

2. Parametric Design and Basic Drawing Functions

It must be remembered that all designer should have a sound training in producing drawings with traditional drawing instruments, before using computers and they also should be familiar with all relevant Standards. There are four basic elements of a drawing: *points*, *straight lines*, *arcs* and *curves*. The points on the screen can be located by selecting the position with the cursor, snapping to the grid points, or numerically using the co-ordinate systems.

2.1. SCREEN COORDINATES

In order to specify the positions of points (and hence lines, polygons and other features) in a displayed image it is necessary to refer these to some system of screen coordinates (x, y). The World Coordinate Systems, the two axes used for 2D drafting are the X axis and the Y axis. The convention is for x-values to increase from left to right, and for y-values to increase from bottom to top, in the normal way. The origin of this coordinate system might be taken typically either at the bottom left corner of the screen, or at the centre of the screen (Figure 2.1).

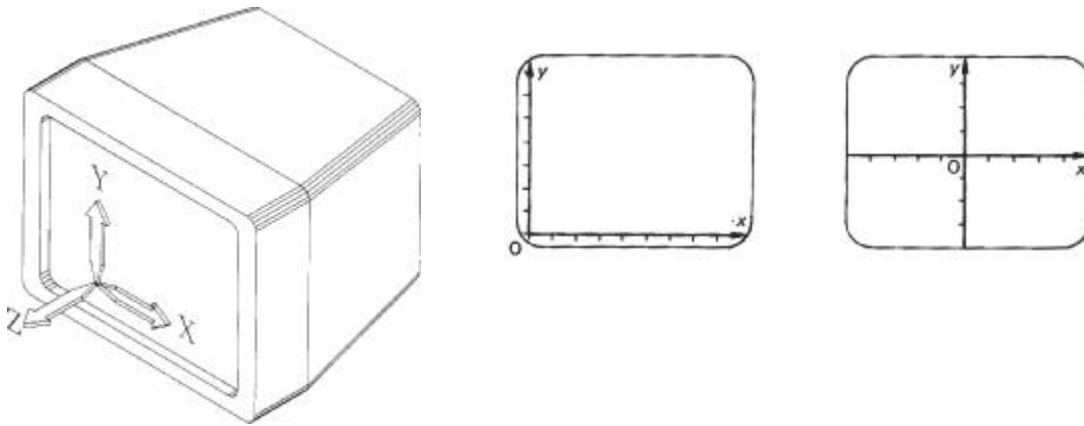


Figure 2.1 Screen coordinates

The Z axis for 3D design is perpendicular to the plane formed by the X,Y axes, extending into and out of the display (Figure 2.2). Program's default coordinate system is called the world coordinate system WCS. The world origin is referred to as simply the origin or as the 0,0 coordinate.



Figure 2.2. Determination of the Z axis

You can use the User Coordinate System (UCS) to define a new coordinate system, locate the origin, and orient the axis the way you want.

2.1.1. Using Coordinates

Cartesian (Absolute) Coordinate Systems

The coordinate discussed previously is called an *absolute* coordinate because it is expressed as a distance from the origin. When prompted for a point you can enter absolute coordinates by typing them on the command line. Type the X value of the coordinate, insert a comma, type the Y value of the coordinate, and then press Enter. The values of the X, Y coordinates of each point of the line are specified individually (Figure 2.3a).

Polar Coordinate Systems

A polar coordinate is expressed as a distance and a direction angle from the origin. Using polar coordinates, the same line would be specified in terms of the distance from the current origin together with the angle suspended. Remember we measure angles in degrees, taking positive angles to be measured counter-clock wise. To specify a polar coordinate, type the distance, followed by a left angle bracket (<) and the angle (Figure 2.3b).

Relative Coordinate Systems

Each time you type or pick a point, the new point becomes the *last point* input. AutoCAD remembers the last point, so you can easily enter new points relative to its location. *Relative coordinates* are based not on the origin, but on the last point. Alternatively relative coordinates may be used, which means that you specify the X and Y distance from the last point on the line to the next point. The @ (at sign) symbol indicates that the coordinates are relative (Figure 2.3c).

Relative Polar Coordinate Systems

Finally relative polar coordinates may be used. These specify the length and the angle of the new line segment relative to the last point on the line. To locate a point based from the last point (relative polar), precede the distance <angle with the @ symbol. @100<30 will result a new line segment of length 100 units drawn at an angle of 30° to the horizontal (Figure 2.3d).

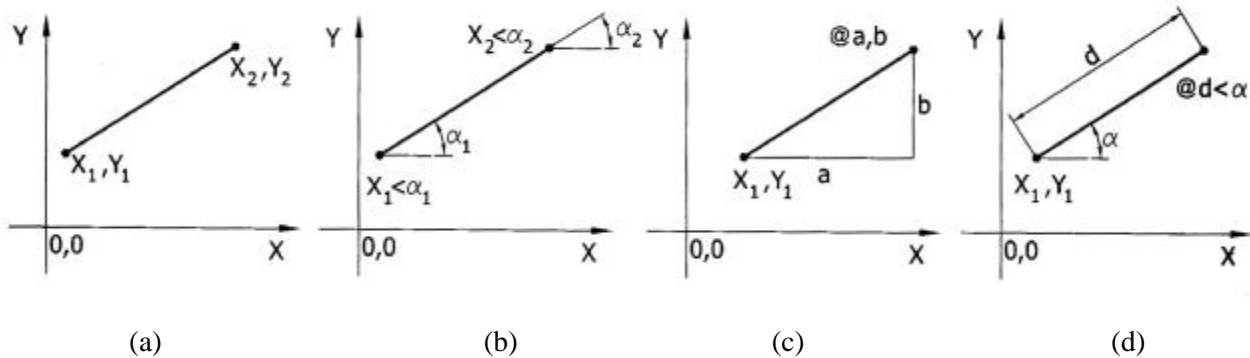


Figure 2.3. Coordinate systems

2.1.2. Angle convention

Computer graphic programs normally measure angles in degrees, taking positive angles to be measured counter-clock wise from a horizontal reference. CADD software normally measures angles in degrees, taking positive angles to be measured counter-clockwise from a horizontal reference, (see Figure 2.4).

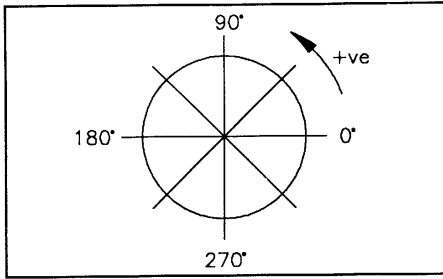


Figure 2.4. Angle convention

2.2. PARAMETRIC DESIGN

At this stage in considering the computer representation of drawings we are only concerned with the representation of the geometry. Some examples of the geometric entity types available on a popular personal computer-based CAD system are shown in Figure 2.5. These entities will normally be defined by the system in terms of numeric values for their point coordinate or other data.

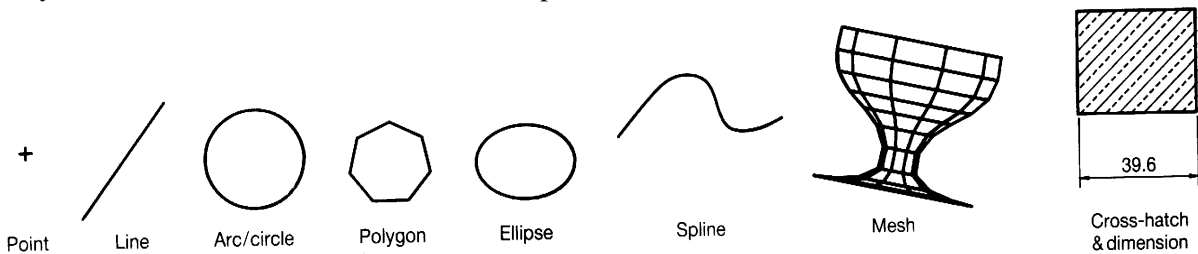


Figure 2.5. Geometric entity types available on a PC-based CAD system.

Computer-aided drawing also provides the designer with a rich variety of techniques for the definition of geometric entities. These are perhaps best illustrated by examples from a typical commercial system.

2.2.1. Geometric Parameters

The geometric parameters for a number of geometric and detail entities for 2D drawing are described below; some typical geometric entities are shown in Figure 2.6. The point entity, which is quite commonly found in 2D and 3D CADD systems, is used mainly as a reference during design synthesis. The geometric parameters are the coordinates of the point x_p, y_p for a 2D package and for a three-dimensional system this definition can be expanded to three coordinates x_p, y_p, z_p .

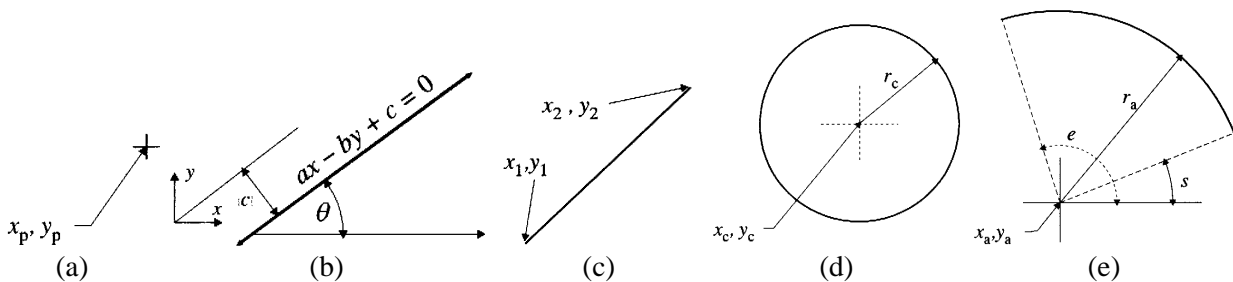


Figure 2.6. Geometric parameters for (a) point; (b) infinite line; (c) line segment; (d) circle; and (e) arc

The straight line entity can often appear in two forms, namely the *infinite line* and the *line segment*. Many CADD packages allow both types of line to be defined and others normally only define line segments. A straight line is defined by the equation:

$$a \cdot x - b \cdot y + c = 0$$

where $a = \sin \theta$ and $b = \cos \theta$

and c is the perpendicular distance from the origin (0, 0) to the line. The line segment is defined by its end points which for two-dimensional systems is four parameters x_1, y_1 and x_2, y_2 . for three-dimensional systems this becomes six parameters or three coordinates for each end point. The length of a line segment is evaluated from the equation:

$$l = \sqrt{(y_2 - y_1)^2 + (x_2 - x_1)^2}$$

and

$$a = \frac{(y_2 - y_1)}{l} \quad b = \frac{(x_2 - x_1)}{l} \quad \text{and} \quad c = b \cdot y_1 - a \cdot x_1$$

The *circle* entity is defined simply by three parameters, whereas the arc entity requires five parameters. Many CADD packages do make a distinction between circles and arcs but they can be in fact defined by one data structure. An arc is defined by the coordinates of its centre point, its radius and its start and end angles measured clockwise from the horizontal axis. A circle is simply defined by its centre coordinates and its radius. However, a circle is a closed arc where the start angle is 0° and the end angle 360° (2π radians). The start and end points of an arc can be easily calculated; the start point by

$$x_s = r_a \cdot \cos s \quad \text{and} \quad y_s = r_a \cdot \sin s$$

and the end point by $x_e = r_a \cdot \cos e$ and $y_e = r_a \cdot \sin e$

Detail entities such as text, cross-hatching and dimensions also require some geometric parameters. Cross-hatching is, in most cases, defined as a series of line segments grouped together so that they can be manipulated as an entirety. Dimensions are again a collection of entities grouped together. A typical dimension such as that shown in Figure 2.7(a) consists of two leader lines (line segments), a dimension line (one or two lines with arrow heads) and some text. Text in CADD systems needs to be defined such that it can be drawn on a plotter (Figure 2.7b).

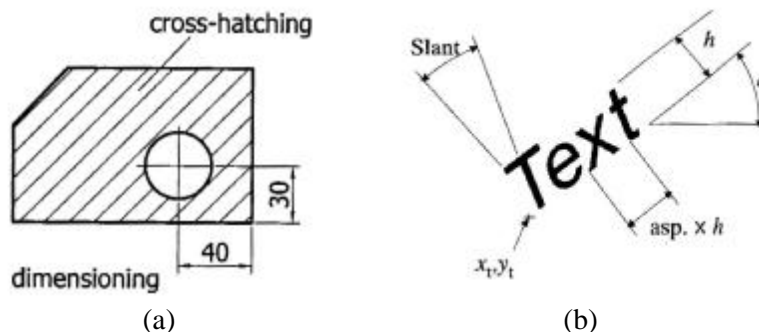


Figure 2.7. Detail entities (a) cross-hatching and dimension; and (b) text parameters

2.3. TWO-DIMENSIONAL COMPUTER DRAWING

The typical 2D computer drawing package can be segmented into eight areas of commands, each area performing a set of requirements for proper graphical representation of an object. These commands may have different names in the various 2D drawing packages.

In the early stages of the development of computer graphics, drawing was accomplished by writing a program of steps necessary to complete the drawing. In fact, it was possible to incorporate drawing commands within a BASIC, FORTRAN, or C program to generate complex drawings. Typical fundamental drawing commands to operate a simple graphic package are

MOVE (X, Y) Move from the current cursor position to the coordinates X, Y without drawing a line
DRAW (X, Y) Draw a line from the current cursor position to coordinates X, Y
TEXT (SAMPLE) Write the text "SAMPLE" beginning the current cursor position

Today, however, the user-friendly, menu-driven software makes the fundamental drawing command transparent to the user. When you ask for a circle, the menu command CIRCLE invokes the program steps to create the series of straight lines that give the appearance of a circle on the display. This interactive capability makes 2D computer drawing a straightforward task, which you can learn quickly with an understanding of fundamental geometry and drawing practices.

Setup Commands

GRID	TEXT FONT
SNAP	LINE FONT
ORTHO	UNITS

Geometry Commands

LINE	CIRCLE
RECTANGLE	ARC
POLYGON	ELLIPSE

Modification Commands

MOVE	COPY
ROTATE	MIRROR
	ARRAY

Automatic Function Commands

FILLET	HATCH
CHAMFER	DIMENSION
OFFSET	TEXT

Management Commands

ZOOM	LAYER
PAN	

Editing Commands

TRIM	ERASE
EXTEND	REDRAW (REGENERATE, REFRESH)
STRETCH	UNDELETE (UNDO)

Managing Commands

SAVE
END
QUIT

Output Commands

PLOT
PRINT

2.3.1. Geometry Creation

Geometry creation is a synthesis type function. A CADD software package should have facilities to create all the necessary geometric entities in a variety of standard line types according to a known drawing standard.

Functions should also be available to create regular shapes consisting of a number of these entities such as rectangles, polygons and polylines. Curved lines can be approximated as chains of short straight-line segments; the number of these segments then depends on the required closeness of the approximation and on the resolution of the display device (Figure 2.8).

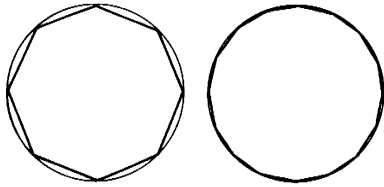


Figure 2.8. Approximation of curve by chain of straight-line segments

Points

Points may be point entities themselves, or implied points related to other entities or intersections. They may also be put in by the designer by entering a coordinate value or pointing to a position on a computer screen, in which case many systems offer the facility to generate a grid pattern in the construction plane of the system such that user-indicated positions are constrained to lie at grid points (Figure 2.8).

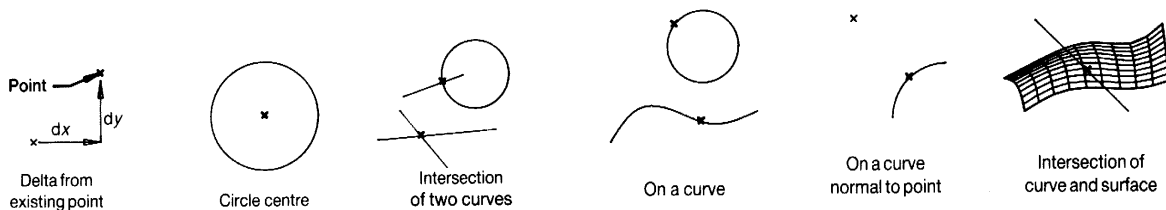


Figure 2.8 Methods for point construction.

Line Construction

One of the simplest drawing entities is the line. A straight line might be entered by specifying the coordinates of two points on the line, or by specifying the coordinates of one point together with the angle which the line makes with some other line (Figure 2.9).

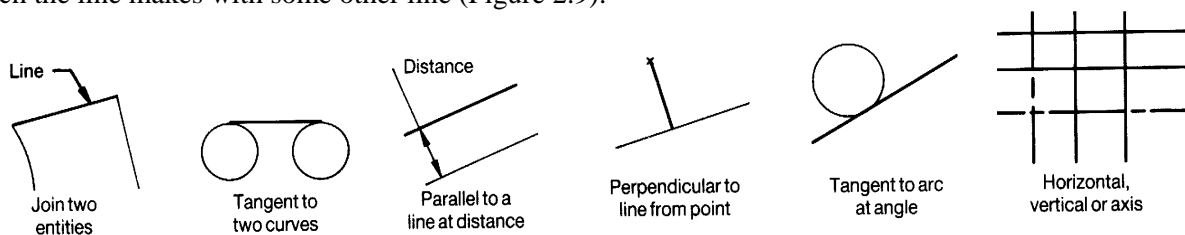


Figure 2.9 Methods for line construction.

The command line will prompt for a starting point with: **From point** To indicate the starting point you may move the cross wires to the required position with the digitizer/mouse and press the “pick” button. The command line prompts for the next point with the words: **To point** (Figure 2.10).

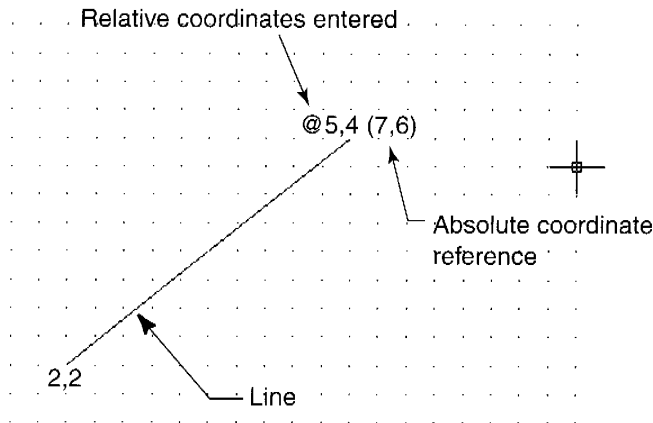


Figure 2.10 Line construction in AutoCAD

If the last segment of your line is intended to close the figure, i.e. to return to the starting point, this can be done automatically by responding with C. **To point: C**

Circle Construction

A circle might be specified by giving the coordinates of its centre point together with its radius, or by giving the coordinates of three points on its circumference (Figure 2.11).

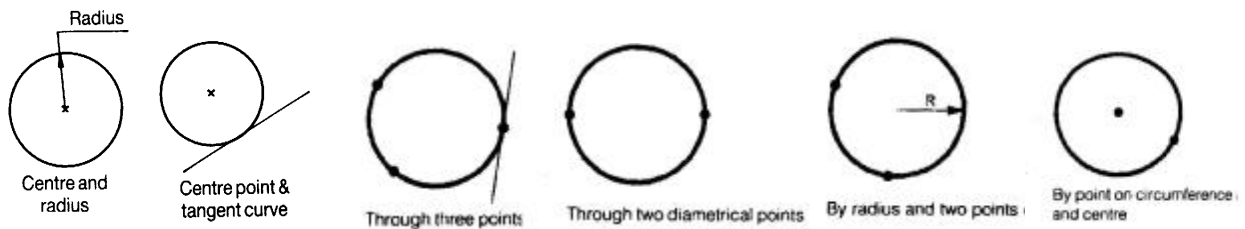


Figure 2.11 Methods for circle construction.

Arc Construction

Arcs are called up by the ARC command and the drawing technique is similar to that for circles. However, AutoCAD allows the arc to be specified in 10 different ways and this again can lead to confusion. The simplest technique is to use the 3-point method (Figure 2.12).

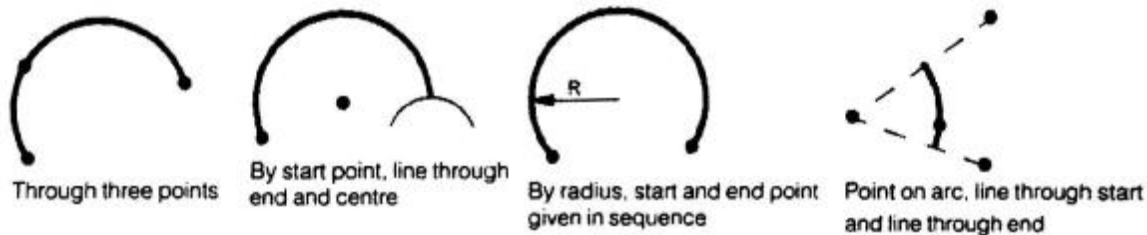


Figure 2.12 Methods for arc construction.

Ellipse Construction

The ELLIPSE command can specify an ellipse in a number of ways, as shown in Figure 2.13. The default route is to specify one axis of the ellipse by defining its two endpoints, P1 and P2. The second axis is then

defined as a distance from the centre point of this axis to P3. Alternatively, the first axis may be defined from a centre point and P2.

Instead of specifying the point P3, the “roundness” of the ellipse may be set by an angle of rotation. If this route is taken, the points P1 and P2 are considered as points on the diameter of a circle. If the angle of rotation is set at 0° then the ellipse is drawn circular. As the rotation angle increases towards 90°, the circle is assumed to be rotated, presenting an increasingly elliptical view and tending to a straight line at an angle of 90°.

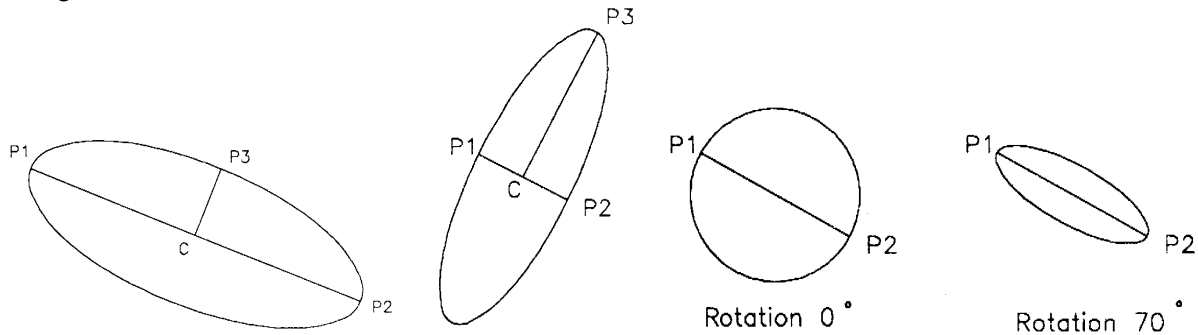


Figure 2.13. Ellipse construction

Polygon Construction

This is a useful command for drawing regular shapes with between 3 and 1024 sides. If you reply with an **I**, you will be prompted for the radius of the circle within the polygon will just fit. If you reply with a **C**, you will be prompted for the radius of the circle within the polygon will just fit around, (see Figure 2.14).

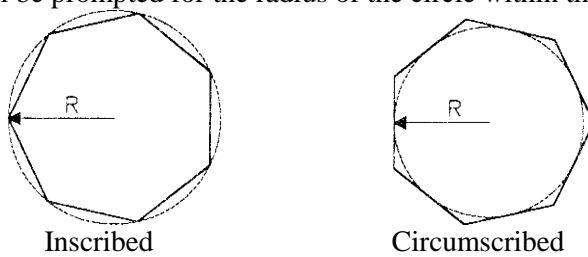


Figure 2.14 Polygon definition

Polylines

Polylines incorporate the properties of both lines and arcs. They also have a number of additional properties which include variable line width (parallel or tapered), curve fining and the construction of complex single entities composed of both lines and arcs, (see Figure 2.15).

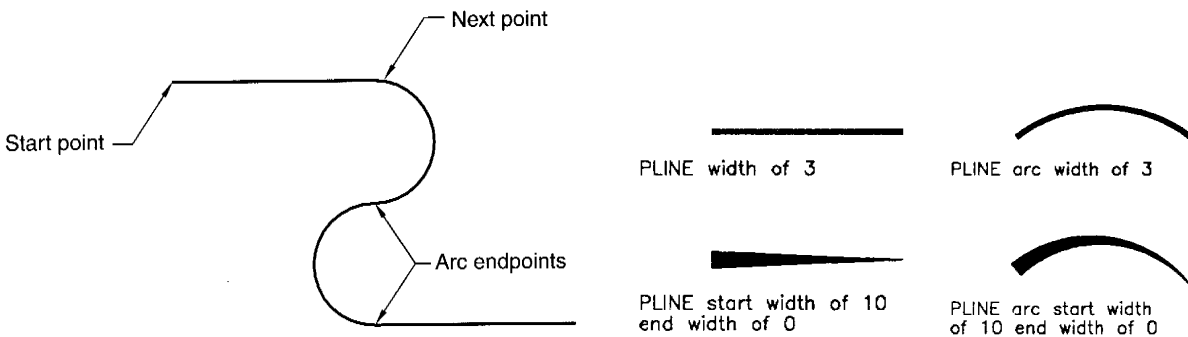


Figure 2.15 Single polyline made up of arcs and lines

Doughnut or Donut

This command will draw filled circles or rings. This last prompt is repeated indefinitely (until terminated with a return), allowing the figure to be replicated at will. If a solid doughnut (without the hole in the middle) is required, the internal diameter may be set to zero. This can be changed to an unfilled figure if the FILL command is used to switch the fill function off (see Figure 2.16).

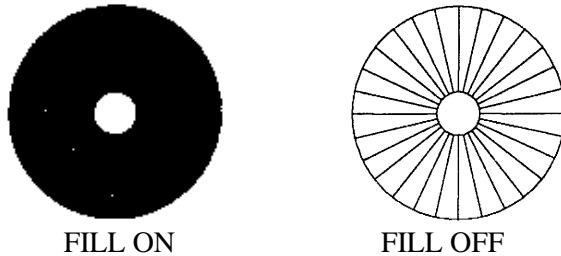


Figure 2.16 Doughnut - Fill On/Off

Trace

A LINE is considered to have zero width. The TRACE command permits the construction of lines which have a finite width. (See Figure 2.17) The command prompts repeat as with the LINE commands until terminated by a return.

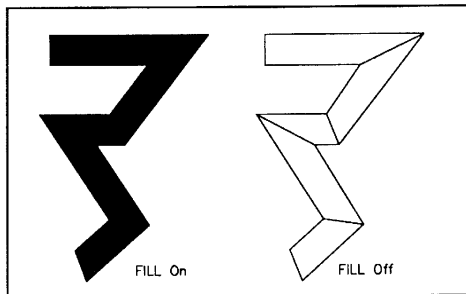
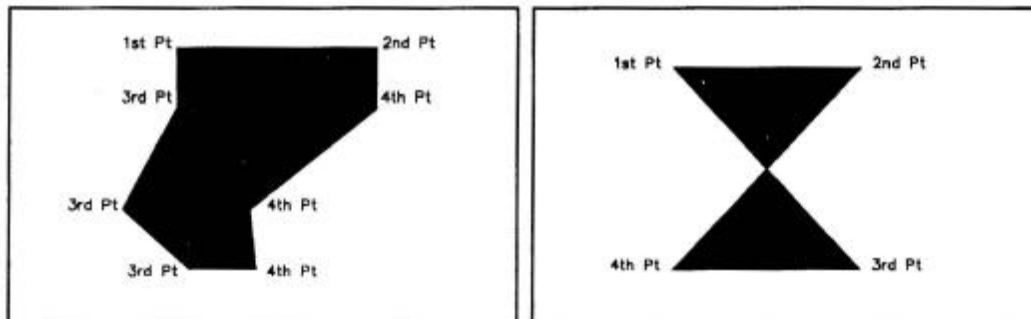


Figure 2.17 Trace - All On/Off

Solid

Solid is used to create solid filled areas composed of either 3- or 4-sided elements. If the fourth point is coincident with the third point, a triangle results; if it is a separate point, an automatic close will be executed to produce a closed four-sided solid (Figure 2.18).



(a) Solid definition

(b) Bow tie effect

Figure 2.18 Solid construction

2.3.2. Modification

The synthesis of a design has been examined as consisting of those functions which create the basic design from the designer's concept. In addition to synthesis the design can be developed further by modification which includes the erasure of entities which have been inserted by mistake or simply added for construction purposes, as well as changing the form, position, size and attitude of already designed features.

There are many useful functions used in a CAD system as shown in Figure 2.19. which are not possible in manual draughting:

- *scaling* is the capability to enlarge or diminish the size of a displayed feature without changing its shape;
- *translating* is the capability to move parts of drawing and redrawing them in the new position to a selected scale;
- *rotation* is the capability of rotating the features about a selected centre and redrawing them at the new angular position;
- *mirroring* is the capability to create the reverse image of a feature about the chosen line of symmetry;
- *duplicating* is the capability of redrawing a feature or component many times and displaying it in an orderly manner linearly or rotationally.

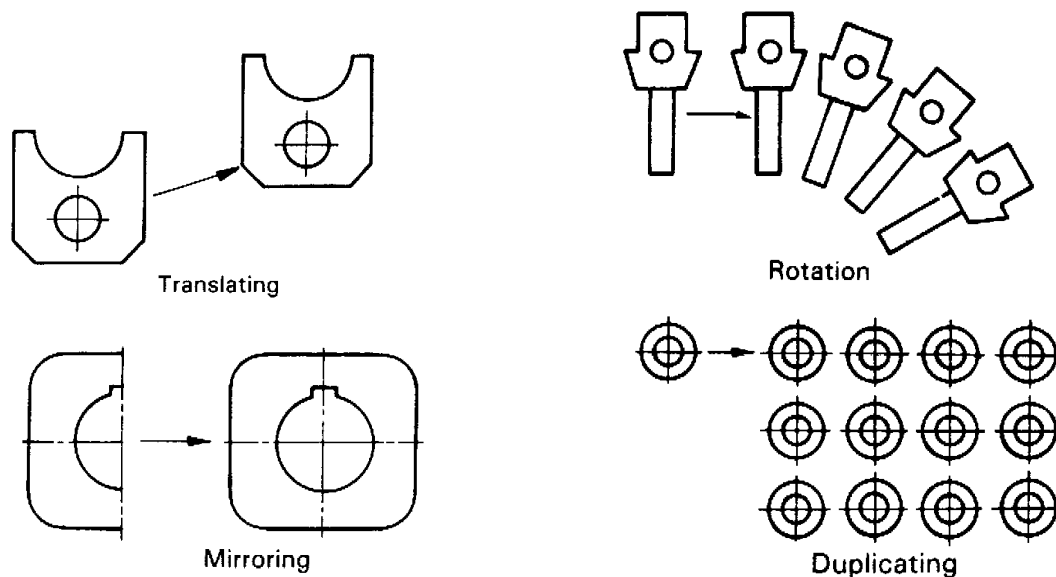


Figure 2.19. Modification functions

The MOVE command simply allows the drawing to be moved to another area of the coordinate system. This permits other drawings to be merged or added. The ROTATE command permits an entity to be reoriented by rotating through a specified angle around a base point. COPY allows like entities to be duplicated on the drawing at specified locations

2.3.3. Automatic Functions

Automatic functions are very powerful in increasing productivity as the arrangement took a fraction of the time that it would have done if each arrangement was created individually. Other features which can be

classified as modification are deletion of entities, trimming entities to different sizes using other entities and the creation of fillet radii and chamfers.

The CAD systems offer the following additional automatic facilities:

- *fillets* arcs blending together into intersecting lines or curves;
- *chamfers* surfaces created by bevelling an edge or corner;
- *cross-hatching* shading of an area defined by its boundaries;
- *dimensioning* the required features are measured by a computer and the correct dimensional value is displayed;
- *text* can be produced in any size and style and can be placed at any angle.

Fillets and Rounds

A fillet is a special kind of geometric entity that consists of an arc of a specified radius that is tangent to two lines on your sketch. Fillets are typically used to round off a square corner. Rounds are similar to fillets, except that they occur on “outside” corners, whereas fillets occur at “inside” corners. Fortunately, most modern software will automatically create the arc and perform all the trimming operations for you when you specify that you want to create a fillet on your sketch (Figure 2.20)

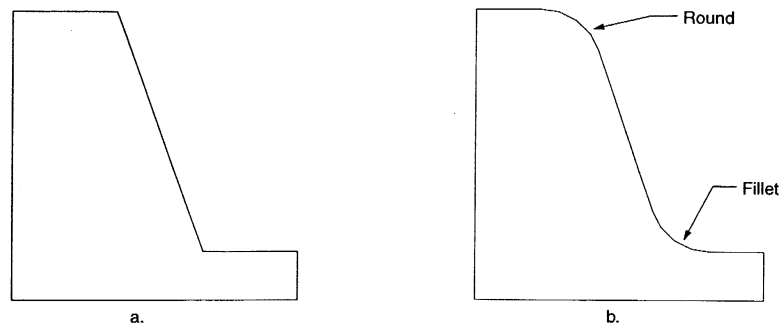


Figure 2.20. 2D fillet and round

Chamfer

Chamfer command lets you specify a distance for the chamfer lengths, which is referred to as the distance method. You can also specify an angle and distance to establish the chamfer (Figure 2.21).

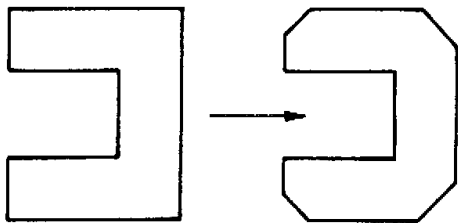


Figure 2.21. Chamfer generation

Dimensioning

The positioning tools were used to pick the end points of the dimensioned line and these points become the reference points. The position of the dimension text was indicated using the cursor and the dimension lines; arrows and text were automatically drawn. Figure 2.22 shows a linear vertical and horizontal dimension.

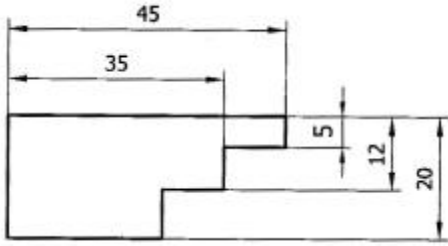


Figure 2.22 Dimensions in CADD system

Cross-Hatching

Cross-hatching needs to be automatic as the algorithm or programming procedure is relatively simple but gives significant productivity gains (Figure 2.23). Many CADD packages provide various cross-hatching styles for different applications.

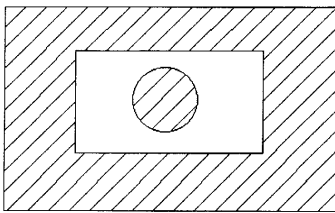


Figure 2.23. Automatic cross-hatching

Text

Text can also be produced from the text command directly for adding notes to the drawing such as the title block. It is recommended that the text entered in this manner be placed on a separate layer.

2.3.4. Management Functions

The management of a drawing or design is an important part of the process. Functions which can be classified as management functions include:

- *Display management.* Zooming, panning of views for ease of display and detailed work. The arrangement (in 3D) of view ports showing views of a model from different directions.
- *Entity attributes.* Colour and plotter pen allocation. Each entity will have its own display colour and possibly its own pen number for plotting
- *Line types.* Each geometric entity will have its own line type or style depending on its purpose in the drawing. Typical line styles include solid, centre lines (line of symmetry) and dashed (hidden lines).
- *Groups, levels or layers.* Entities can be grouped together for easier handling in modification. Entities can be placed on different layers or levels similar in concept to overhead projection slides.

Display management

Display management is perhaps the most important factor in increasing productivity during the design synthesis. The ability to zoom into some detail for creation or modification allows for greater detail and accuracy (Figure 2.24).

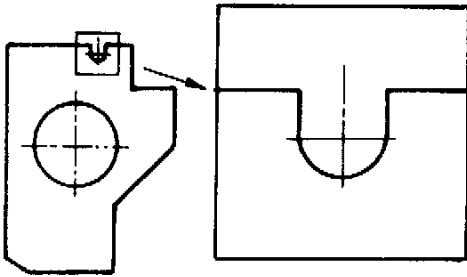


Figure 2.24. Zoom function

PAN lets you view a different portion of the drawing without changing the magnifications. It lets you see details that were off screen before panning or to reorient your view of a specific portion of the screen.

The ability to view a 3D model from many directions gives a better understanding of the design created so far. Screen set up with four viewports, three of the which show the design in the three basic planes: xy , yz and xz . The fourth viewport shows the design in isometric projection.

Entity Attributes

In addition to the parameters which define the geometry of entities a CADD package must also store, for each entity, its attributes. Examples of entity attributes are:

- colour;
- plotter pen number;

The colour attribute is that in which the entity appears on the graphics screen and is stored as a field in the record for the entity as an integer representing the colour number in the palette of the graphics system. It may also refer to the pen number which is selected when the entity is plotted, in which case the pen number attribute is not required. Some CADD systems only use the colour number for pen selection, others give the user a choice as to whether to use colour or an alternative pen numbering system.

Line Types

The line type attribute would be an integer value representing the line type style to be used for the entity. The numeric value for each line type will then be mapped to a physical type or style in the CADD system, for example solid, dashed, dotted, chain-dot or chain-dashed.

Layering

In many systems the data by which a drawing is represented are organised into a number of separate *layers*. From the user's point of view these can be thought of as a series of transparent sheets which are superimposed to make up the complete drawing (Figure 2.25). The use of *layers* or *levels* in CADD systems is common. Thus the actual outline of a component might be on one layer; construction lines on another; dimensional annotations on a third; other notes and labels on a fourth; hatching on a fifth; and so on.

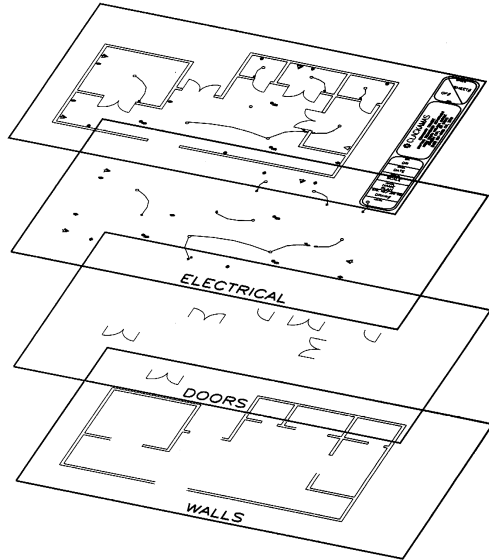


Figure 2.25. Layers in AutoCAD

2.3.5. Editing Functions

Stretching

This very useful command allows you to move a group of entities around within a drawing while at the same time stretching or compressing any lines, arc which connect the group to the rest of the drawing. An example of stretching is shown in Figure 2.26.

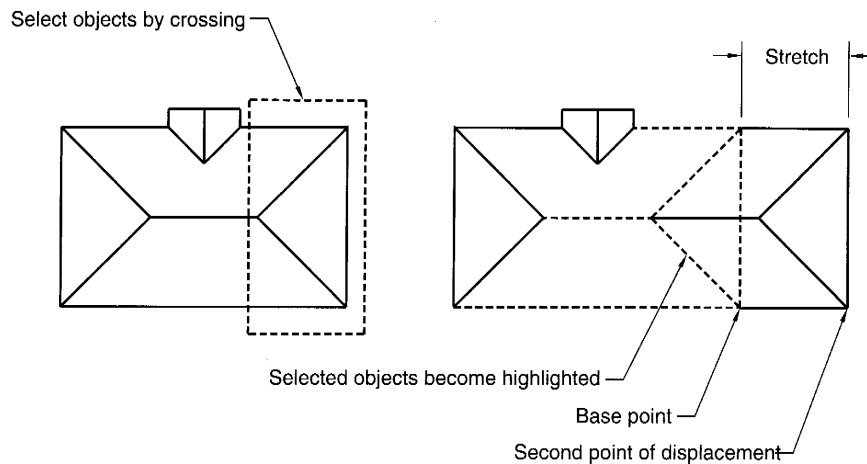


Figure 2.26 Stretching operation

Trim Operations

As you create 2-D entities on the screen, you may need to modify them in order to end up with the desired final shape. Two of the more useful operations are Trim and Extend. With Trim, you remove a portion of a given entity up to its point of intersection with another entity on the screen. Typically, software-specific peculiarities are involved in trimming circles. For example, some software packages trim circles clockwise, some counterclockwise (Figure 2.27).

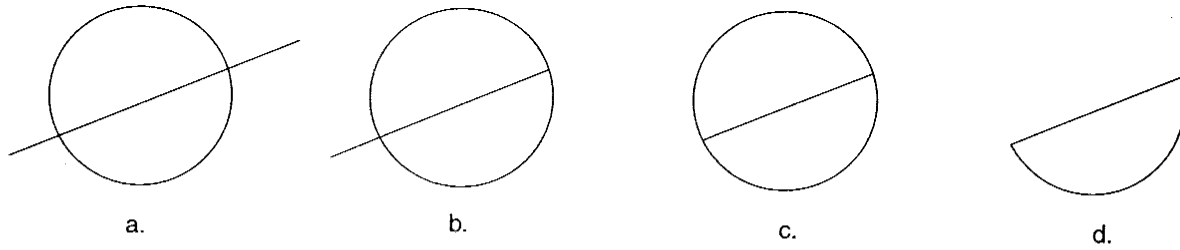


Figure 2.27 Trimming Operations

Extend Operations

Extend is used to extend geometric entities to their point of intersection with another entity on your sketch. Sometimes the point to which a line will be extended is an “implied” point of intersection (Figure 2.28).

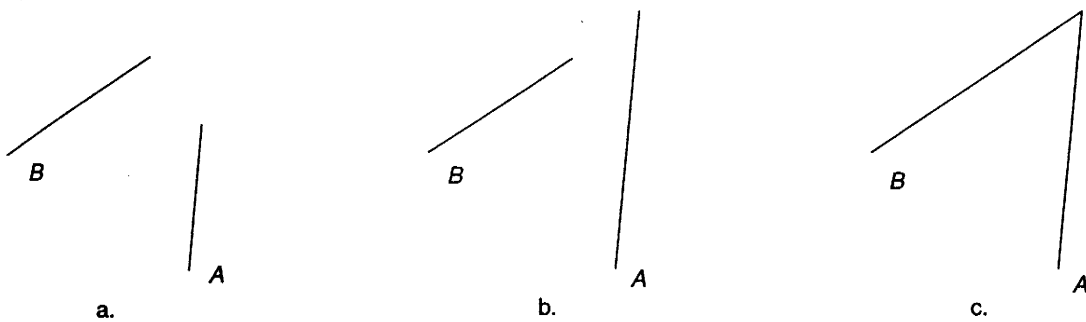


Figure 2.28 Extend Operation

Undelete (UNDO)

If you make a mistake and wish to remove the last segment of a line, it is possible to do this without exiting the LINE command. If you respond to the To point: prompt with U (for UNDO), this erases the last segment. Multiple U’s backtrack through the line segments, erasing the most recent one each time.

2.3.6. Managing Commands

Careful attention must be paid to the administrative aspects of retaining drawing files you have generated. First you should periodically save the partially completed drawing in case a network problem or power failure occurs. Saving at intervals of 15 to 20 minutes is appropriate. Before exiting your drawing program, you should be aware of the conditions of the exit commands. In some instances the exit commands will not save the file. For example, in AutoCAD, QUIT exits without saving the drawing, SAVE sends the file to disk and leaves you in the drawing editor, while END saves the file to disk and exits from the editor.

2.3.7. Output Commands







A paper record of a CAD drawing is usually kept in the form of a laser print or plotter print. The quality of laser printers has improved greatly, and very high resolution prints are now possible. Color prints are becoming popular for generating graphics for reports and presentations. The most common plotters are pen plotters and electrostatic plotters. They are used primarily to generate large drawings.

Different colours and thicknesses of line are obtained by using different pens. A plotter of this type is comparable with a vector display in that it draws line segments by moving the pen along the actual path of the required line.

2.4. DRAWING ACCURACY

Function and Control Keys

Many of the function keys act as toggle switches (On/Off) for some of AutoCAD's most useful functions. These include:

	Help		Grid On/Off
	Text window		Ortho On/Off
	Coordinates On/Off		Snap On/Off

Some of the function key commands are duplicated by control (Ctrl) keys. These are activated by holding down the Ctrl key together with the corresponding letter key.

Ctrl B	Snap On/Off	Ctrl O	Ortho On/Off
Ctrl C	Cancel	Ctrl Q	Print echo
Ctrl D	Coordinate control	Ctrl T	Tablet On/Off
Ctrl E	Isoplane left/top/right	Ctrl X	Delete keyboard errors
Ctrl G	Grid On/Off		

Grid/Snap

The SNAP command enables you to set an increment to which your cursor jumps, or snaps. In some cases, SNAP eliminates the need to type coordinates, which speeds drawing time considerably. The SNAP command restricts cursor crosshair movement to the snap increments. A grid often is more useful than the coordinate display because it provides a visual frame of reference on the drawing. A grid is a square or rectangular array of temporary dots on the drawing screen. The GRID command sets the spacing between grid dots in both the X and Y axes. The grid spacing can be set to any value, or it can be set to different spacing values for the horizontal and vertical increments.

The system then accepts the nearest grid-point to the current cursor position as the designated point (Figure 2.29). Thus points and lines are automatically 'snapped to' the grid.

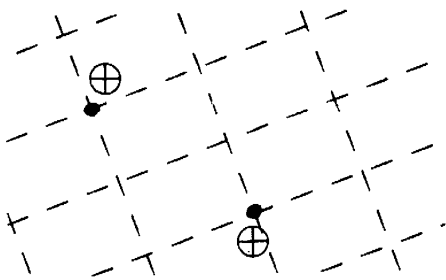


Figure 2.29 Points automatically snapped to intersections of grid

Using the Computer Mouse

The three mouse buttons have a consistent use in each of the CADD applications. By default, the left mouse button (mouse button one, or MB1) selects, the middle mouse button (MB2) accepts, and the right mouse button (MB3) provides menus. Shift+click (hold the Shift key and click) individually on entities to pick more than one entity at a time. For 2 button mouse pop-up menu appears when holding Shift key and the right mouse button (Figure 2.30).

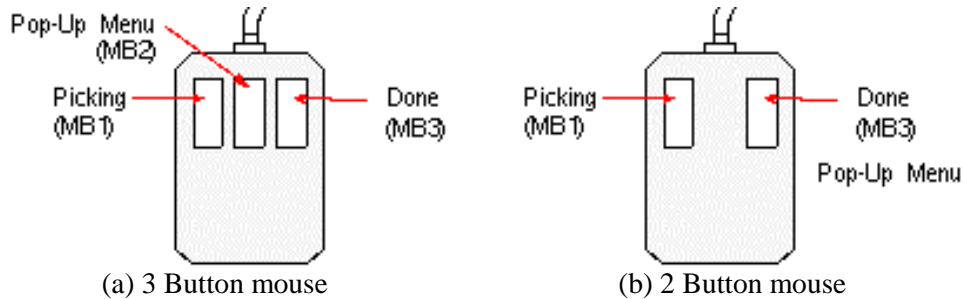


Figure 2.30 Mouse button functions

Ortho Mode

Ortho, which stands for orthogonal, enables you to constrain lines to be drawn horizontally or vertically with respect to the current snap grid. When ortho is turned on, you still can move the crosshair pointer to any location on the display and pick a point, but the rubber-band cursor extends either horizontally or vertically from the last point to one of the crosshairs (Figure 2.31)

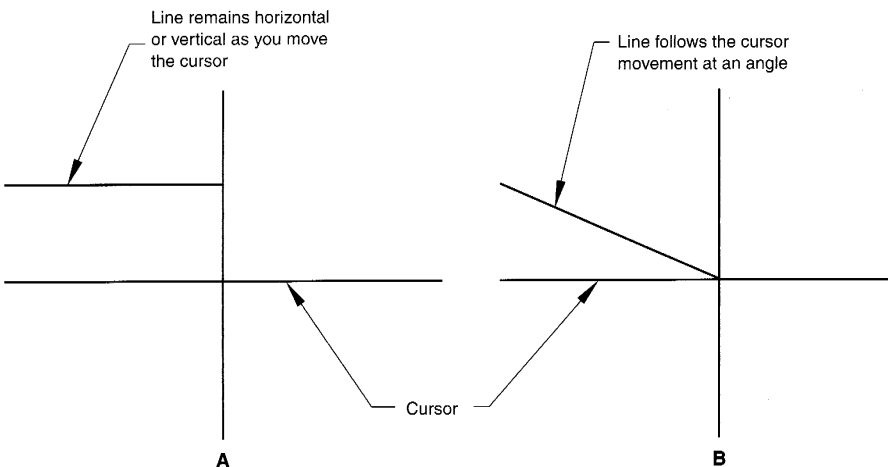


Figure 2.31 Ortho mode on/off

Drawing Limits

The idea of drawing limits is to aid you in laying out a drawing if it must conform to a particular size. AutoCAD assumes that all drawings will be constructed full size. It is therefore useful to set the limits of the drawing to a value slightly larger than the outside dimensions of the object(s) to be drawn.

Object Snaps

Object snaps allow you to quickly pick exact locations on an object without having to be directly at the desired location. Object snaps are available for almost any need. When using object snaps, an AutoSnap marker is displayed at the snap location when you move the cursor to a related object or snap point.

AutoSnap gives you a visual symbol, known as a marker, that lets you know which snap is being used (Figure 2.32).

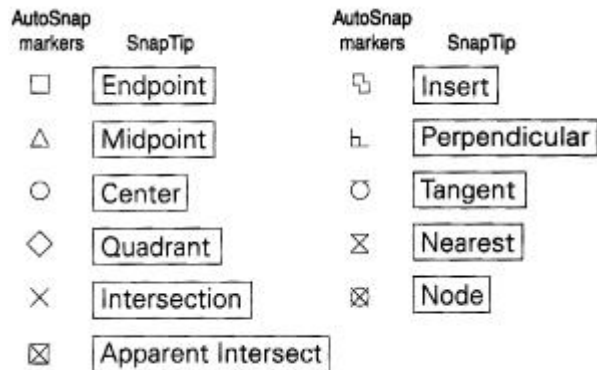


Figure 2.32 Object snaps

Selecting Objects

When editing a drawing it is necessary first to select the objects which are to be edited. AutoCAD has a number of different selection modes including:

- Pointing to a specific object
- Using one of the window commands (window or crossing; see Figure 2.)
- Selecting the last object drawn
- Selecting a set of previously selected objects.

Remove

An R at the Select objects: prompt changes the prompt to Remove objects: and allows you to remove subsequently-selected objects from the selection set. This is useful if you have included some objects by mistake.

Add

After removing objects from a selection set you can return to the default Select objects: prompt, Addition mode, by entering an A.

Undo

If an object(s) is selected by mistake its selection can be cancelled by using Undo. This may be applied repeatedly to step back and undo previous selections.

Window

Entering a W at the Select objects: prompt allows the user to use the mouse to specify two opposite corners of a window. Any objects *totally*

Crossing

A crossing box is formed at an object selection prompt in the same way as a window is formed, except that rectangle is dragged to the left rather than the right.

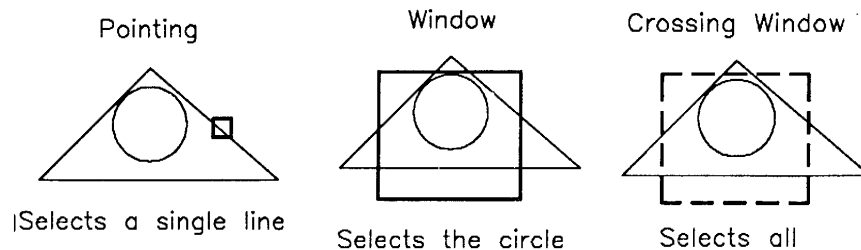


Figure 2.33 Selecting entities

Questions

1. List the curve types of a CAD system to which you have access and sketch the methods offered for the construction of each entity type.
2. Discuss how layers may be used in the organization of an engineering drawing. Can you suggest applications in which there may be particular benefit in using drawing ?
3. Name the primary text-based menu system in AutoCAD
4. What is a menu access key and how is it used ?
5. What are cascading submenus ?
6. List the four toolbars visible the first time you start AutoCAD
7. Why is it important to watch the Command window very carefully?
8. What is the purpose of the Limits setting?
9. Describe grid and snap
10. Identify three ways to access the LINE command in AutoCAD
11. What are the four coordinate entry options?
12. What is the symbol @ used for ?
13. What is the fastest way to repeat the previous command in AutoCAD?
14. What is the purpose of ORTHO ?
15. What is a selection set ?
16. Can you name three ways to access the ERASE command ?
17. What are the difference between a selection window and a crossing box ?
18. How can you remove or add objects to a selection set ?
19. Identify two ways to draw regular polygons?
20. Name the command used to draw rectangles
21. What command is used to draw filled rings?
22. How do you draw a filled circle?
23. Describe the purpose of object snap
24. List the name of at least 10 object snap options.
25. What command is used to remove a portion from an object back to another object?
26. Name the command used to make rounded corners?
27. What command is used to draw angled corners ?
28. How must a closed polyline be constructed to automatically have all corners rounded with the FILLET command?
29. What is the result of exploding a polyline ?

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