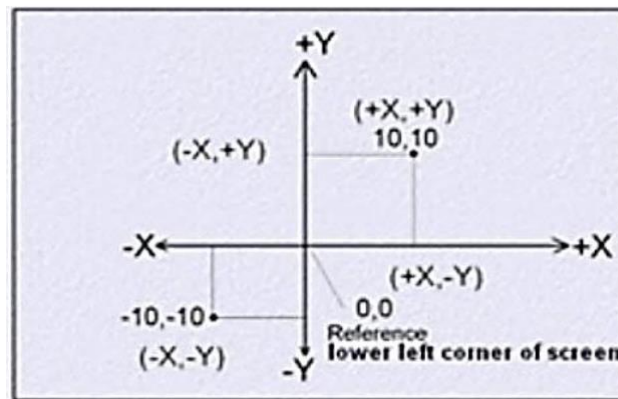
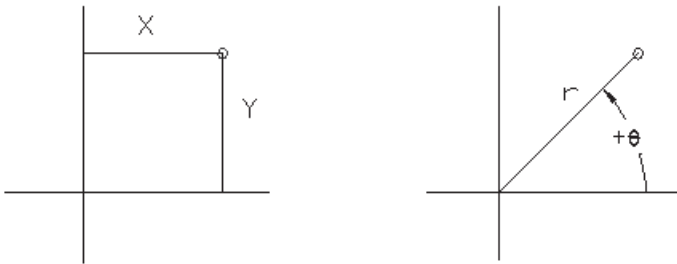
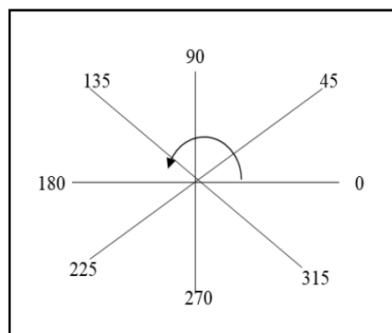


2.1 Cartesian and Polar Coordinate Systems

In a two-dimensional space, a point can be represented using different coordinate systems. The point can be located, using a Cartesian coordinate system, as X and Y units away from the origin. The same point can also be located using the polar coordinate system, as r and θ units away from the origin.



For planar geometry, the polar coordinate system is very useful for certain applications. In the polar coordinate system, points are defined in terms of a radial distance, r, from the origin and an angle θ between the direction of r and the positive X axis. The default system for measuring angles in AutoCAD® 2014 defines positive angular values as counter-clockwise from the positive X-axis.



2.2 Absolute and Relative Coordinates

AutoCAD allows us to use absolute and relative coordinates to quickly construct objects. Absolute coordinate values are measured from the current coordinate system's to origin point. Relative coordinate values are specified in relation to previous coordinates.

2.3 Method of Entering Coordinate

In AutoCAD, there are five methods for specifying the locations of points

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

For example:

1. (Absolute Cartesian Coordinate). Example: 7,3
2. (Relative Cartesian Coordinate). Example :@7,3
3. (Relative Polar Coordinate). Example : @6<45

2.4 Basic Objects in AutoCAD

Draw tool bar used to access AutoCAD draw commands or type the command in command line. The draw commands create objects such as lines, rectangles, circles, arcs, and ellipses. An object is the smallest component of the drawing. A drawing is made up of combinations of these objects. Drawing commands can be entered from the keyboard (command line), the draw toolbar, or the draw pull down menu.

1. Line

Draw Line simple lines or series of lines segments .access to command line by one way of the following:

1. **Command Line: Line or L**
2. **Menu Bar :Draw → Line**
3. **Draw Bar**

When execute command line the program requires specify first point, display in command line:

Specify first Point:

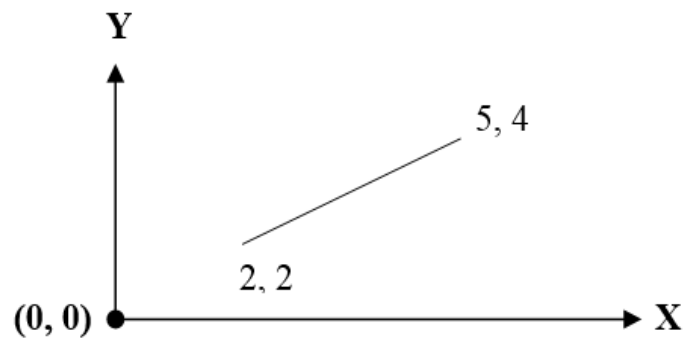
Specify next point or [Undo]:

Specify next point or [Close/Undo]:

Options:

- **C:** this option close series of lines, connect first point with last point by line.
- **U:** enter U for undo. AutoCAD backs up one segment, undoing it so that you can recreate it.

Example (2-1):



1. Draw Line using Absolute Cartesian Coordinate System.

Command line: Line

Specify start Point: 2, 2

Specify next Point: 5, 4

2. Draw Line using Relative Cartesian Coordinate System.

Command line: Line

Specify start Point: 2, 2

Specify next Point: @3, 2

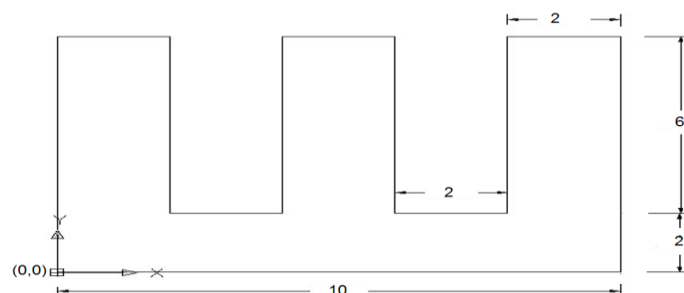
3. Draw Line using Relative Polar Coordinate System

Command line: Line

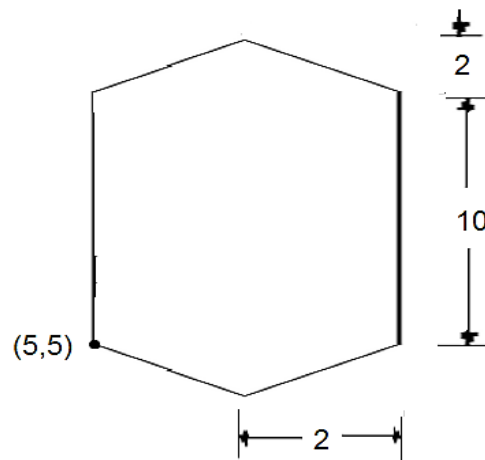
Specify start Point: 2, 2

Specify next Point: @3.60<33.69

Example (2-2): Draw the following figure, note the figure start from the original point (0,0).



Homework (2-1): Draw the following figure in board its sides (25×25mm), note the figure is start from the point (5,5).



2. Rectangle

AutoCAD Provides the Rectangle command, which allows creating rectangles with perfect corners and as a single object. Access the command by:

1. **Command Line:** *Rectangle, Rec.*
2. **Menu Bar:** *Draw* □ *Rectangle.*
3. **Draw Bar.**

When enter command, the program require specify first corner:

Specify first corner point or [Chamfer/ Elevation / Fillet / Thickness / Width]:

Specify other corner Point:

Options:

C: the chamfer command enables to place a chamfer at the rectangle corner. The distance specify is the distance from the intersection of the two lines (the corner) to the start of the bevel, or chamfer. You can set the chamfer distance for the two lines independently.

Specify first chamfer distance for rectangles<0.000>:

Specify second chamfer distance for rectangles<0.000>:

E: identify high of rectangle (this option use in 3D drawings).

Specify the elevation for rectangles<0.000>:

F: the fillet command creates both fillets and rounds on any combination of two lines, arcs, or circles.

Specify fillet radius for rectangles<0.000>:

T: identify thick of rectangle (this option use in 3D drawings)

Specify thickness rectangles<0.000>:

W: rectangle can have a width (the default width is 0).

Specify line width for rectangles<0.000>:

Example (2-3): Draw the following figure. Use rectangle command



Example (2-4): Draw the Following figure in board its' sides are (15×15 cm), note the figure is start from the point (1,1).

