Engineering Drawing with Autocad

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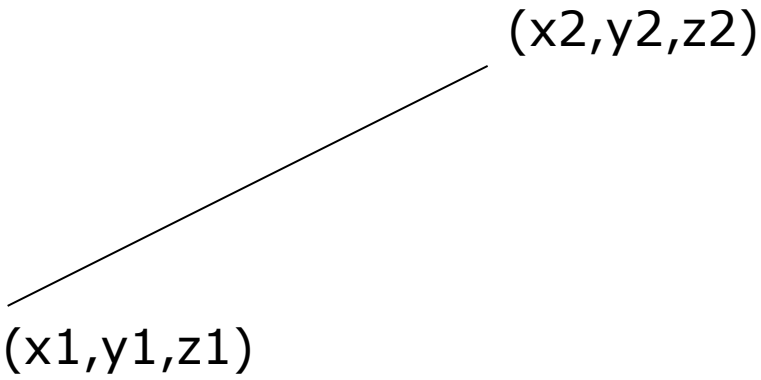
Lecture 2
Basic Drawing & Editing Commands

Basic Drawing & Editing Commands

1- Drawing Lines

*************(To Draw Lines)**

- Click Home tab > Draw panel > Line. Find.
- Specify the start point and end point of the line segment by clicking in the drawing area.
- Continue specifying additional line segments. ...
- Press Enter or Esc when done or enter c to close a series of line segments.



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

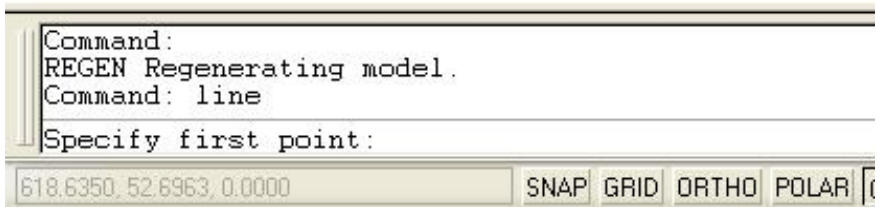
✓ You can start at a random point on your WCS for your FIRST POINT, but you should specify the coordinates of your SECOND POINT.

How do you draw a line in CAD?

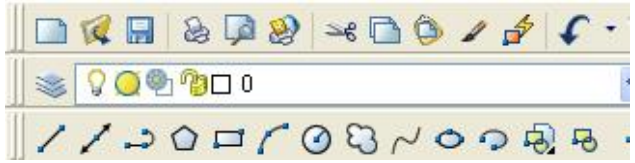
Step 1: For creating an object, take a line command.
Step 2: Specify the first point.
Step 3: Give direction in which you want to draw a line, give length, then press Enter key.
Step 4: Don't exit from line command

Lets draw a LINE (1):

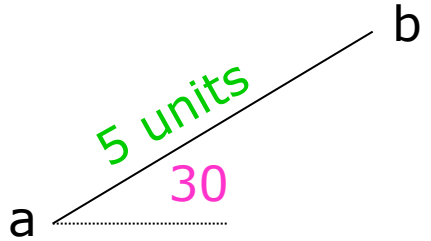
1. Give the command
   - Type “line” on the command window, OR
   - Click on the line icon on the Draw toolbar, OR
   - Select Line on the Draw menu



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



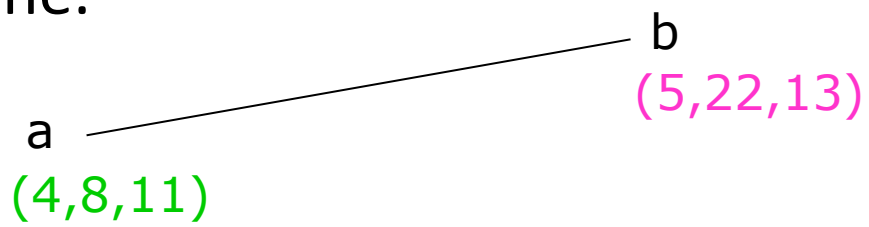
Lets draw a LINE (2):

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^o-90^o-180^o-270^o
   - 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)

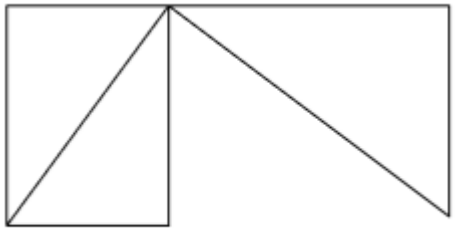
2- Erasing Objects

***********(To Erase an Object)**

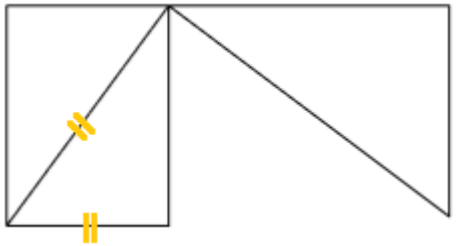
- Enter L (Last) to erase the last object drawn.
- Enter p (Previous) to erase the last selection set.
- Enter all to erase all objects from the drawing.
- Enter ? to see a list of all selection methods.

Example 1:

Consider the below figure in this example



Here, we will erase the lines marked in the below figure:



The steps are listed below:

1 - Select the **Erase icon** from the ribbon panel, as shown below:

Or



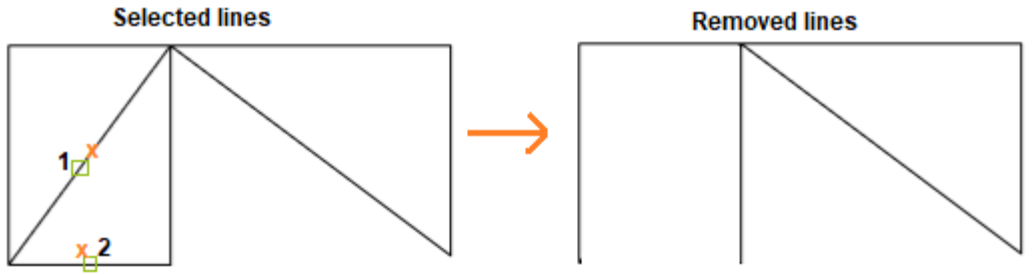
Type **E** or **Erase** on the command line or command prompt a
press **Enter**.

2 - Select the two lines marked in the above image.

3 - Press **Enter**.

The two lines will be removed.

The process is shown in the below image:



3 - Drawing Rectangles:

The rectangle command is used to create rectangles in our drawing.
There are two methods to draw a rectangle. In the first method, we can create rectangles randomly by specifying the points. In the second method, we can create rectangles by specifying the value (length and width).

Example 1:

To draw rectangles randomly.

1 - Select the rectangle command from the ribbon panel. The rectangle icon will look like the below image:



Or

Type Rec or Rectangle in the command line and press Enter.

2 - Specify the first corner point on the viewport.

3 - Specify the second corner point (diagonally opposite to the first point) on the viewport.

Example 2:

To draw a rectangle by specifying the length and width.

The steps are given below:

1 - Select the rectangle command from the ribbon panel. The rectangle icon will look like the below image:



Or

Type Rec or Rectangle in the command line and press Enter.

2 - Specify the first corner point on the viewport.

3 - Specify the length and breadth of the rectangle in the form of @length, width. For example, @4,5.

Where, 4 is the length of the rectangle, while 5 is the width of the rectangle.

4 - Press Enter.

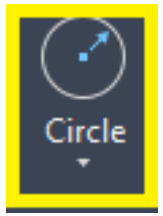
4 - Drawing Circles:

The **circle** command is used to **draw** a **circle** by specifying the center point and radius. Let's understand by two examples.
... Type **Circle** or C on the command line ..

Example 1:

The steps to create a circle are given below:

1 - Click on the Circle icon on the Ribbon Panel, as shown in the below image:



OR

Type **Circle** or C on the command line and press **Enter**.

2 - Specify the center point of a circle on the workspace or viewport.

3 - Specify the **radius** of the circle.

Or

To specify **diameter**, type **D** - press **Enter** - specify the diameter of the circle.

4 - Press **Enter**.

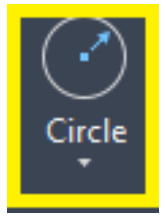
Example 2:

The steps to create a circle are given below:

1 - Click on the Circle icon on the Ribbon Panel, as shown in the below image:

OR

Type **Circle** or C on the command line and press **Enter**.



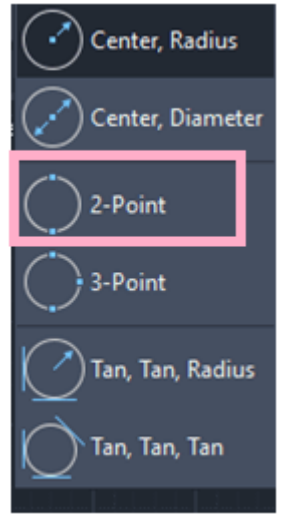
2 - Specify the center point of a circle on the workspace or viewport.

3 - Specify the endpoint of the radius through the cursor.

Types of Circles

There are three types of circles, which are listed below:

1. **2 point circle**
2. **3 point circle**
3. **Tangent circle**



5 - Undo and Redo Actions:

To UNDO and REDO using keyboards shortcutsUNDO: Use Ctrl (Windows) or ⌘ (Mac) +Z. REDO: Use Ctrl or ⌘ +Y

The UNDO command will take you back through your drawing session, one command at a time, all the way back to the start of your drawing. If you go back too far, the REDO command will move you forward, one command at a time, until you've restored everything