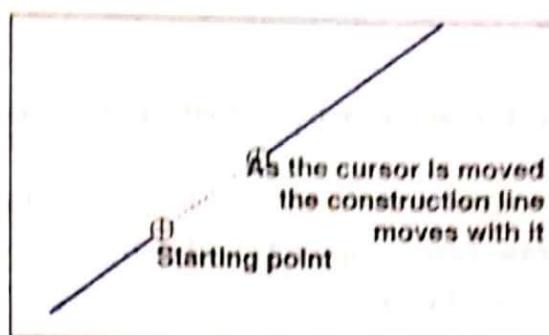


Fig. 19.21 Drawing a construction line

19.10 MODIFY COMMANDS

These commands are used to establish specific functions such as editing, transformation etc. on an existing graphic. These commands are categorically arranged in modify dropdown menu bar. A standard modify toolbar is shown in Fig. 19.22.

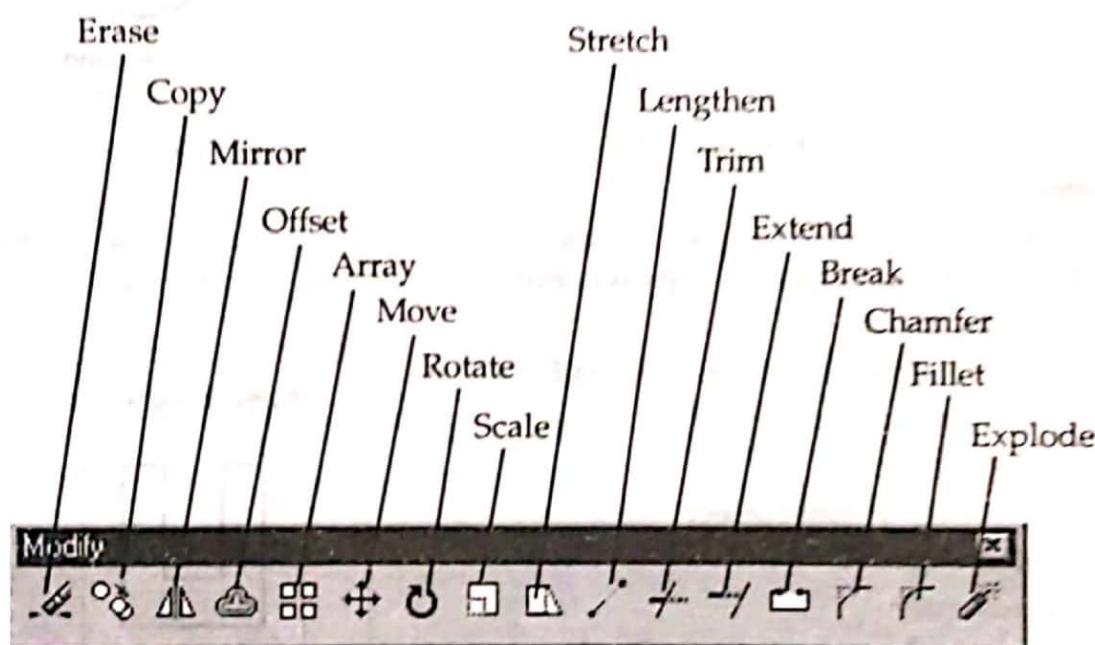


Fig. 19.22 Modify toolbar

Erase

This command is used to erase any number of entities, which have been created in the current drawing. There are two ways to erase lines : select individual lines or window a group of lines.

To erase individual lines

- Select the erase tool from the modify toolbar.

Command : _erase ; enter

Select objects.

- Select the two open lines by placing the rectangular cursor on each line, one at a time and pressing the left mouse button.
- Press the right mouse button or enter to complete the erase command.

To erase a group of lines simultaneously

- (i) Select the erase tool from the modify toolbar.
Command : _erase ; enter
Select objects.
- (ii) Place the rectangular select cursor above and to the left of the lines to be erased and press the left mouse button.
- (iii) Move the cursor and a window drag from the selected first point.
- (iv) When all the lines to be erased are completely within the window, press the left mouse button.
- (v) Press the right mouse button or enter to complete the erase command. See Fig. 19.23.

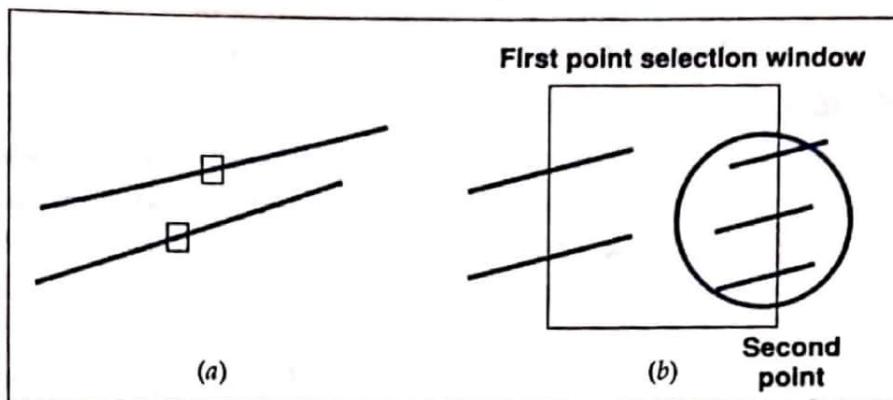


Fig. 19.23 Different methods to erase a group of lines

Copy

This command is used to make an exact copy of an existing line or object. This command can also be used to create more than one copy without reactivating the command.

To copy an object

- (i) Select the copy tool from the modify toolbar
Command : _copy ; enter
Select objects.
- (ii) Window the entire object
Select objects.
- (iii) Press enter
Specify base point or displacement or [multiple] :
- (iv) Select a base point
Specify second point of displacement or < use first point of displacement :
- (v) Select a second displacement point.

The original object remains in its original location and a new object appears at the second displacement point as shown in Fig. 19.24.

Similarly, other options of a copy command may also be useful depending on the requirement.

Mirror

This command is very useful when drawing symmetrical objects. Only half of the object need to

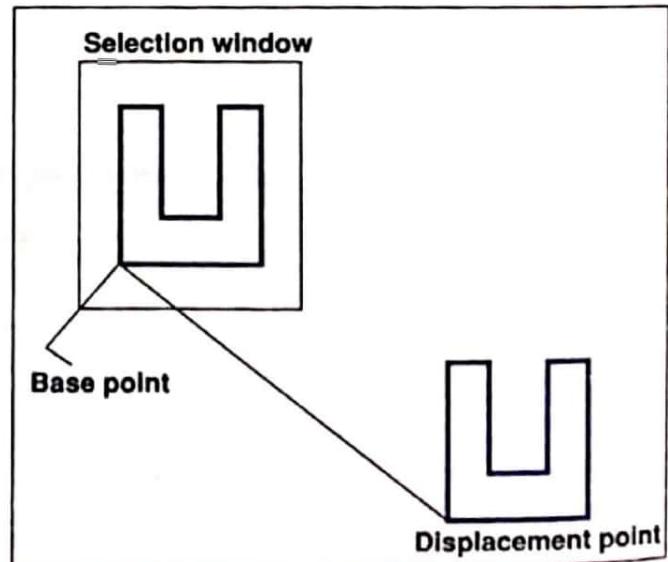


Fig. 19.24 Use of copy command.

be drawn. The second half can be created using the mirror command.

To mirror an object

- (i) Select the mirror tool from the modify toolbar

Command : _mirror ; enter

Select objects.

- (ii) Window the object

Select objects :

- (iii) Press enter.

Specify first point of mirror line.

- (iv) Select a point on the mirror line.

Specify second point of mirror line :

- (v) Select a second point on the mirror line.

Delete source object ? [Yes/No] < N > :

- (vi) Press enter, see Fig. 19.25.

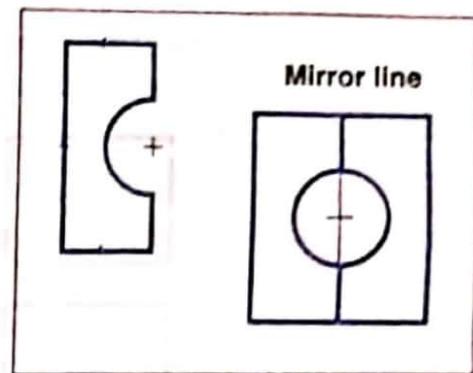


Fig. 19.25 Use of mirror command

✓ Offset

This command is used to construct a new entity that is parallel to an existing entity.

To offset an object.

- (i) Select the offset tool from the modify toolbar.

Command : _offset ; enter

Specify offset distance or [Through] :

- (ii) Set the value, enter.

Select object to offset or < exist > :

- (iii) Select a line.

Specify point on side to offset :

- (iv) Select a point to the right side of the line.

Specify object to offset or < exist > :

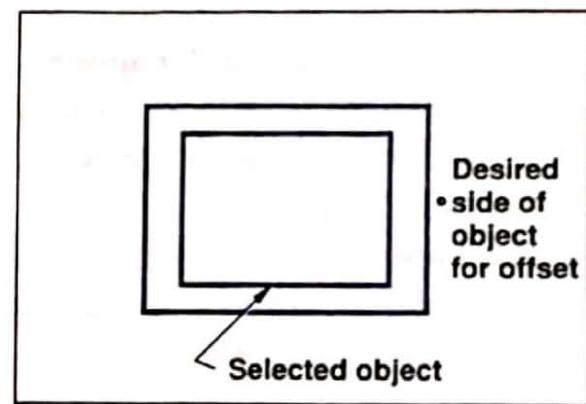


Fig. 19.26 Use of offset command

This process may be repeated by double clicking the right mouse button. See Fig. 19.26.

✓ Array

This command produces multiple copies of selected objects arrayed in a rectangular or polar pattern.

To array an object : **rectangular option**

- (i) Select the array tool from the modify toolbar

Command : _array ; enter

Select objects.

- (ii) Press enter.

Enter type of array [rectangular/polar] < R > :

- (iii) Press enter.

Enter te number of rows (_) < 1 > :

- (iv) Type any value, press enter.

Enter the number of columns (I II) < 1 > :

- (v) Type any value, press enter.
 Enter the distance between rows or specify unit cell [___] :
 (vi) Type any value, press enter.
 Specify distance between columns (III) :
 (vii) Type any value, press enter. See Fig. 19.27.

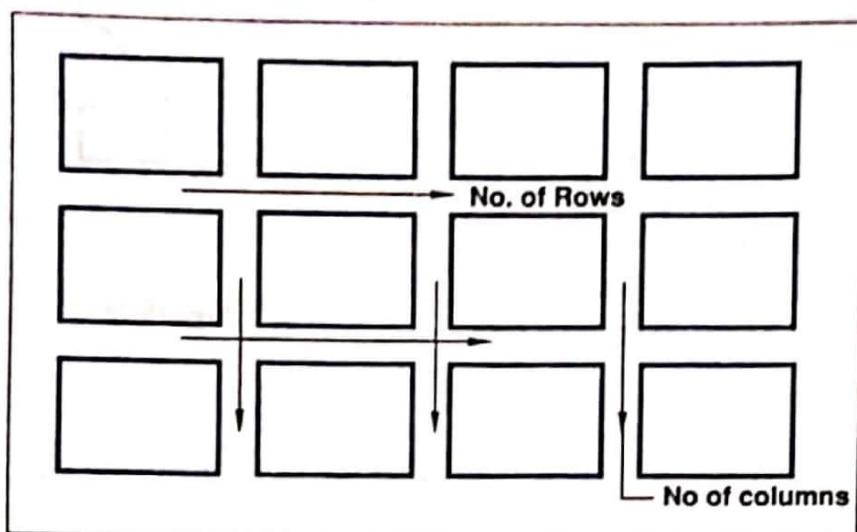


Fig. 19.27 Use of array command – rectangular option

To array an object : polar option

- (i) Select the array tool from the modify toolbar
 Command : _array ; entry
 Select objects.
 (ii) Press enter.
 Enter type of array [rectangular/polar] < R > :
 (iii) Type P, press enter.
 Specify the center point of array :
 (iv) Select a center point
 Enter the number of items in the array :
 (v) Type value, press enter
 Specify angle to fill (+ = ccw, - = cw) < 360 > :
 (vi) Press enter
 Rotate array objects ? [Yes/No] < Y > :
 (vii) Press enter, see Fig. 19.28.

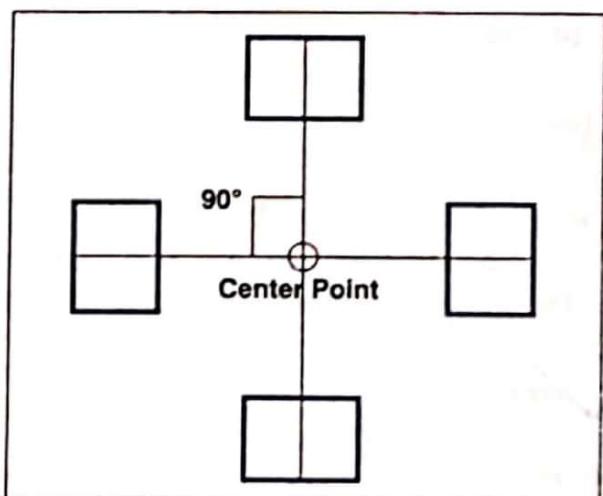


Fig. 19.28 Use of array command – polar option.

Move

This command is used to move a line or object to a new position on the drawing.

To move an object

- (i) Select the move tool from the modify toolbar
 Command : _move ; enter
 Select objects.
 (ii) Window the entire objects
 Select objects.

- (iii) Press enter.
- Specify base point or displacement.
- (iv) Select a base point.
- Specify second point of displacement or < use first point of displacement > :
- (v) Select a second displacement point.
- Press enter, see Fig. 19.29.

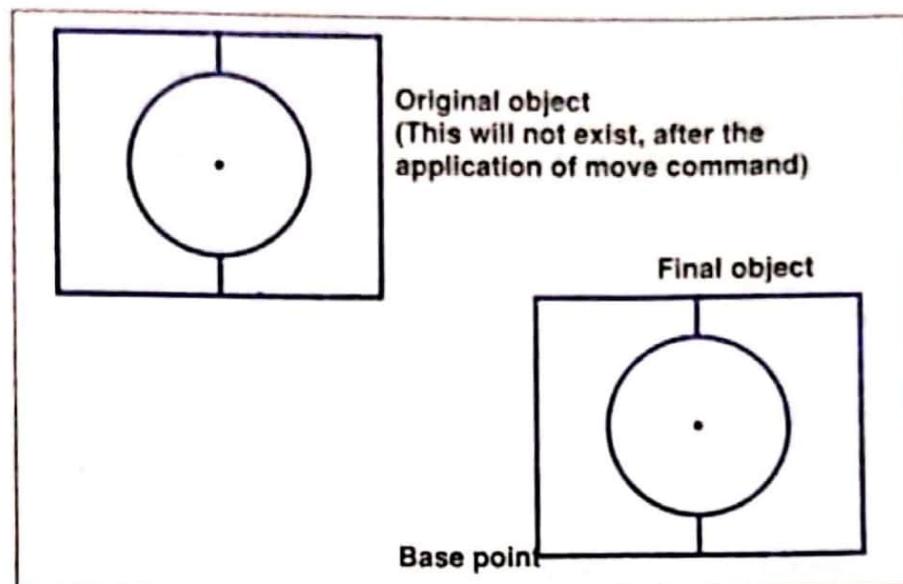


Fig. 19.29 Use of move command

Rotate

This command is used to rotate a group of entities about a base point through a given angle.

To rotate an object

- (i) Select the rotate tool from the modify toolbar

Command : _rotate ; enter

Select objects :

- (ii) Window the objects

Select objects :

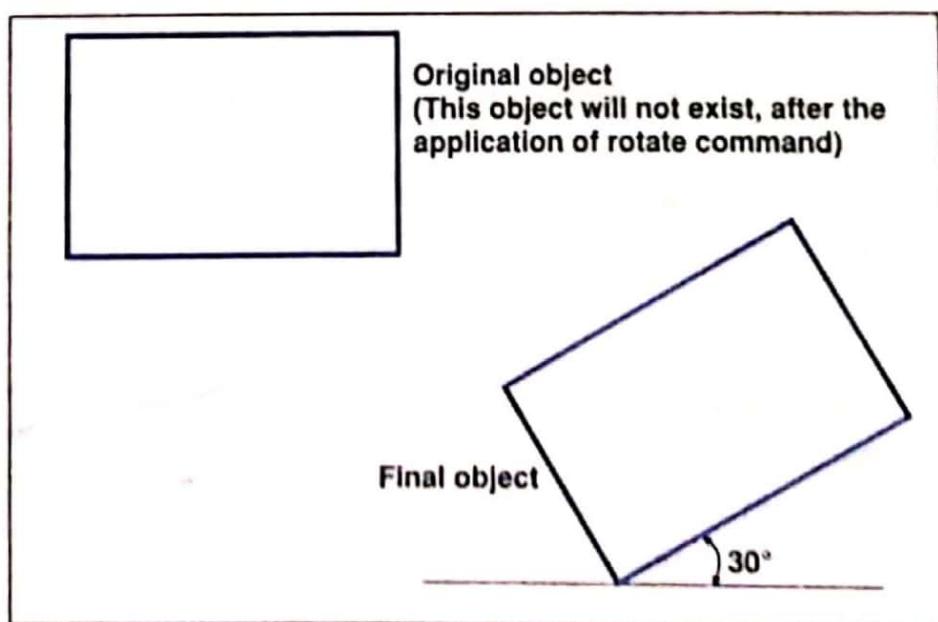


Fig. 19.30 Use of rotate command

(iii) Press enter.

Specify base point :

(iv) Select a base point.

Specify rotation angle or (reference) :

(v) Type given value, press enter. See Fig. 19.30.

Trim

This command is used to cut away excessively long lines.

To use the trim command.

(i) Select the trim tool from the modify toolbar.

Command : _trim ; enter

Select cutting edges.

Specify objects.

(ii) Press enter

Select object to trim or [Project/Edge/Undo] :

(iii) Press enter. See Fig. 19.31.

Extend

This command is used to extend the given lines. It produces the effect opposite to that by the trim command.

To use the extend command.

(i) Select the extend tool from the modify toolbar

Command : _extend ; enter

Select boundary edges.

Select objects.

(ii) Select a line that can be used as a boundary edge.

Select object to extend or [Project/Edge/Undo] :

(iii) Select the lines to be extended, press enter.

See Fig. 19.32.

Break

This command is used to split an existing line, arc, circle into two parts.

To use the break command

(i) Select the break tool from the modify toolbar

Command : _break ; enter

Select object

(ii) Select the line

Specify second break point or [First Point] :

(iii) Type f, press enter

Specify first break point :

(iv) Select the first point of the break.

Specify second break point :

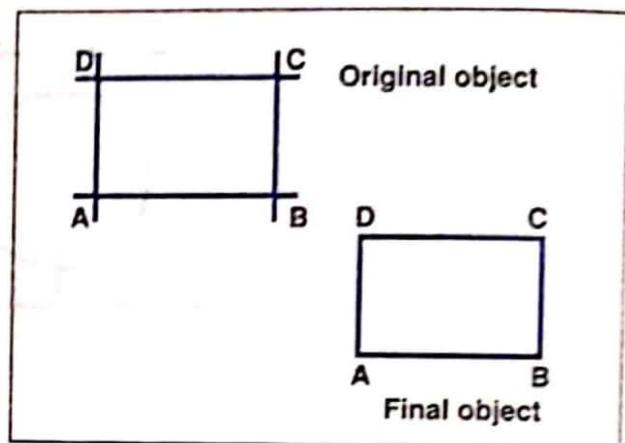


Fig. 19.31 Use of trim command

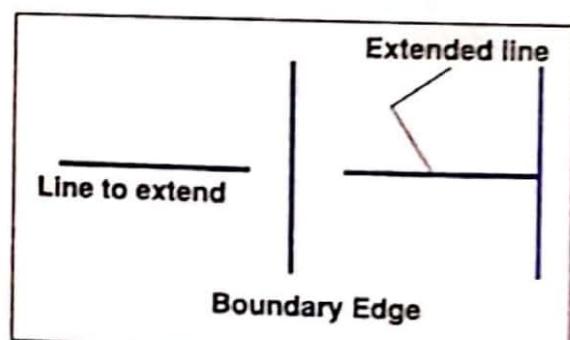


Fig. 19.32 Use of extend command

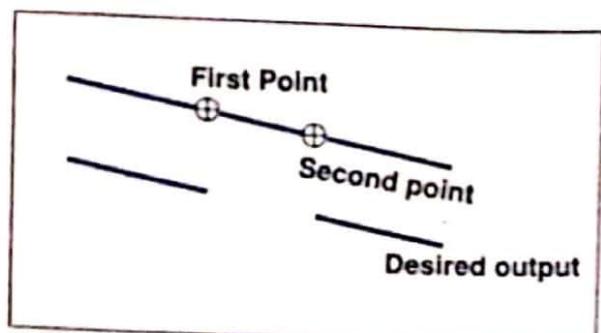


Fig. 19.33 Use of break command

- (v) Select the second point of the break. See Fig. 19.33.

Fillet

This command is used to connect two existing lines, circles or arcs by means of a arc of a given radius.

To use the fillet command.

- (i) Select the fillet tool from the modify toolbar.

Command : _fillet ; enter

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

- (ii) Type R, press enter

Specify fillet radius < 0.5000 > :

- (iii) Type given value, press enter. See Fig. 19.34.

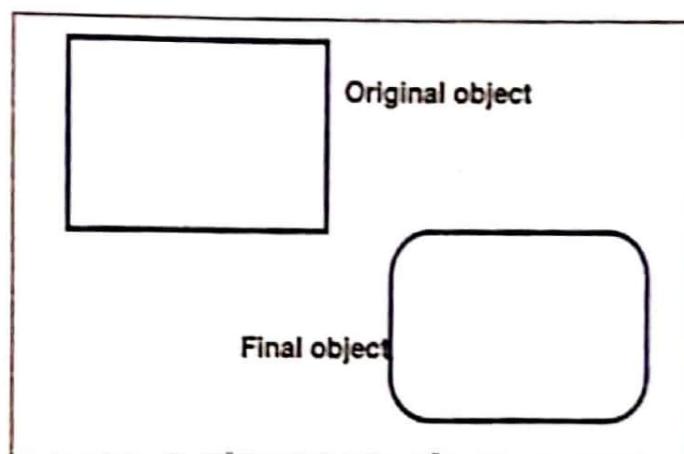


Fig. 19.34 Use of fillet command

The student is advised to work with different options mentioned above.

Chamfer

This command is similar to the fillet except that in chamfer, the corners with a straight edge, which intersects the two objects at a specified distance from their point of intersection.

To use the chamfer command.

- (i) Select the chamfer tool from the modify toolbar.

Command : _chamfer ; enter

Select first line or [Polyline/Distance/Angle/Trim] < Select first line > :

- (ii) Type D, press enter.

Specify first chamfer distance < 0.5000 > :

- (iii) Type given value, press enter

Specify second chamfer distance < 0.5000 > :

- (iv) Type given value, press enter. See Fig. 19.35

The student is advised to work with different options mentioned above. Some important commands of draw and modify toolbar have already been described, which will help the students in making the drawings on computers.

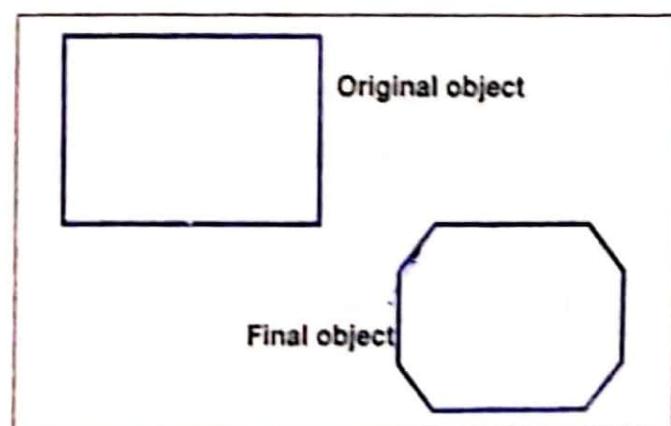


Fig. 19.35 Use of chamfer command