

14.4.3 Setting drawing limits

It is normal when using AutoCAD to draw objects full size, so it's usually necessary to reset the drawing limits to (about) the size of the object being drawn. Move the cursor to the bottom left of the screen, you can notice Command box. We can fix required paper size like A0, A1, A2, A3, A4 etc. from the Command box.

```
Command:
Command: 'limits
Reset Model space limits:
Specify lower left corner or [ON/OFF] <0.0000,0.0000>:
Specify upper right corner <420.0000,297.0000>: 297,210
Command:
```

Fig. 14.4

14.4.4 Erasing Objects

Removes objects from the drawing. Activate from Modify pull-down

Prompt will appear to.... select objects

Cursor changes to selection box

Ways to select objects for erasure

Pick with selection box

Create a window to select multiple objects

Type ALL to select everything visible on screen

14.4.5 Saving a Drawing File

Save

Saves drawing to current name (Quick save)

Allows user to input name if drawing has never been saved

Saveas

Allows input of a drawing name or location every time

Provides the ability to change file saving version

14.4.6 Exiting an AutoCAD Session

Close

- Closes the drawing but does not leave the software

Exit

- Closes the drawing AND leaves the software

Note: Both will give an extra chance to save the changes in your work

14.5 The Coordinate System

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

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

- Cartesian coordinates
- Polar coordinates

14.5.1 Cartesian Coordinates

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

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

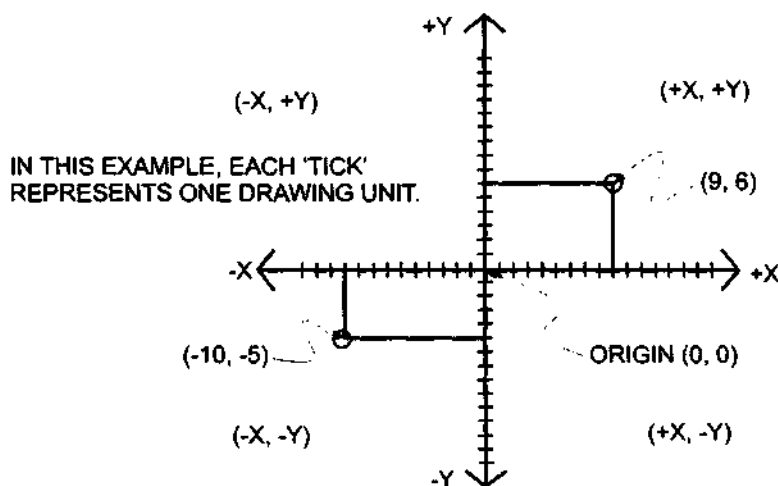


Fig. 14.5

14.5.2 Polar Coordinates

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

14.6 The Formats to Enter Coordinates

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

- Absolute
- Relative

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

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

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

Examples

Cartesian Coordinates

- Sounds like math, and it is exactly the same as in math
- Input as either Absolute or Relative Coordinates
 - Absolute *X,Y*
 - Relative *@X,Y*

Polar Coordinates (Vector Coordinates)

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

14.6.1 User-Defined Coordinate System

CADD allows you to create a user-defined coordinate system that can help simplify drawing. When you need to work with a complex drawing that has many odd angles this mode of measurement is very useful.

Let's say you need to draw or modify an odd-shaped diagram, it is very difficult to use Cartesian or polar coordinates because they would involve extensive calculations. In this case, you can create a custom coordinate system that aligns with the odd angles in the diagram.

To define a new coordinate system, you need to specify where you want the origin point and the direction of the X and Y-axis. The computer keeps working according to this customized coordinate system unless you set it back to normal

The user-defined coordinate system is especially helpful when you are working with 3D. In a 3D drawing, you need to define each point with three coordinates and work with various surfaces of a 3D model. The user-defined coordinate system allows you to align coordinates with a specific surface.

NOTE: At the bottom left of the AutoCAD is the coordinate display.

Move your cursor around the drawing area and watch the coordinate change

Function Keys

- F1 - AutoCAD Help Screens
- F2 - Toggle Text/Graphics Screen
- F3 - OSNAP On/Off
- F4 - Toggle Tablet Modes On/Off
- F5 - Toggle Isoplanes Modes On/Off
- F6 - Toggle Coordinates Modes On/Off
 - Coordinates has 2 modes when in a drawing command
 - *X,Y coordinates*
 - *Polar coordinates (Distance<Angle)*
- F7 - Toggle Grid Modes On/Off
- F8 - Toggle Ortho Modes On/Off
- F9 - Toggle Snap Modes On/Off
- 4F10 - Toggle Polar Modes On/Off
- F11 - Toggle Object Snap Tracking Modes On/Off

Definitions :

- Click
 - Press once and release
 - Also commonly used to refer to left-click
- Left-click
 - Press left-mouse button (LMB) once and release
 - Commonly used to pick or choose an item
- Right-click
 - Press right-mouse button (RMB) once and release
 - Commonly used to access pop-up menu
- Double-click
 - Commonly referred to clicking left mouse button twice
- Click-and-drag
 - Commonly referred to pressing left mouse button (and *not* releasing it) and move the mouse as required
 - The left mouse button is released to finish the command

14.7 Choosing Commands in AutoCAD

Methods of choosing commands or to execute the command

- Pick from pull-down menu
- Select from Toolbar
- Type command on Command Prompt Line

14.7.1 Pull-down Menus [pd menu] (Fig 14.6)

Select pull-down through left mouse button

Move mouse to command and click left button to select command

– Example:

- To draw straight line,
- [pd menu] > Design > Line

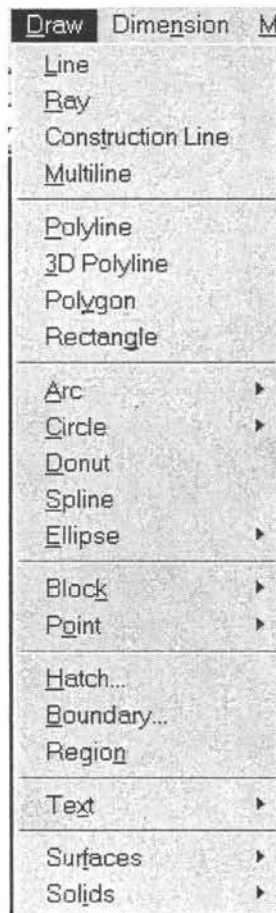


Fig. 14.6

14.7.2 Tool Bar Selection

Use mouse to track over Toolbar image for button detection

Left mouse button click will select command

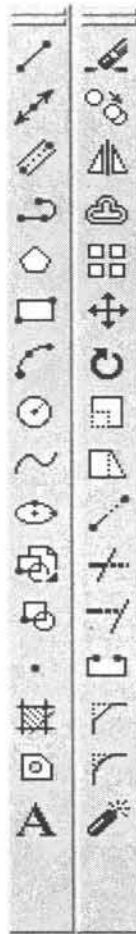


Fig. 14.7

14.7.3 Activating Tool Bars

Right mouse click on any toolbar button will bring up the toolbar activation menu

Pick toolbar menu to activate it

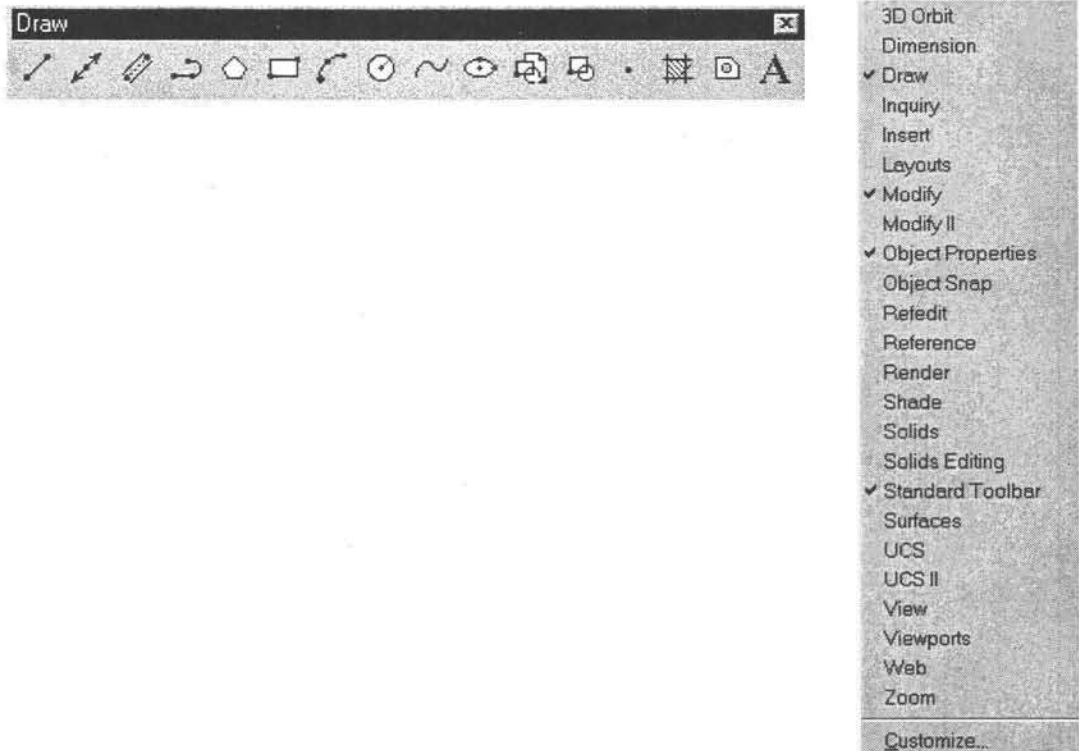


Fig. 14.9

Command Prompt Line (Fig. 14.9)

Located at bottom of screen

Left click in command prompt area and type in command

Press Enter key to input command

To draw a straight line type in

- Command: line {enter}

```
Command: line
Specify first point:
```

- You just have to type in the word line and press the key Enter (on the keyboard)
- The command is not case sensitive

14.8 Right Mouse Clicking

- It will get you everywhere!

Command and area specific menus will appear upon right mouse clicking

The default Right mouse click menu is shown on the right

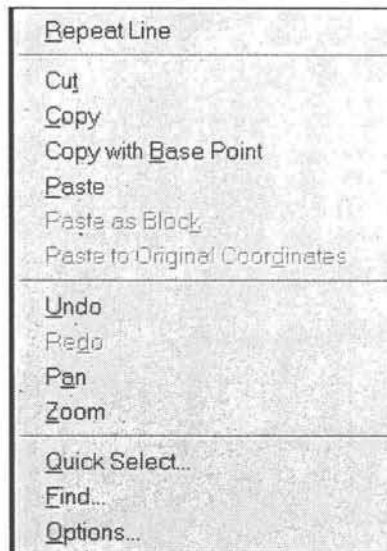


Fig. 14.10

14.8.1 Right Mouse Click Menus

Short-Cut menus will appear within a command (All commands)

The Default editing menu appears on the right

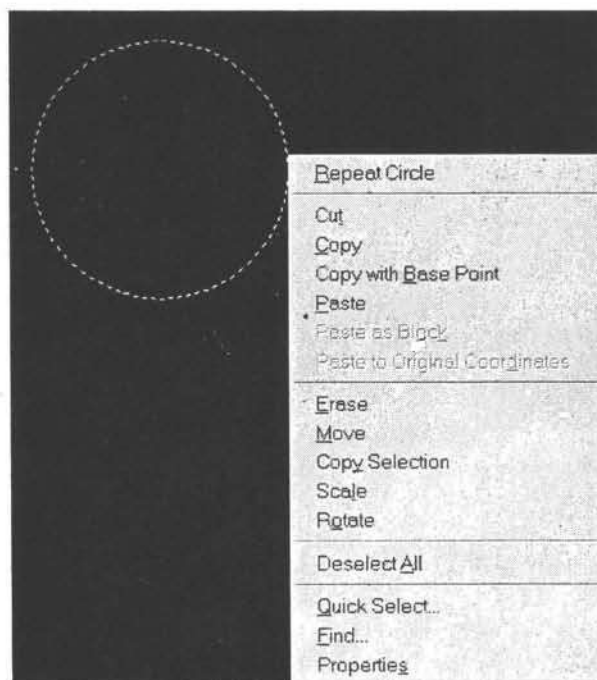


Fig. 14.11

14.9 Object Snaps

Uses geometry to location specific positions

- Activation of Object Snaps (one time use only)
 - *Typing the first 3 letters of the object snap name*
 - *Holding down the shift key while clicking the right mouse button*

14.9.1 Types of Object Snaps

Center

Center of circle or arc

Endpoint

End of line or arc

Extension

Extends lines & arcs by a temporary path

From

Must be used with another Object snap to establish a reference point

Insert

Locates an insertion point of an object

Intersection

Finds the common intersection point between 2 objects

Extended Intersection

Locates the intersection between objects that do not touch

Midpoint

Locates the middle of arcs & lines

Node

Snaps to a point

Parallel

Assists in constructing a line parallel to another

Perpendicular

Snaps to an angle of 90° to the selected object

Quadrant

Finds the 0° , 90° , 180° , and 270° locations about a circle

Tangent

Assists in creating lines, arcs or circles tangent to another object

Deferred Tangent

Occurs when multiple tangent selections are needed to complete a task

[ex. Drawing a line tangent to 2 circles requires 2 tangent picks, one for each circle. the first tangent selection is a deferred selection (line does not appear) until both tangents have been selected]

Object Snap Tracking

- *Allows the user to select more than one object snap location to determine a specific position. [ex. Use for finding the center of a rectangle in conjunction with the midpoint object snap]*

14.9.2 Running Object Snaps

The object snaps that never stop working for you

Activate from Tools pull-down, drafting settings

Right click on OSNAP button on status bar

On/Off by OSNAP button

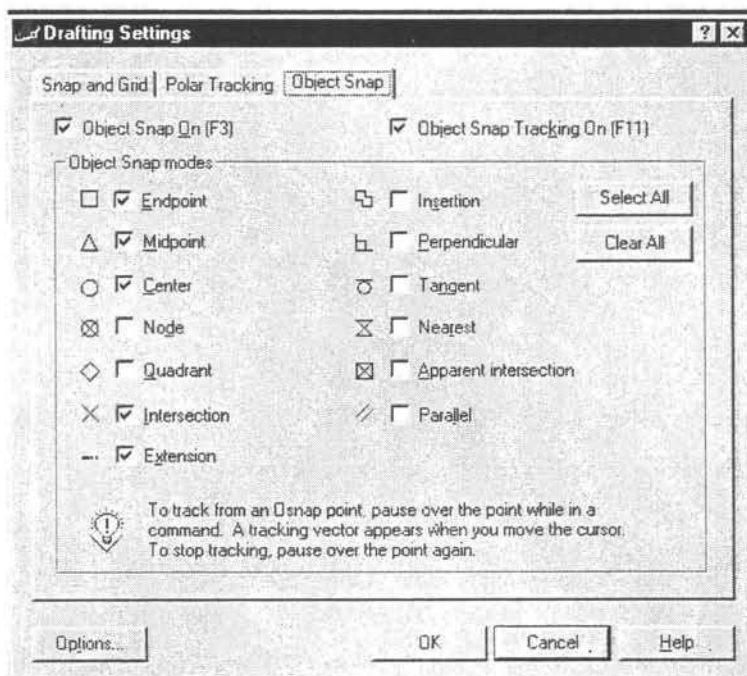


Fig. 14.12