



Compare between Methods of Entering Coordinate System (Absolute, Relative, Polar) in AutoCAD

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ABSTRACT

Learning to use a CAD system is similar to learning a new language. It is necessary to begin with the basic alphabet and learn how to use it correctly and effectively through practice. This will require learning some new concepts and skills as well as learning a different vocabulary. Today, the majority of the Mechanical CAD systems are capable of creating three-dimensional solid models. Nonetheless, all CAD systems create designs using basic geometric entities and many of the constructions used in technical designs are based upon two-dimensional planar geometry. The method and number of operations that are required to accomplish the basic planar constructions are different from one system to another [1]. In this work we Draw the Bathroom floor plan on board its' sides are (12, 9ft), using AutoCAD objects line, circle, rectangle, arc and ellipse, apply the three methods of coordinate system and compare between them: Absolute Coordinate System, Relative Coordinate System and Polar Coordinate System. The results show that using absolute coordinate system, enter the points as they relate to the origin of the world coordinate system. When the relative coordinate entry method allows to enter points in reference to the last point picked. And Using polar coordinate entry if wants to draw a line certain distance at a particular angle.

Keywords: Cartesian Coordinate System; Absolut; Relative; Polar; Draw Bar; Modify Bar

1. INTRODUCTION

AutoCAD provides countless methods and tools for producing, viewing, and editing 2-dimensional drawings and 3- dimensional models. The software permits designers, drafters, engineers, and others to create , revise, model, and document industrial parts and assemblies for prototyping, model making, and manufacturing around the world, organizations also use AutoCAD for the design of maps, buildings, bridges, factories, and about every product imaginable, ranging from car parts and stereo equipment to snow skis and cellular phones [2-4].

2. AUTOCAD INTERFACE

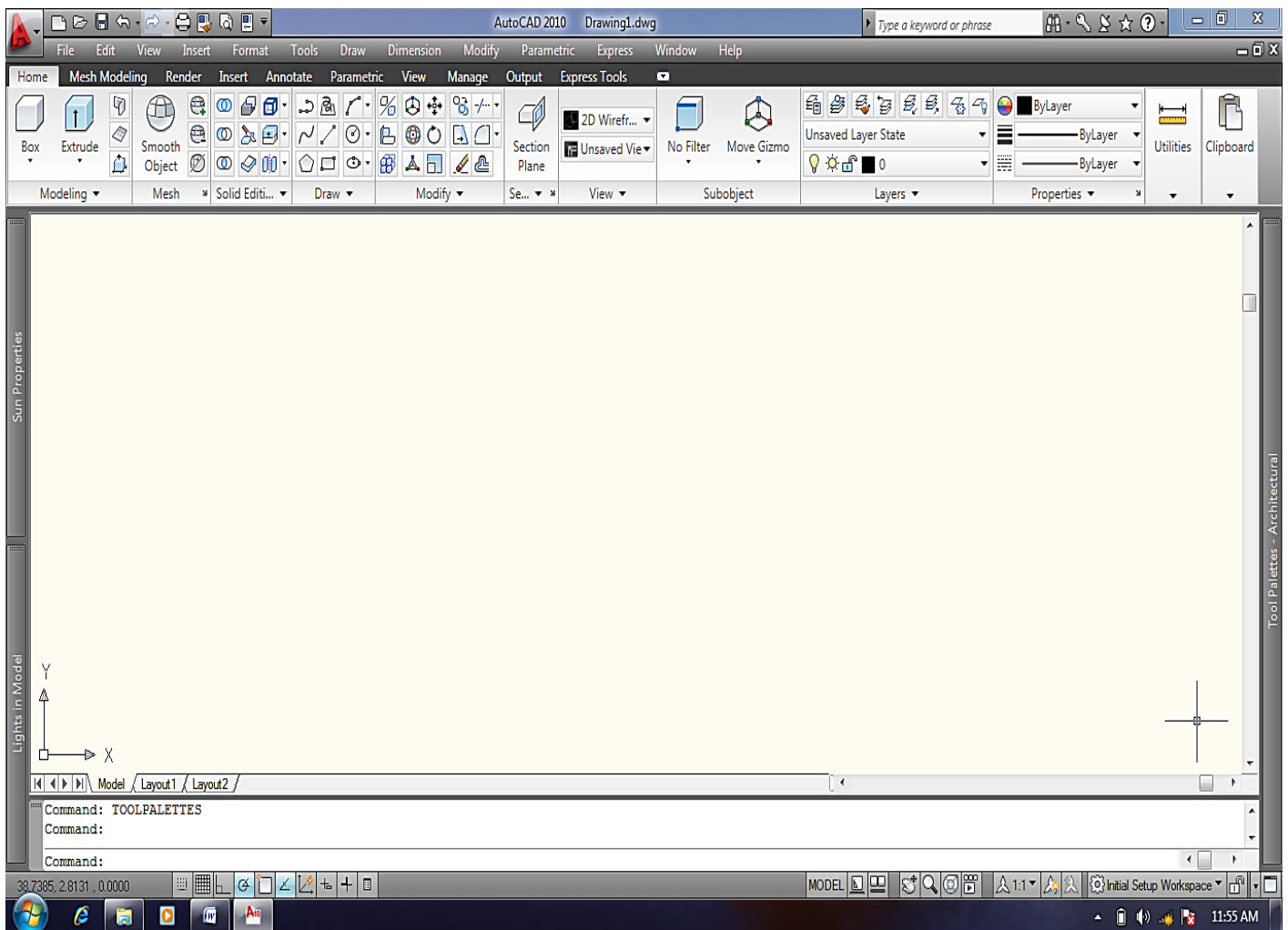


Figure 1. Autocad Screen.

The AutoCAD screen is divided into six distinct areas:

- Title bar
- Menu bar

- Toolbars
- Document window or drawing area
- Command window
- Status bar

Figure 1. Shows the typical layout of the AutoCAD screen [5].

3. THE STATUS BAR

The Status Bar is the area below the command line that shows messages as well as coordinates, modes, and the current time. To activate SNAP, GRID, ORTHO, OSNAP, OTRACK, POLAR, DYN, LWT, and MODEL you must double-click on the mode to change. Figure 2 shows the status bar in AutoCAD system [6].



Figure 2. The Status Bar.

3. 1. Alignment Grid

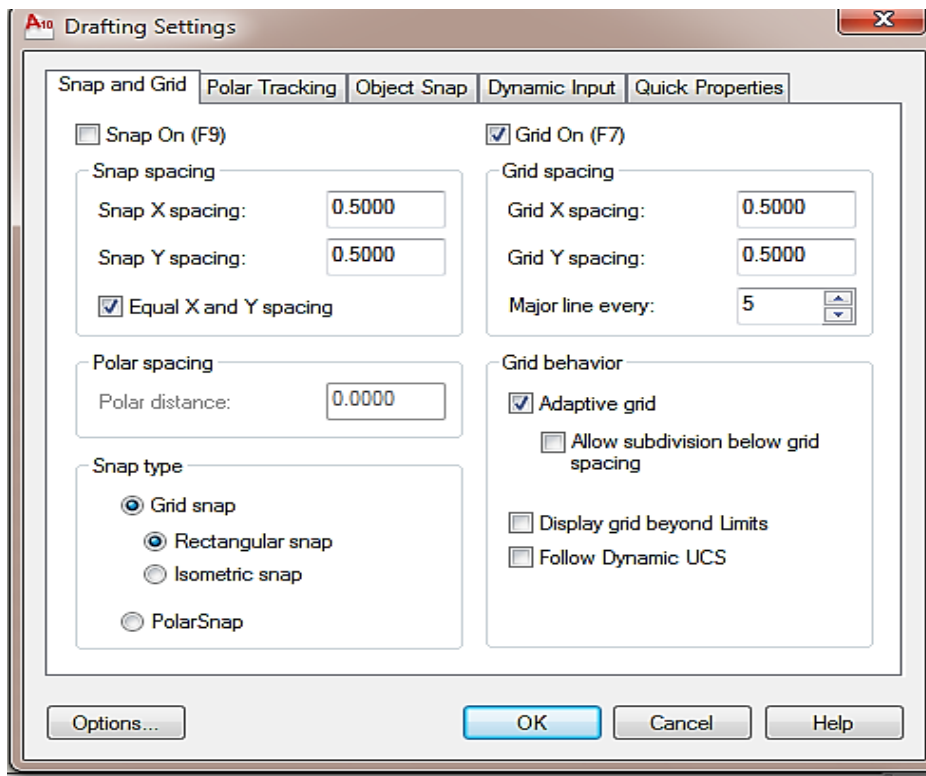




Figure 3. Drafting Setting Dialog Box.


The grid command allows you to set an alignment grid of dots of any desired spacing, making it easier to visualize distances and drawing size. you can turn the grid on and off by pick the grid button  located in the status bar or (by pressing Ctrl + g or the F7 function key). The drafting setting dialog box permits you to review and make changes to the grid settings as shown in Figure 3:

- *Menu bar* ⇒ *Tools* ⇒ *drafting setting*


3. 2. Snap grid

The snap grid is similar to the visual grid, but it is an invisible one. You cannot see the snap feature, but you can see the effects of it as you move the crosshairs. It is like a set of invisible magnetic points. The crosshairs jump from point to point as you move the pointing device. This allows you to layout drawings quickly, yet you have the freedom to toggle snap off at any time. Pick the snap button  in the status bar to turn on the snap grid, (or pressing Ctrl + b or the F9 function key).

3. 3. The Ortho Mode

Ortho short for orthogonal, allows you to draw horizontal or vertical lines quickly and easily. Ortho is on when the Ortho button  on the status bar is depressed. You can toggle Ortho on and off by clicking the Ortho button. (or press F8 function key).

3. 4. Polar

Polar makes it easy to draw lines at regular angular increments, such as 30, 45, or 90 degree. Using the F10 key or polar button  toggles polar tracking on or off.

3. 5. Object Snap (Osnap)

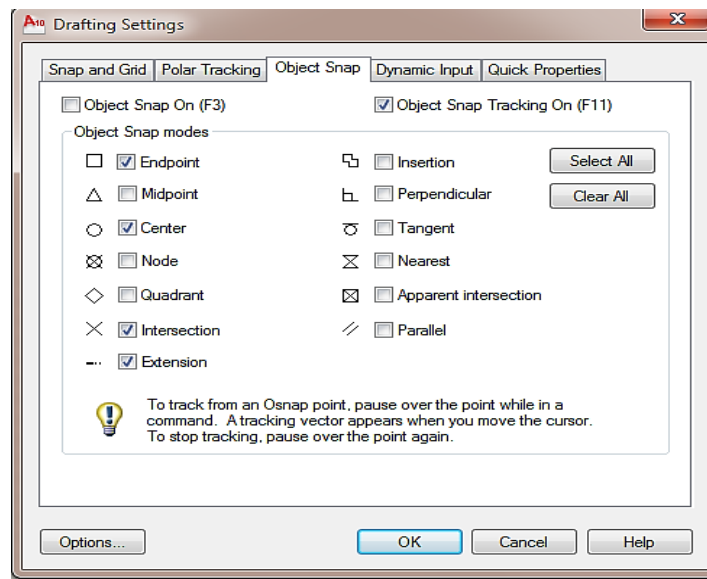




Figure 4. Object Snap Mode.

AutoCAD provides a capability called "object snap" , or OSNAP for short, that enables you to "snap" to existing object end points, midpoints, centers, intersections, etc. available object snap mode are illustrated in Figure 4.

3. 6. Object Snap Tracking (Otrack)

Object snap tracking is active when the alignment paths appear from one or more acquired object snaps. This feature is a part of AutoCAD's Auto tracking. You can toggle Auto Tracking on and off with the Otrack button  on the status bar or by toggling F11.

3. 7. Units

The units command open dialog box to determine measurement units for drawing angles and direction and precision as shown in figure 5. Access the command by:

- **Menu bar:** *format* ⇒ *units*

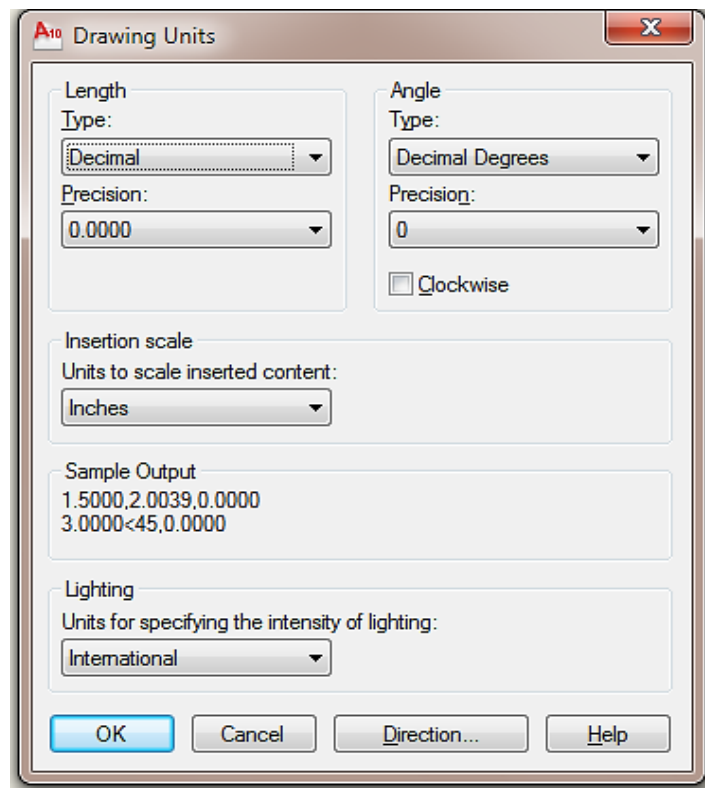


Figure 5. Drawing Units Dialog Box.

- **Length type:** determine length types (scientific, decimal, engineering, architectural, and fractional).
- **Angle type.** Determine angle type. (The default decimal degree).
- **Precision.** determine accuracy of angles and lengths. Precision is the number of places to the right of the decimal to display.
- **Clock wise** control angle direction.

3. 8. Limits

Limits used to determine the limits of board. To access the command by: [3,4].

- *Menu bar* ⇒ *format* ⇒ *Drawing limits*
Specify lower left corner or [on / off] <0.0000, 0.0000>:
Specify upper right corner <420,297>:

4. CARTESIAN AND POLAR COORDINATE SYSTEM

AutoCAD provides the user with an infinite two dimensional area to work with. Any entities placed on the working two dimensional plane can be defined relative to the Cartesian coordinate system. The Cartesian coordinate system divides a two dimensional plane with two perpendicular axes. The X axis runs horizontal across the bottom of the screen. The Y axis runs vertically along the left side of the screen. These two axes intersect at the bottom left corner of the screen. Each of these axes is further divided into segments. Each segment is given a value. The X axis segments increase in value to the right. The positive X values are to the right of the intersection of the two axes. The negative X values are to the left. The positive Y values are above the intersection and increase up. The negative Y values are below as illustrated in Figure 6 [7-9]. Polar coordinates use a distance and an angle to locate a point. With both Cartesian and polar coordinates, you can enter absolute coordinates based on the origin (0,0), or relative coordinates based on the last point specified.

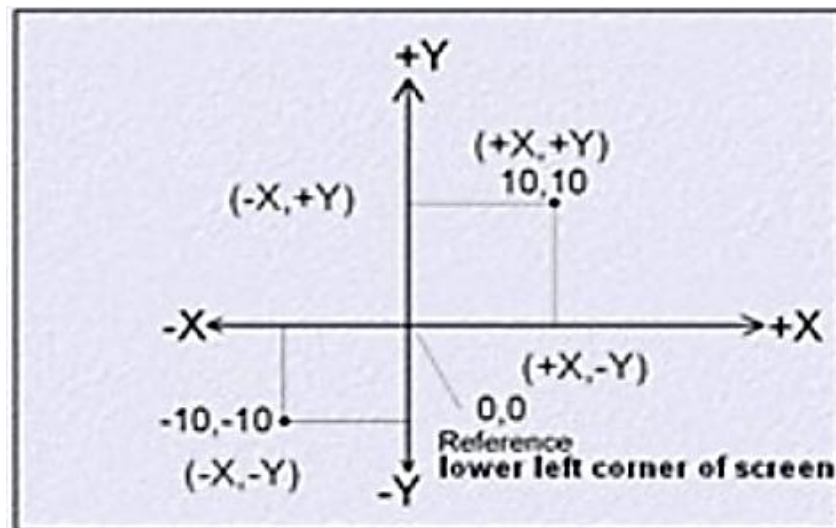


Figure 6. Cartesian Coordinate System.

5. ANGLES IN AUTOCAD PROGRAMS

Angles measure sets the direction for angle 0 .east (x positive) is the AutoCAD default. (Select clock wise from units dialog box if you want to change the AutoCAD default setting for measuring angles). Figure 7 illustrate angle's direction [3,4].

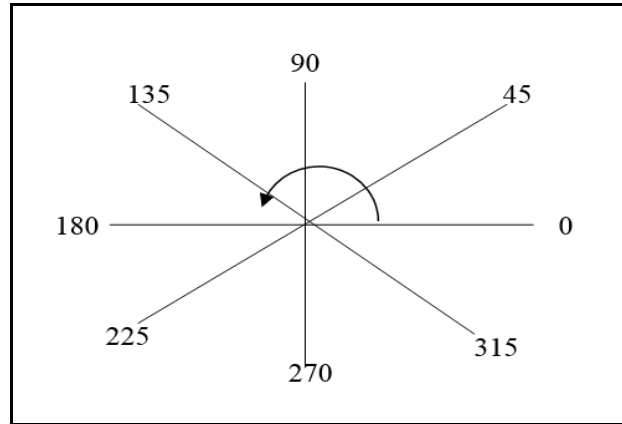


Figure 7. Angles Direction.

6. METHOD OF ENTERING COORDINATE

1. (Absolute Cartesian coordinate). Example: 7,3
2. (Relative Cartesian coordinate). Example: @7,3
3. (Absolute Polar Coordinate). Example: 6<45
4. (Relative Polar Coordinate). Example: @6<45 [3,4].

By draw line in Figure 8 we explain the four methods of entering coordinates.

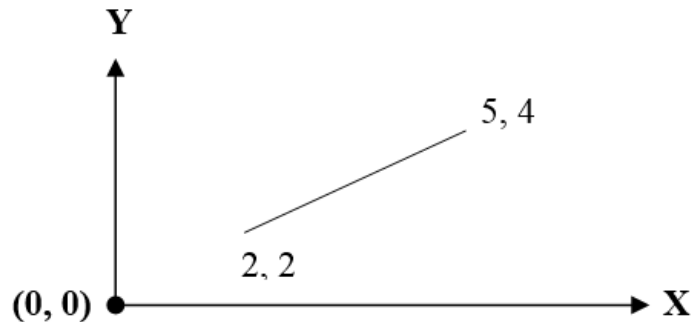


Figure 8. Line Drawing.

6. 1. Draw Line using Absolute Cartesian Coordinate System.

Command line: Line

Specify start Point: 2, 2

Specify next Point: 5, 4

The length of line calculated from original point (0, 0)

6. 2. Draw Line using Relative Cartesian Coordinate System.

Command line: Line

Specify start Point: 2, 2

Specify next Point: @3, 2

The length of line calculated from point (2, 2)

6. 3. Draw Line using Absolute Polar Coordinate System.

Command line: Line

Specify start Point: 2, 2

Specify next Point: 6.40<38.65

The length of line calculated from original point (0,0) using fethagorian triangle as following:

$$L = \sqrt{X^2 + Y^2} = \sqrt{5^2 + 4^2} = 6.40$$

The angle is calculated as following:

$$\sigma = \frac{4}{5} = 0.8 \quad , \quad Tan^{-1}(0.8) = 38.65$$

6. 4. Draw Line using Relative Polar Coordinate System

Command line: Line

Specify start Point: 2, 2

Specify next Point: @3.60<33.69

The length of line calculated from point (2, 2) using fethagorian triangle as following [10]:

$$L = \sqrt{X^2 + Y^2} = \sqrt{3^2 + 2^2} = 3.60$$

The angle is calculated as following:

$$\sigma = \frac{2}{3} = 0.66 \quad , \quad Tan^{-1}(0.66) = 33.69$$

7. BASIC OBJECTS IN AUTOCAD

Draw tool bar used to access AutoCAD draw commands you can also use draw pull down menu 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. When creating objects with draw commands, AutoCAD always prompt ask you to indicate points such as endpoints, centers, radii, to describe size and location of the object to be drawn. Drawing commands can be entered from the keyboard (command line), the draw toolbar, or the draw pull down menu. Figure 9 shows draw bar [6,8].

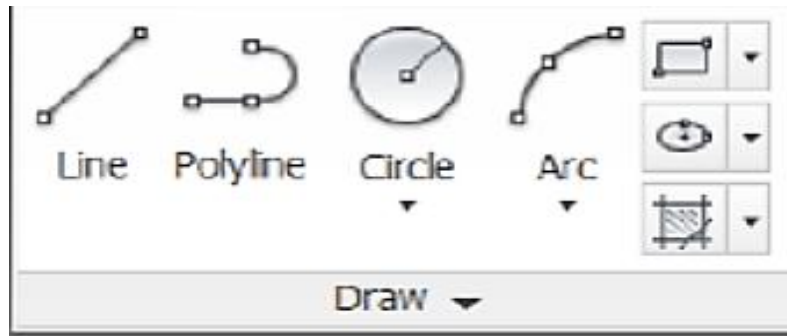


Figure 9. Draw Bar.

7. 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 [3.4]

7. 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>:

Figure 10 illustrate rectangle options [3,4,10].

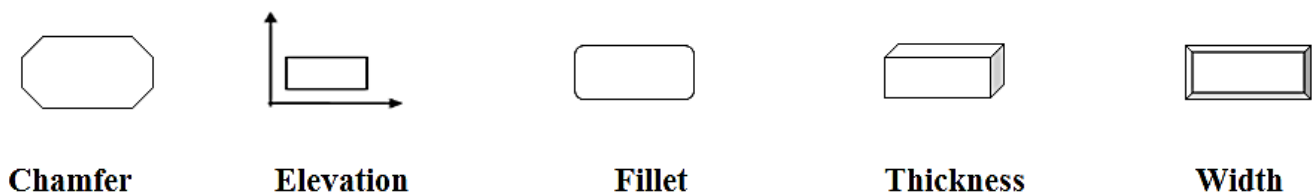


Figure 10. Rectangle Options.

7. 3. Circle

Provide Circle many ways to draw circles. Access the command from:

1. **Command Line:** Circle , C
2. **Menu Bar:** Draw ⇒ Circle
3. **Draw Bar:** 

Display following message:

Specify center point for circle or [3P/2P/Ttr(tan tan radius)]:
Specify radius of circle or [Diameter]:D
Specify diameter of circle<current default>:
Options:

- **3P (3 point)**.the circle passes through all three points specified.
- **2P (2 point)**.the two points specify the location and diameter.
- **Ttr (tan tan radius)**.specify two objects for the circle to be tangent to, then specify the radius.
- **TTT.** (Draw ⇒Circle⇒Tan Tan Tan). Figure 11 illustrate circle option [3,4,6].
-

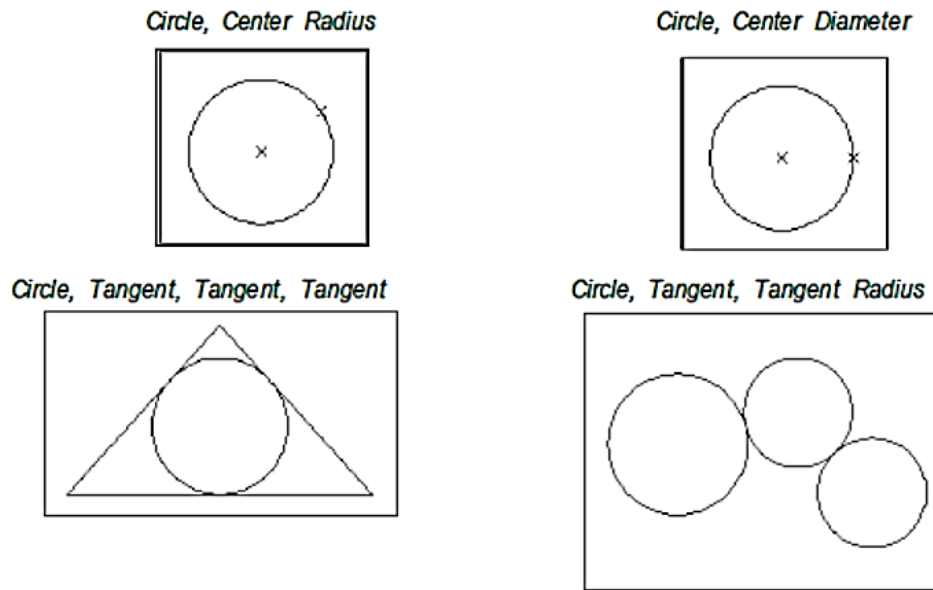


Figure 11. Circle Option.

7. 4. ARC

In AutoCAD, the Arc command is often used to produce arcs. Access the command by:

1. **Command Line:** Arc or a
2. **Menu Bar:** Draw ⇒Arc
3. **Draw Bar:** 

The following message appear:

Specify start point of arc or [Center]:
 Specify second point of arc or [Center/ End]:
 Specify end point of arc or [Angle / chord length] :
 Specify center point of arc or [Angle/ Direction / Radius]:

Options:

Angle. Draw arc using angle.

Chord Length. Draw arc using chord length [3,4,6,7]

Figure 12 illustrate ARC options.

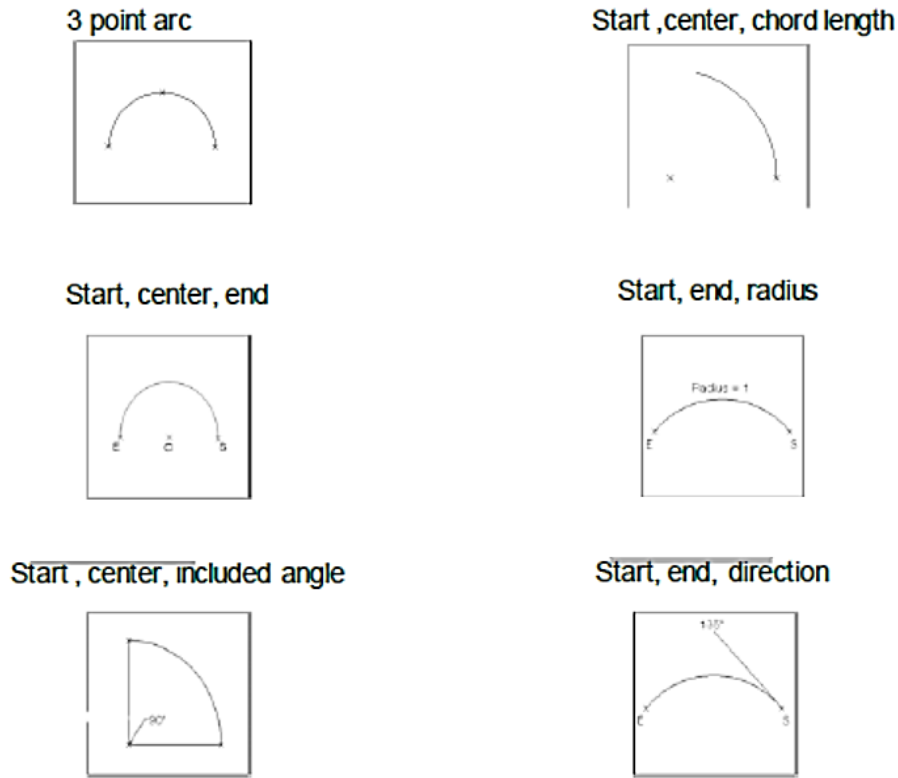


Figure 12. ARC Options.

7. 5. Ellipse

An ellipse is one object. AutoCAD ellipses are NURBS curves. Access command by:

1. **Command Line:** *El*
2. **Menu Bar:** *Draw* ⇒ *Ellipse*
3. **Draw Bar:** 

In AutoCAD three ways to draw ellipse:

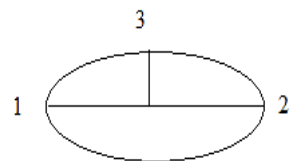
- a. Axis end (Specify one axis and the end of second)

Command : ellipse

Specify axis end point of ellipse or [Arc / Center]:

Specify other end point of axis:

Specify distance to other axis or [Rotation]:



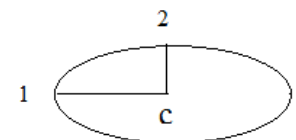
- b. Center (Specify the center and the end of each axis)

Specify axis end point of ellipse or [Arc / Center]:C

Specify center of ellipse:

Specify end point of axis:

Specify distance to other axis or [rotation]:



c. Arc (create an ellipse arc)

Specify axis end point of ellipse or [Arc / Center]:A

Specify axis endpoint of elliptical arc or [center]:

Specify other endpoint of axis:

Specify distance to other axis or [rotation]:

Specify start angle or [parameter]:

Specify end angle or [parameter / included angle]: [3,4,11-13].

8. MODIFY COMMANDS IN AUTOCAD

Modify toolbar used to access AutoCAD draw commands can also use modify pull down menu or type the command in command line. After you have created some objects in AutoCAD, objects that make up a technical drawing, such as lines or circle or a combination of both, they are sometimes just not the way you would like them, they may needed to be rotated, moved, copied, scaled larger or smaller, etc. ,to do these things you needed to have a good basic understanding of some of the AutoCAD editing commands.

1. **Erase.** This command use to delete objects.
2. **Move.** This command use to move objects to new locations.
3. **Copy.** The copy command copy single object or group of objects.
4. **Offset.** This command use to draw object parallel to selected object by determine distance.
5. **Mirror.** This command use to do mirror to object. i.e. (draw the half of the object then copy the object to create another half).
6. **Trim.** This command allows to trim (shorten) the end of an object back to the intersection of anther object. The middle section of an object can also be trimmed between two intersecting objects.
7. **Extend.** Extend command use to length object to meet another object. The objects can be extended include arc, ellipse, lines, Mlines, rays.
8. **Break.** This command use to break object.
9. **Rotate.** This command use to rotate slected objects around point by determining angle.
10. **Scale.** This command use to zoom in or zoom out the drawing in draw board.
11. **Lengthen.** From this command can know length of selected line, and do changes to line length.
12. **Stretch.** This command stretch selected object from selected side.
13. **Fillet.** This command used to change two object meeting to meeting by circular arc.
14. **Chamfer.** This command used to change two object meeting to meeting by line.
15. **Array.** This command used for create multiple copies of object in the pattern, Can create rectangular array or polar array.
16. **Explode.** This command used to destroy the connection between one object. Figure 13 shows modify bar [6-8,11].



Figure 13. Modify Bar.

9. BATHROOM FLOOR PLAN.

In this work we Draw the Bathroom floor plan on board its' sides are (12, 9ft), as shown in Figure 14, using AutoCAD objects line, circle, rectangle, arc and ellipse by apply the three methods of coordinate system and compare between them:

1. Absolute Coordinate System.
2. Relative Coordinate System.
3. Polar Coordinate System.

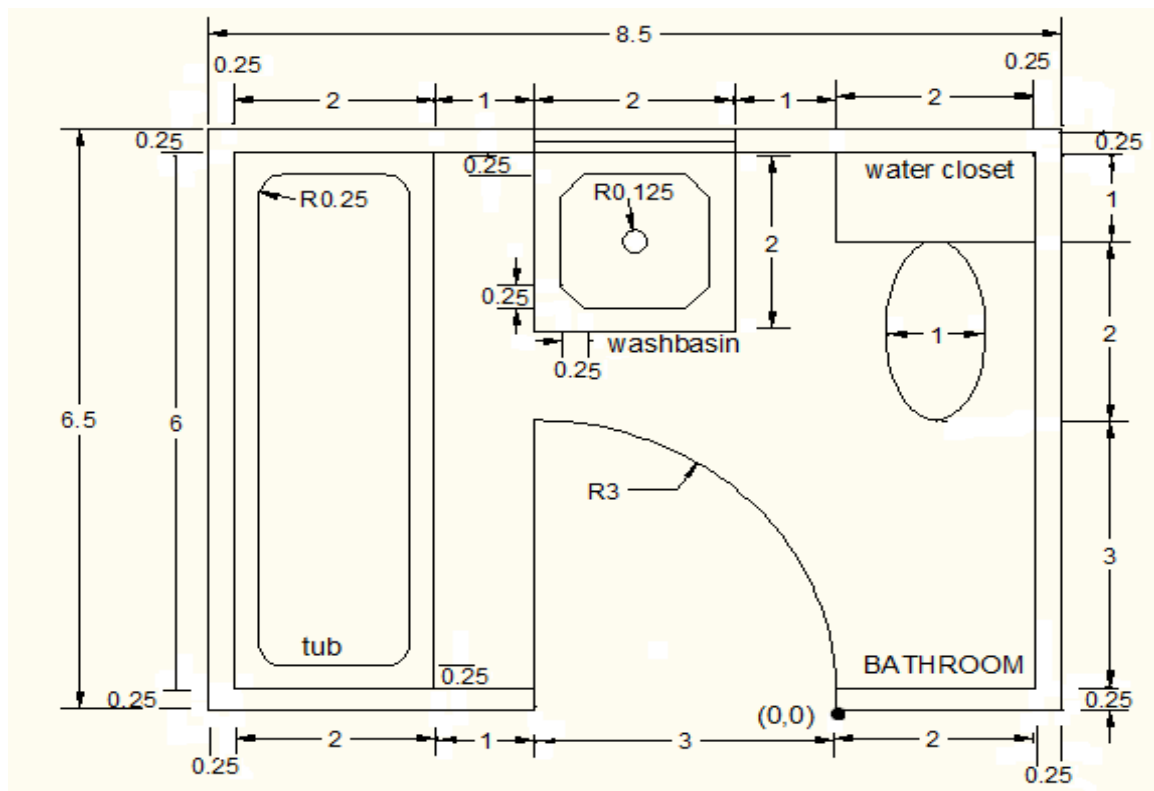


Figure 14. Bathroom Floor Plan.

9. 1. Drawing Units Setup

Every object we construct in a CAD system is measured in units. We should determine the system of units within the CAD system before creating the first geometric entities.

1. In the Menu Bar select: [Format] ----- [Units]
2. Click on the Length Type option to display the different types of length units available. Confirm the Length Type is set to [Decimal].
3. In the Drawing Units dialog box, set the Length Type to Decimal. This will set the measurement to the default English units, feet.
4. Set the Precision to three digits after the decimal point as shown in the above figure.
5. Pick OK to exit the Drawing Units dialog box.

9. 2. Drawing Area Setup

Next, we will set up the Drawing Limits by entering a command in the command prompt area. Setting the Drawing Limits controls the extents of the display of the grid. It also serves as a visual reference that marks the working area.

1. In the Menu Bar select: [Format] ----- [Drawing Limits]
2. In the command prompt area, the message “Reset Model Space Limits: Specify lower left corner or [On/Off] <0.00,0.00>:” is displayed. Press the ENTER key once to accept the default coordinates <0.00,0.00>.
3. In the command prompt area, the message “Specify upper right corner <12.00,9.00>:” is displayed. Press the ENTER key again to accept the default coordinates <12.00,9.00>. The Drawing Limits command is used to set the drawing area, but the display will not be adjusted until a display command is used.
4. Inside the Menu Bar area select:
[View] ----- [Zoom] ----- [All]
The Zoom All command will adjust the display so that all objects in the drawing are displayed to be as large as possible. If no objects are constructed, the Drawing Limits are used to adjust the current viewport.
5. Hit the function key [F7] once to turn off the display of the Grid lines.

9. 3. Solve using Relative Polar Coordinate System

Draw Walls

Command:Line

Specify first point:0,0

Specify next point:@2.25<0

Specify next point:@6.5<90

Specify next point:@8.5<180

Specify next point:@6.5<270

Specify next point:@3.25<0

Specify next point:@0.25<90

Specify next point:@3<180

Specify next point:@6<90

Specify next point:@8<0

Specify next point:@6<270

Specify next point:@2<180
Specify next point:@0.25<270 or C

Draw Tub

Command:Rectangle
Specify first corner point:-6,0.25
Specify other corner point:@6.324<71.565

Command:Rectangle
Specify first corner point or []:F
Specify fillet radius for rectangles:0.25
Specify first corner point:-5.750,0.25
Specify other corner point:@5.700<74.744

Draw Water Closet

Command: Rec
Specify first corner point:0,5.25
Specify other corner Point:@2.236<26.565

Command:ellipse
Specify axis end point of ellipse or []:1,5.25
Specify other end point of axis:@2<270
Specify distance to other axis or [Rotation]:0.5

Draw Washbasin

Command: Rec
Specify first corner point:-3,4.25
Specify other corner point:@2.828<45

Command:Rectangle (enter)
Command:Chamfer
Specify first chamfer distance for rectangle :0.25
Specify second chamfer distance for rectangle: 0.25
Specify first corner point: -2.750,4.5
Specify other cornerpoint:@2.121<45

Draw Window

Command:Line
Specify first point:-3,6.25
Specify next point:@0.25<90

Command:Line
Specify first point:-1,6.25
Specify next point:@0.25<90

Command:Line
Specify first point:-3,6.375
Specify next point:@2<0

Draw Door

Command: Line
Specify first point:-3,0.25
Specify next point:@3<90

Command:arc
Specify first point:0,0.25
Specify center point:@3<180
Specify included angle:90

9. 4. Solve using Relative Coordinate System

Draw Walls

Command:Line
Specify first point:0,0
Specify next point:@2.25,0
Specify next point:@0,6.5
Specify next point:@-8.5,0
Specify next point:@0,-6.5
Specify next point:@3.25,0
Specify next point:@0,0.25
Specify next point:@-3,0
Specify next point:@0,6
Specify next point:@8, 0
Specify next point:@0,-6
Specify next point:@-2.0
Specify next point:@0,-0.25 or c

Draw Tub

Command:Rectangle
Specify first corner point:-6,0.250
Specify other corner point:@2,6

Command:Rectangle
Specify first corner point or []:F
Specify fillet radius for rectangles:0.25
Specify first corner point:-5.750,0.5
Specify other corner point:@ 1.5,5.5

Draw Water Closet

Command:Rectangle
Specify first corner point:0,5.25
Specify other corner point:@2,1

Command:ellipse
Specify axis end point of ellipse:1,5.25
Specify other end point of axis:@0,-2
Specify distance to other axis or :0.5

Draw Washbasin

Command:Rectangle
Specify first corner point:-3,4.25
Specify other corner point:@2,2

Command:Rectangle
Specify first corner point:C
Specify first chamfer distance for rectangles:0.25
Specify second chamfer distance for rectangles:0.25
Specify first corner point:-2.75,4.5
Specify other corner Point:@1.5,1.5

Command:C
Specify center point for circle:-2,5.25
Specify radius of circle:0.125

Draw Window

Command:Line
Specify first point:-3,6.25
Specify next point:@0,0.25

Command:Line
Specify first point:-1,6.25
Specify next point:@0,0.25

Command:Line
Specify first point:-3,6.375
Specify next point:@2,0

Draw Door

Command:Line
Specify first point:-3,0.25
Specify next point:@0,3

Command:arc
Specify first point:0,0.25
Specify center point:@-3,0
Specify included angle:90

9. 5. Solve using Absolute Coordinate System

Draw Walls

Command:Line

Specify first point:0,0

Specify next point:2.25,0

Specify next point:2.25,6.5

Specify next point:-6.25,6.5

Specify next point:-6.25,0

Specify next point:-3,0

Specify next point:-3,0.25

Specify next point:-6,0.25

Specify next point:-6,6.25

Specify next point:2, 6.25

Specify next point:2,0.25

Specify next point:0,0.25

Specify next point:0,0 or C

Draw Tub

Command:Rec

Specify first corner point:-6,0.250

Specify other corner point:-4,6.25

Command: Rec

Specify first corner point or []:F

Specify fillet radius for rectangles:0.25

Specify first corner point:-5.750,0.5

Specify other corner Point:-4.25,6

Draw Water Closet

Command:Rec

Specify first corner point:0,5.25

Specify other corner point:2,6.25

Command:ellipse

Specify axis end point of ellipse:1,5.25

Specify other end point of axis:1,3.25

Specify distance to other axis or :0.5

Draw Wash Basin

Command:Rec

Specify first corner point:-3,4.25

Specify other corner point:-1,6.25

Command:Rec

Specify first corner point :C

Specify first chamfer distance for rectangles:0.25

Specify second chamfer distance for rectangles:0.25
Specify first corner point:-2.75,4.5
Specify other corner point:-1.25,6

Command:C
Specify center point for circle:-2,5.25
Specify radius of circle:0.125

Draw Window

Command:Line
Specify first point:-3,6.25
Specify next point:-3,6.5

Command:Line
Specify first point:-1,6.25
Specify next point:-1,6.5

Command:Line
Specify first point:-3,6.375
Specify next point:-1,6.375

Draw Door

Command:Line
Specify first point:-3,0.25
Specify next point:-3,3

Command:arc
Specify 1st point:0,0.25
Specify center point:-3,0.25
Specify included angle:90

10. CONCLUSIONS

- Using absolute coordinate system ,enter the points as they relate to the origin of the WCS(world coordinate system).the origin of the WCS is at the lower left corner of the drawing area. Where the USC icon is.
- The relative coordinate entry method allows to enter points in reference to the last point picked. The @ symbol tells AutoCAD that use the relative coordinate entry to locate another point. This symbol must always be put in front of X and Y coordinate.
- Using polar coordinate entry if want to draw a line certain distance at a particular angle.
- The tools grid , snap, osnap. ortho, track used To Assist In Accuracy of the Drawing.
- In order to become effective and efficient in using a CAD system, we must learn to create geometric entities quickly and accurately.
- In learning to use a CAD system, lines and circles are the first two, and perhaps the most important two, geometric entities that one should master the skills of creating and

modifying. As one gains some experience in creating lines and circles, similar procedures can be applied to create other geometric entities.

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