



University of Engineering & Technology Peshawar

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Course Name: Uni-elective I (CAD)

Credit hours: 3

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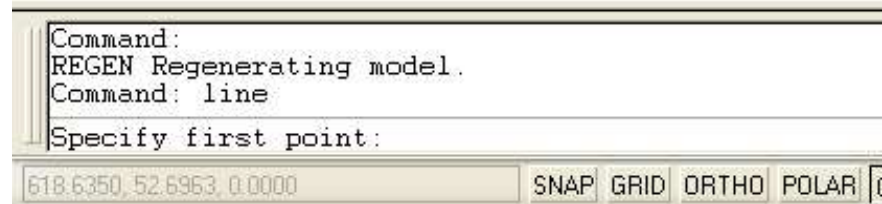
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INTRODUCTION TO AUTOCAD



How do we give a command?

- Command line



- Toolbars



- Menu bar

What is OSNAP?

- ❑ Osnap (Object Snap) settings make it easier to select a 2d object's points
 - ❑ Endpoint
 - ❑ Midpoint
 - ❑ Perpendicular
 - ❑ Center
 - ❑ Intersection
- ❑ Osnap will be active when AutoCAD is expecting you to pick a point on the working area
- ❑ Type osnap on your command window:



Zooming...

- You will need to zoom in and out while drawing with AutoCAD. This doesn't change your objects or UCS, only the way you see your working space. This can be done in many ways:
 1. Scroll bars
 2. Typing **z** or **zoom** in your command window.
 - All
 - Center
 - Dynamic
 - Extents
 - Previous
 - Scale
 - Window

CARTESIAN COORDINATE SYSTEM

- The word “Cartesian” comes from the philosopher Descartes, who came up with the idea. One day Descartes was staring at a fly on the floor. It occurred to him that he could describe the fly’s location floor by noting its distance from two of the walls.
- Note the UCS icon in autocad. It is composed of stubs/ arrows sticking out in the x-direction, the y-direction, and, the z-direction. The z-direction is hidden if you are working in 2D, but you can think of it as coming out of the page (or monitor) The place where the stubs intersect is called the origin.

TYPES OF COORDINATE SYSTEMS

- ABSOLUTE COORDINATE
- RELATIVE COORDINATE
- POLAR COORDINATE

ABSOLUTE COORDINATE ENTRY

Absolute coordinate entry is the default. Absolute coordinate entry always counts from the origin. When drawing lines, AutoCAD is ready to place the next vertex wherever you tell it to go:

1. Type L on the command line and press enter
2. Type 4,6 on the command line and press enter
3. Type 5,3 on the command line and press enter
4. Press enter to finish the Line command

RELATIVE COORDINATE ENTRY

For relative coordinates, AutoCAD does not count from 0,0. Instead, the reference point is the previous point in the drawing. Let's draw another line, and see what it looks like compared to the old one:

1. Type L on the command line and press enter
2. Type 4,6 on the command line and press enter
3. Type @5,3 on the command line and press enter
4. Press enter to finish the line command

POLAR COORDINATE SYSTEM

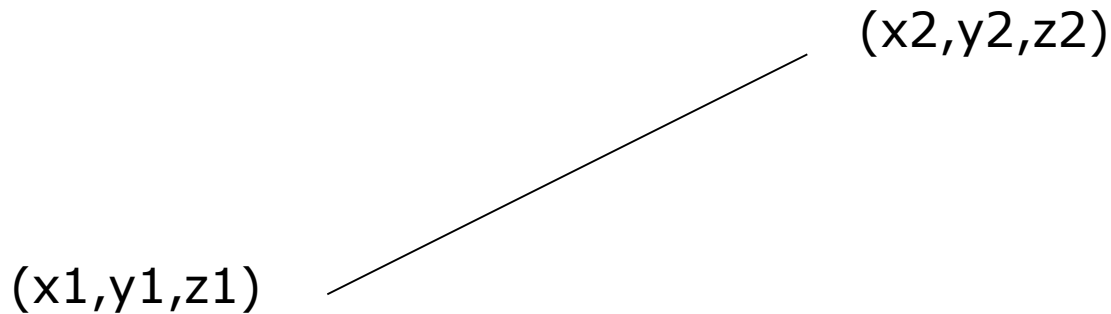
Polar coordinates are named for their “pole”; the reference point to start counting from, which is similar in concept to the origin. Polar coordinates employ one radial distance and one angle (by convention, the angle is measured counterclockwise from the x-axis). The angle signifies the direction, and the distance is how far to go in that direction from the pole.

Polar coordinates in AutoCAD are given as R<Degrees, where:

- “R” is the radial distance
- “<” is the less than sign (but in this case represents the angle)
- “Degrees” is the value of the angle in degrees

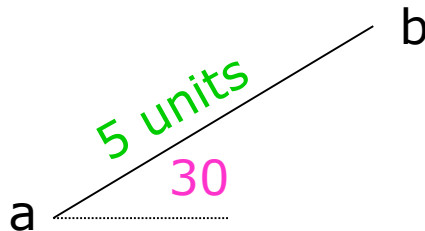
Lets draw a LINE:

- Remember that AutoCAD recognizes an object by its coordinates. You will need two given points to draw a line.



- You can start at a random point on your WCS for your FIRST POINT, but you should specify the coordinates of your SECOND POINT.

Lets draw a LINE (1):



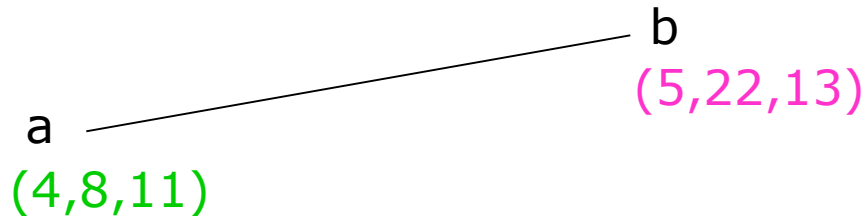
1. Give the command
 - ❑ Type “line” on the command window, OR
 - ❑ Click on the line icon on the Draw toolbar, OR
 - ❑ Select Line on the Draw menu
2. Specify the first point (a)
 - ❑ Click on a random point on your working area (black space)
3. Specify the second point in relation to the first point
 - ❑ @**distance**<**degrees**
 - ❑ @5<30

Lets draw a LINE (2):

a 7 units b

1. Hit F8 (ortho on)
2. Give the command
3. Specify the first point
4. Specify the second point in relation to the first point
 - ▣ Point the cursor to the left hand side. You will see that the cursor snaps only to 0° - 90° - 180° - 270°
 - ▣ Type 7 and hit enter

Lets draw a LINE (3):



If we know the coordinates of the line we want to draw, we can simply type them into the command line. (However, this mostly is not the case.)

1. Give the command
2. Specify the first point (4,8,11)
3. Specify the second point (5,11,23)