A MANUAL ON USING THE COMMANDS

IN

AUTOCAD 2007

Submitted To:

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In AutoCAD work is done using a few commands. On the bottom portion of the screen the command windows is displayed. Aside from selecting a command from the menu or toolbar, a command can also be activated by typing in the command in the command window. Some of the command also have shortcuts for the ease of use.

A brief description of the commands in AutoCAD 2007 is given below:-

Drawing Tools

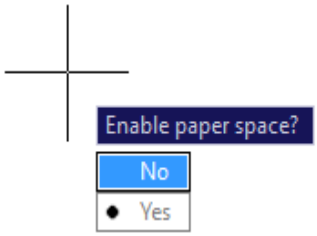
Command Name: MVSetup

Command Shortcut: mvsetup

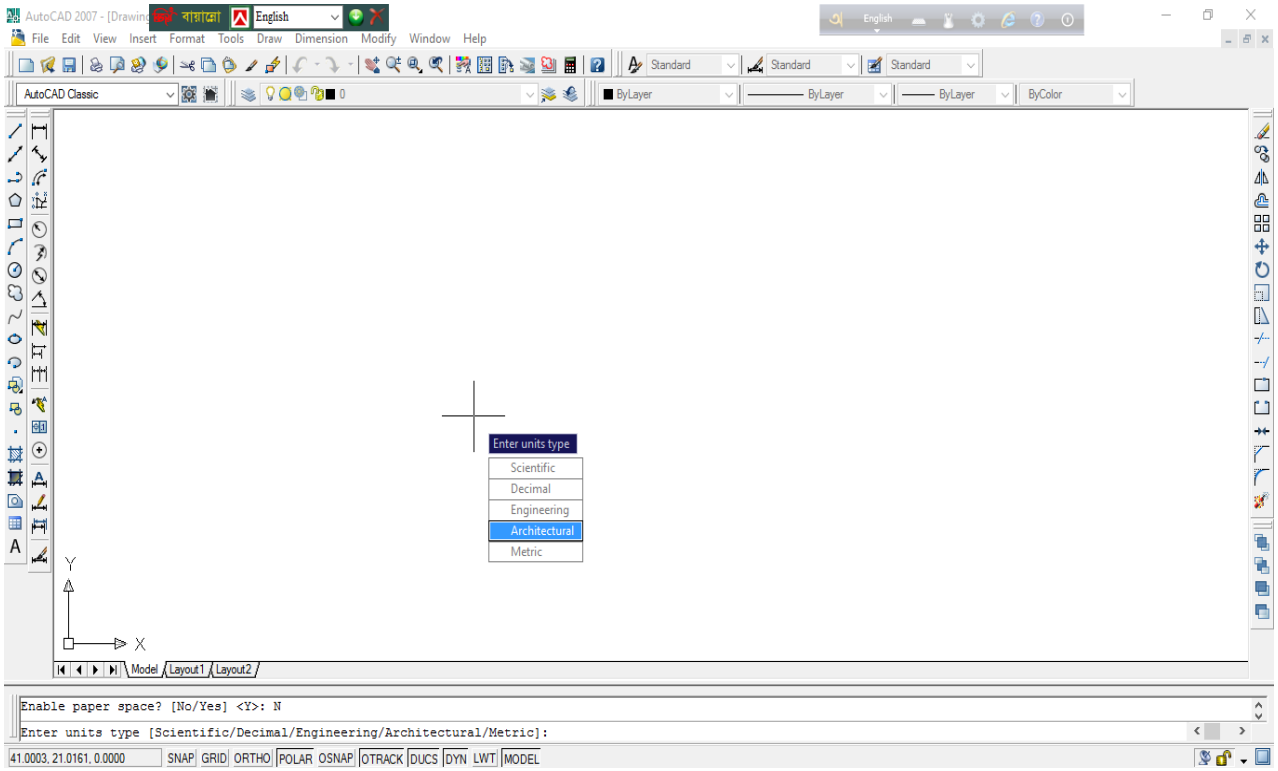
Description: It is the primary set up for AutoCAD. It is performed for setting the units, scale factor and paper size for measurement.

Use:

1. Type **MVSETUP**, enter.
2. Select **No** for enable paper size.
3. Choose unit type. For feet, inch we have to select **Architectural**.
4. Select the scale factor, enter.
5. Type paper width, enter.
6. Type paper height, enter.



Enable paper size>No



Choosing unit

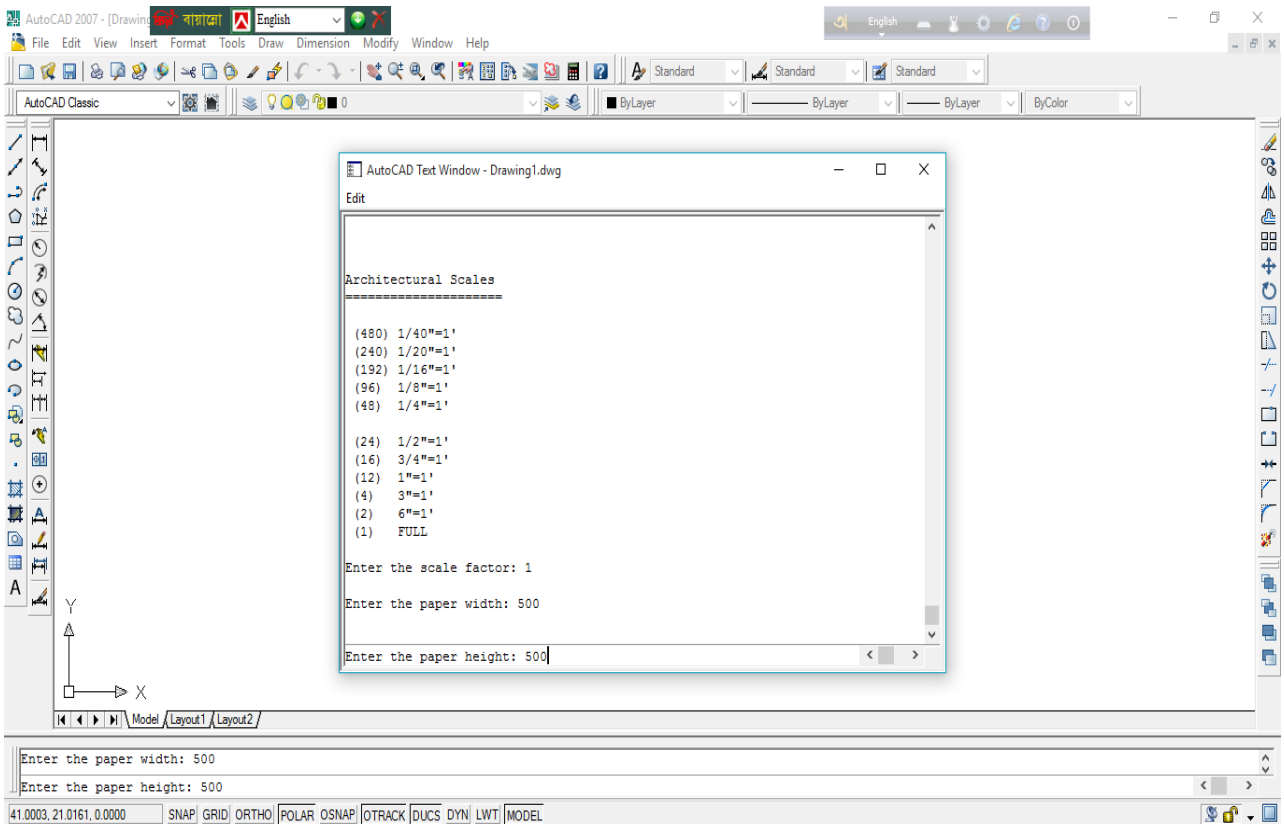


Fig: Scale factor, paper width and height

Command Name: Ortho

Command Shortcut: F8 or ortho

Description: It is used for drawing lines along X and Y axis.

Use:

1. Ortho can be off or on by clicking **Ortho** mode icon of status bar.
2. Or by pressing hot key "**F8**".

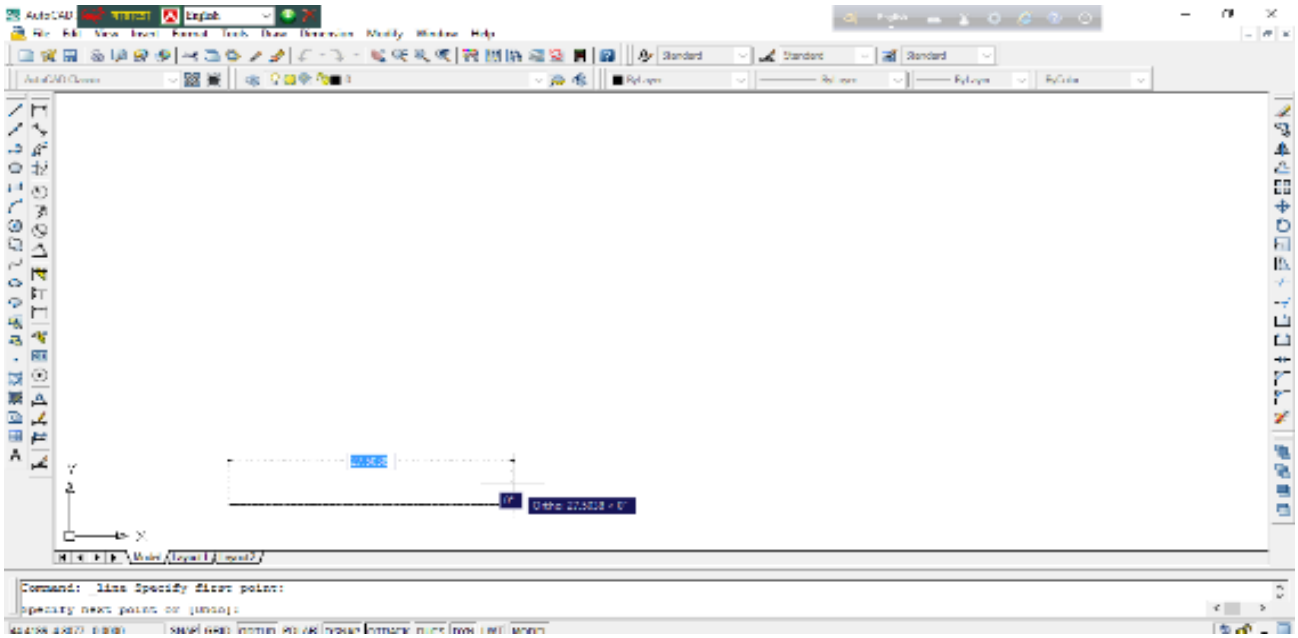


Fig: Movement along X axis

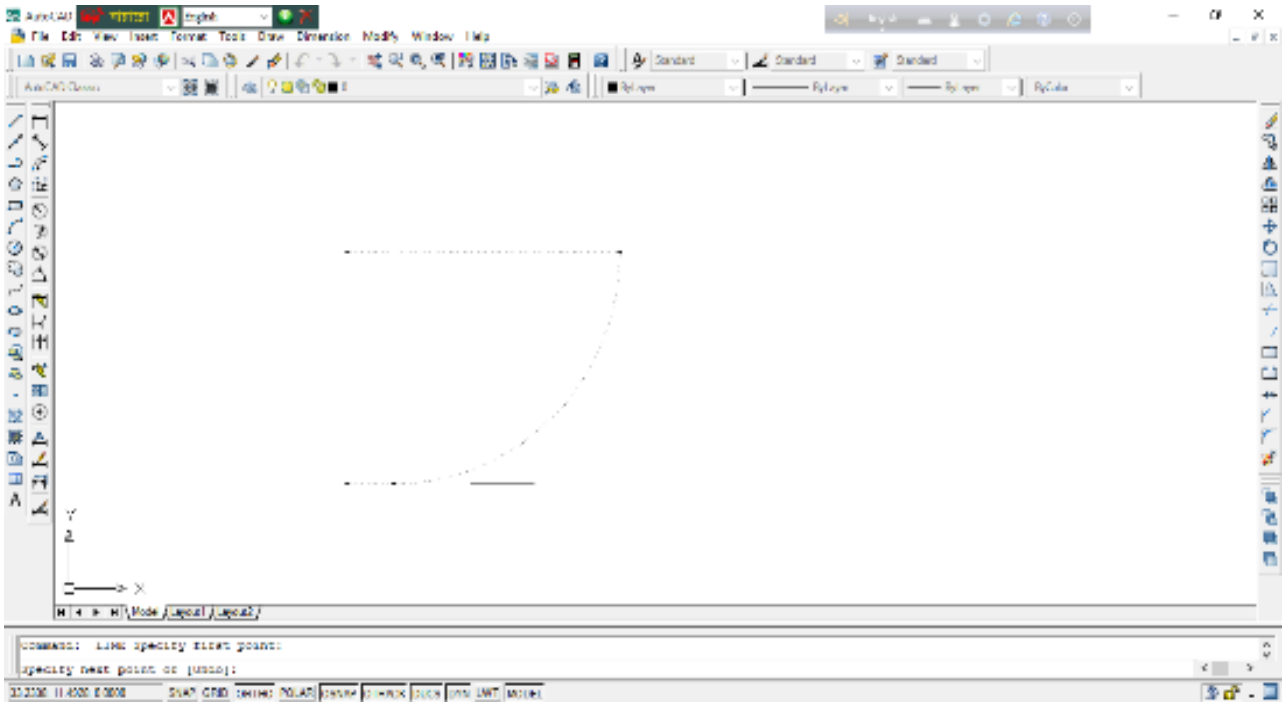


Fig: movement along Y axis

Command Name: Polar tracking

Command Shortcut: polar or F10

Description: Polar is used for regulating the movement of cursor with a particular angle polar tracking is used.

Use:

1. Clicking "**Polar**" button of status bar, it can be on or off.
2. Or by pressing hot key **"F10".**
3. To fix polar snap, set increment angle on polar tracking tab of drafting setting.
4. To get the drafting setting dialogue you have to select **Tools>Drafting setting** or press enter typing **DS** on command line, then the dialogue box will appear.

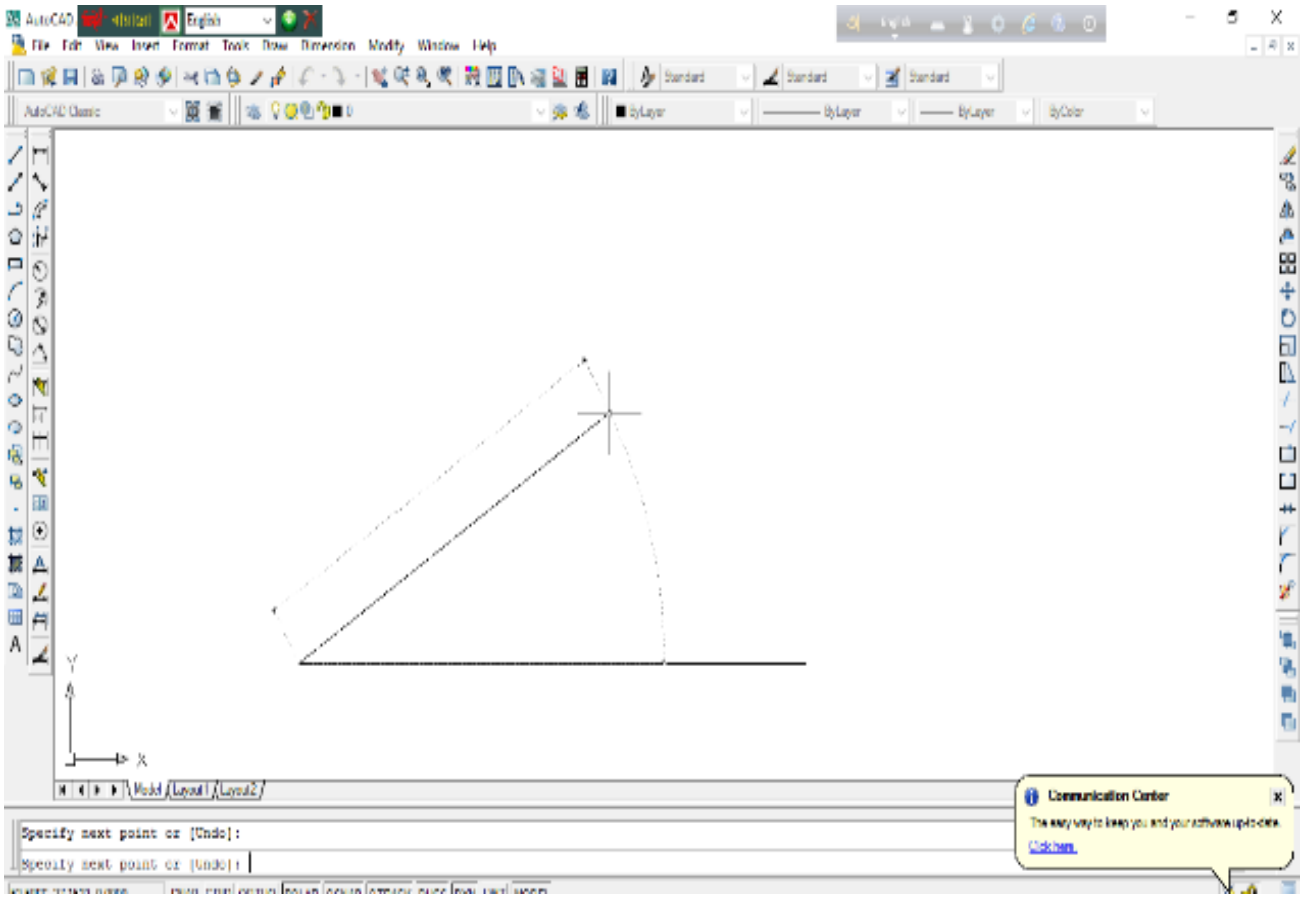


Fig: Polar movement

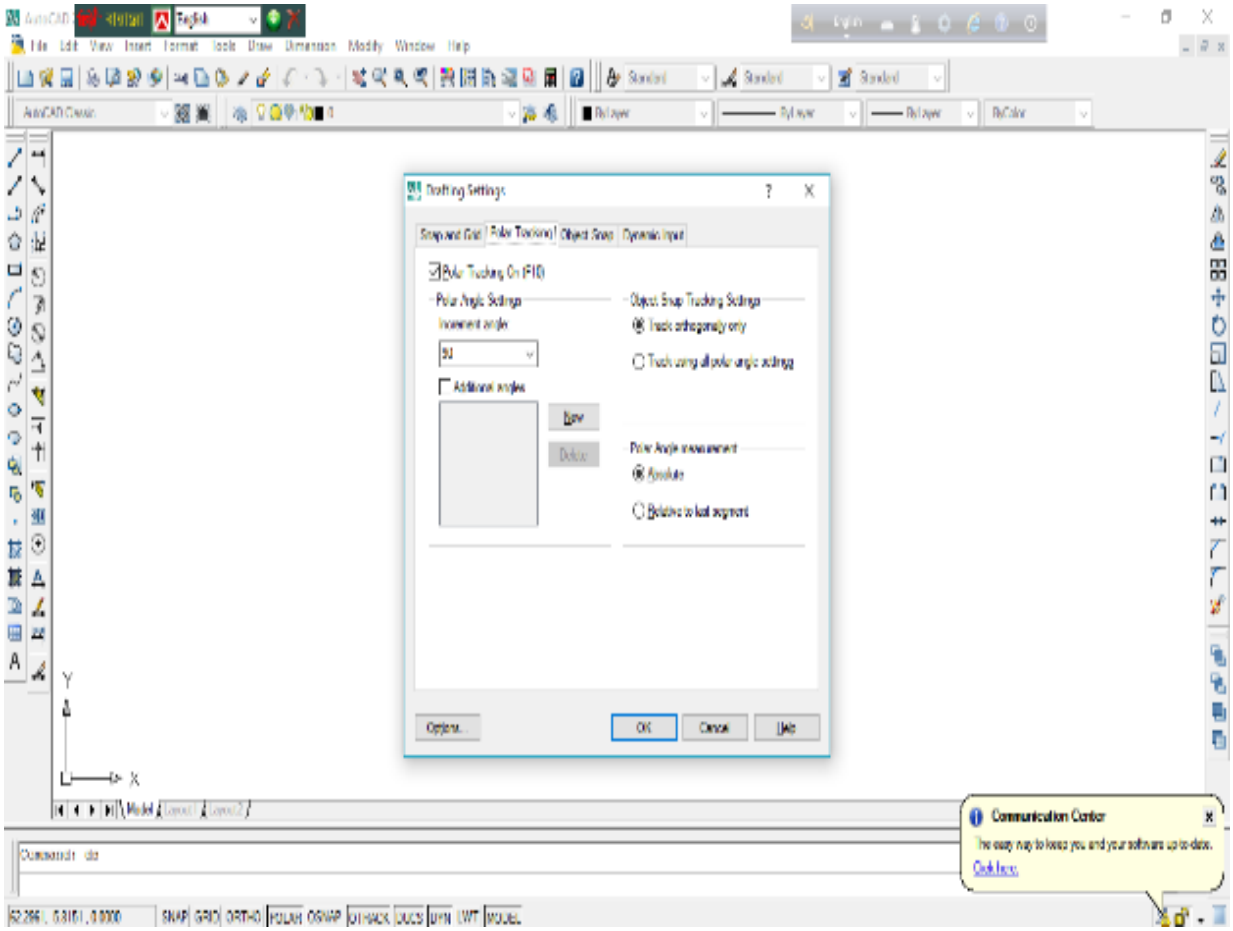


Fig: Drafting setting

Command Name: Object snap

Command Shortcut: osnap

Description: Object snap or OSNAP is used move the cursor at a particular point on the object.

Use:

1. Enter the command **OSNAP** in the command line.
2. Or by clicking on **"Object Snap on"** box of object snap tab or by pressing **F3** key or clicking **OSNAP** button of status bar.

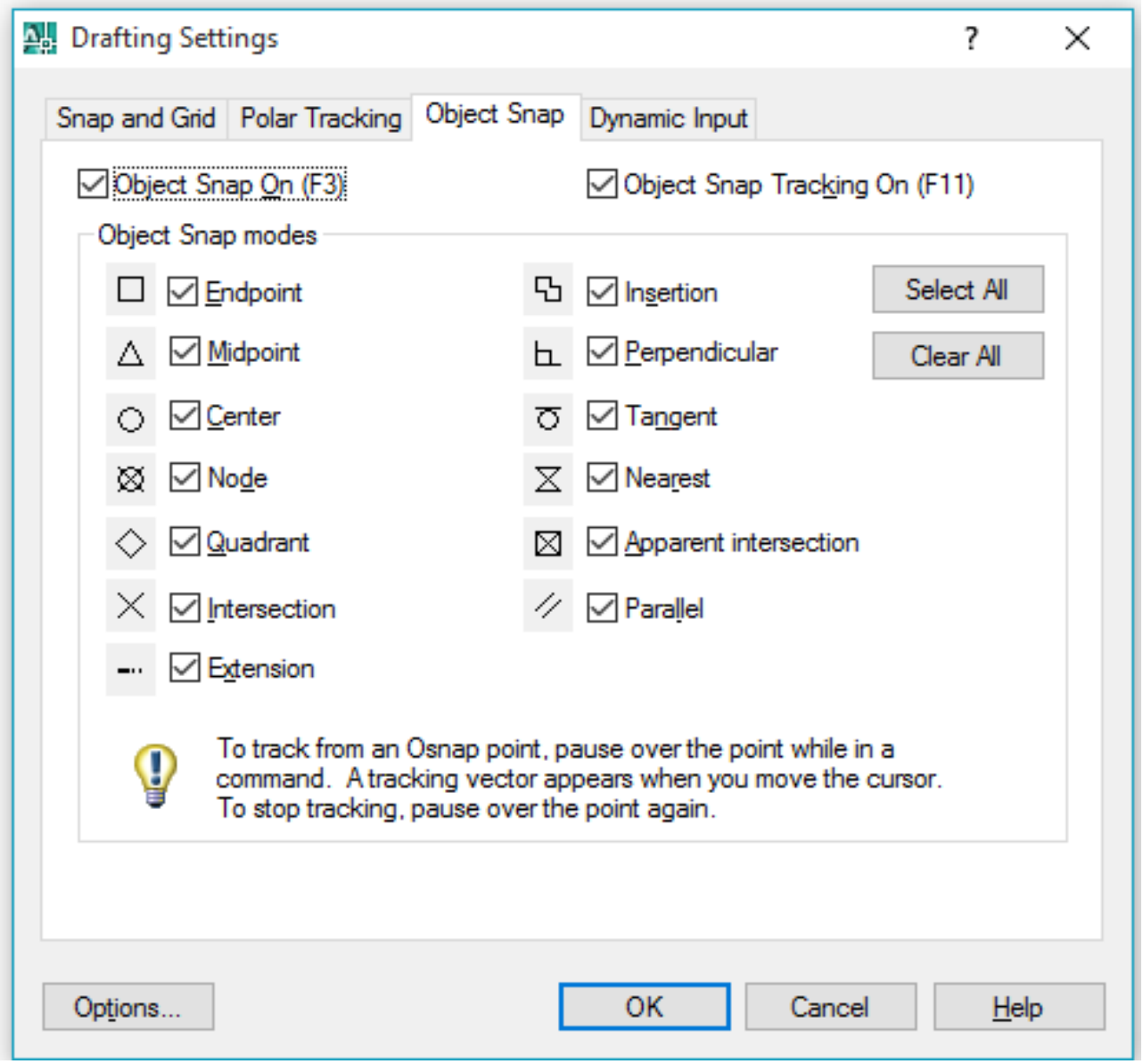


Fig: Drafting Settings

Object snap has 13 snap modes. They are:-

End point mode: Traces the end point of arc, ellipsoidal arc, line, multiline, polyline segment. It snaps at the corner point of any solid or 3D face i.e. limits the cursor movement at that point.

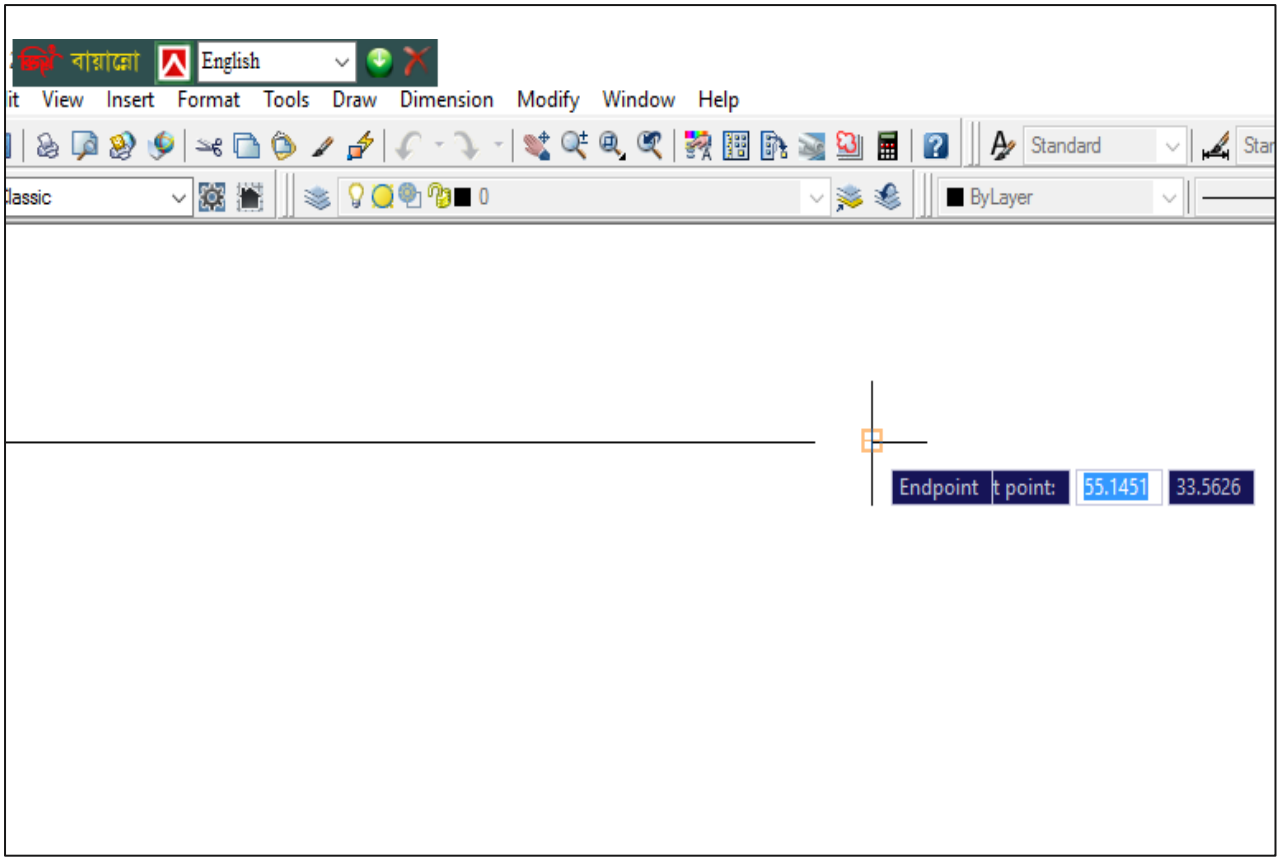


Fig: End point

Midpoint mode: Snaps at the midpoint of arc, ellipsoidal arc, line, multiline, polyline segment, solid sp line or x-line.

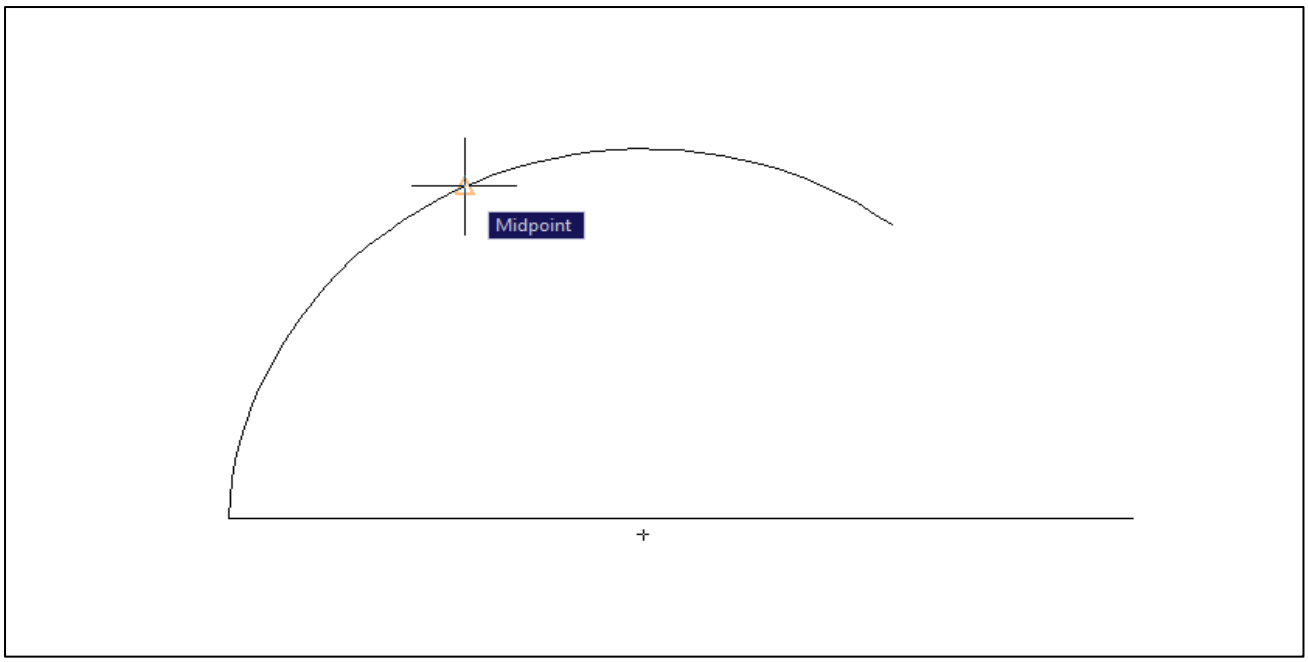


Fig: Midpoint

Center mode: Snaps at the center point of any arc, circle, ellipse or ellipsoidal arc.

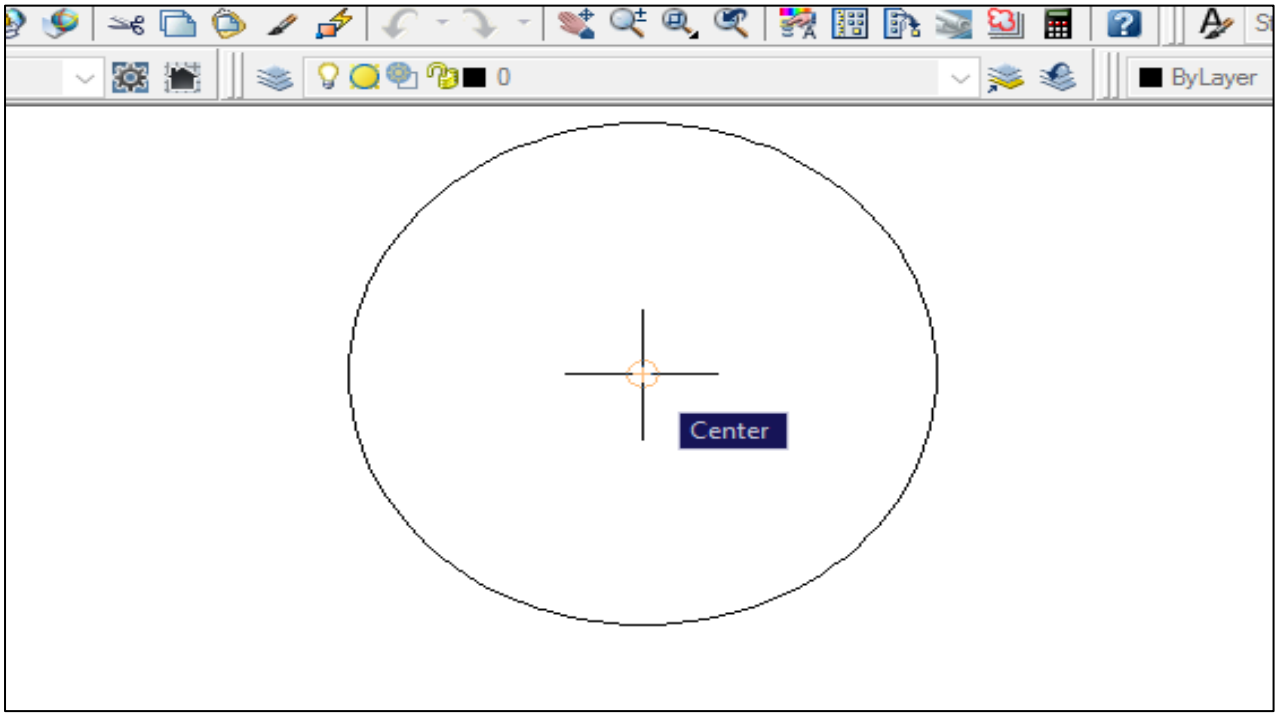


Fig: Center point

Node mode: Snaps at the quadrant point of any arc, circle, ellipse or ellipsoidal arc.

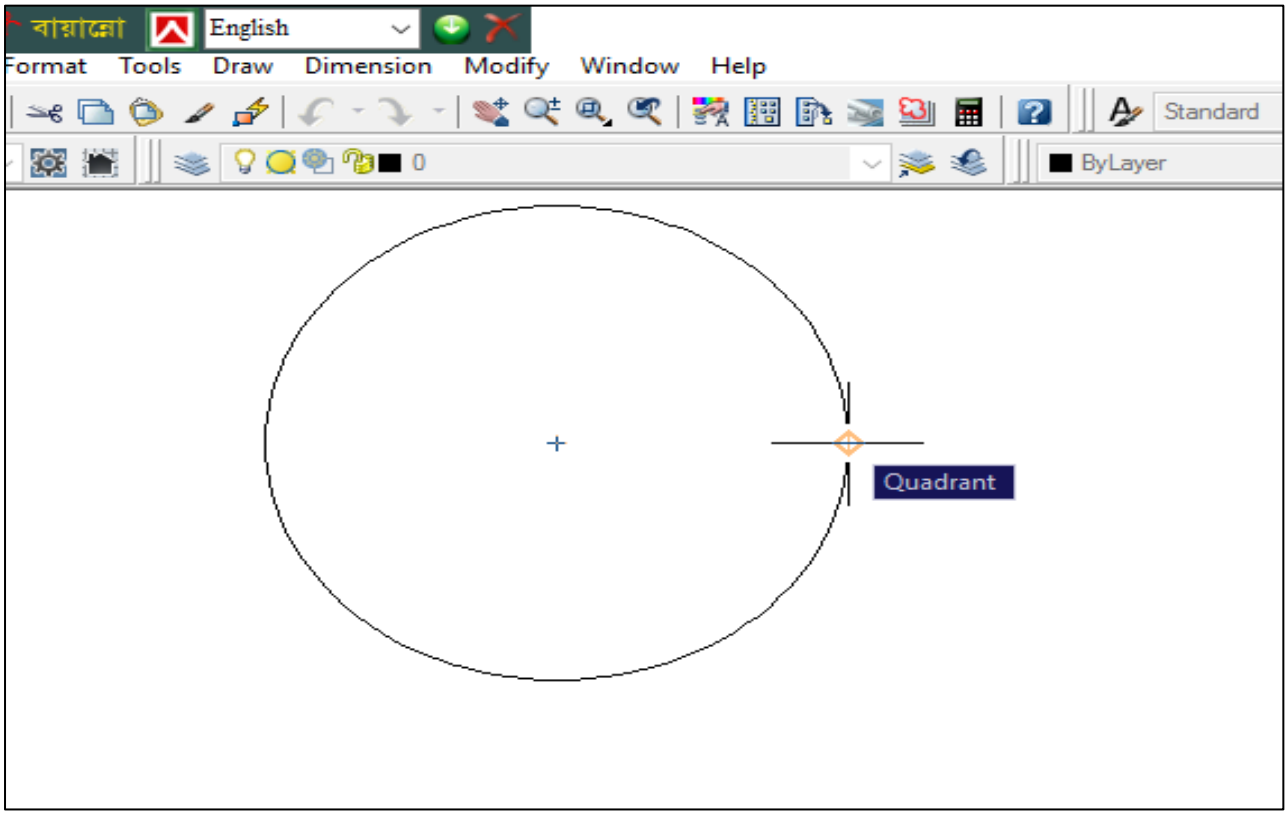


Fig: Quadrant point

Intersection mode: Snaps at the intersection point of arc, ellipsoidal arc, line, multiline, polyline segment, solid polyline or x-line.

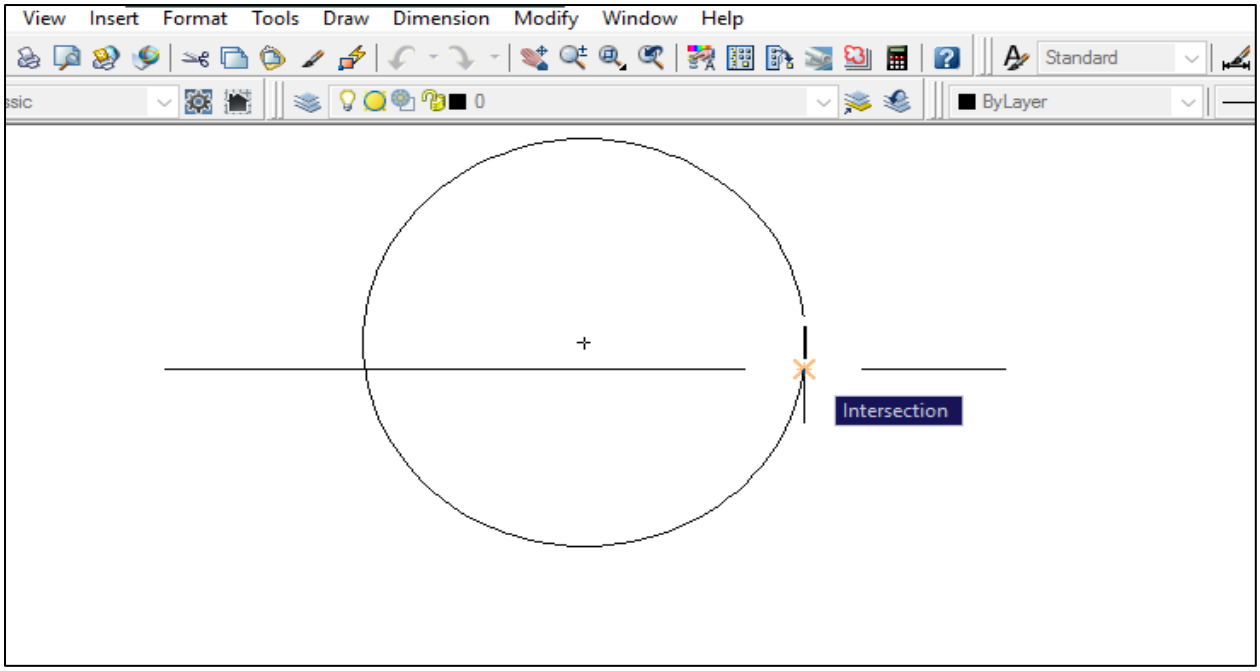


Fig: Intersection point

Extension mode: When cursor exceeds end point of any object, there exhibits a temporary extended line.

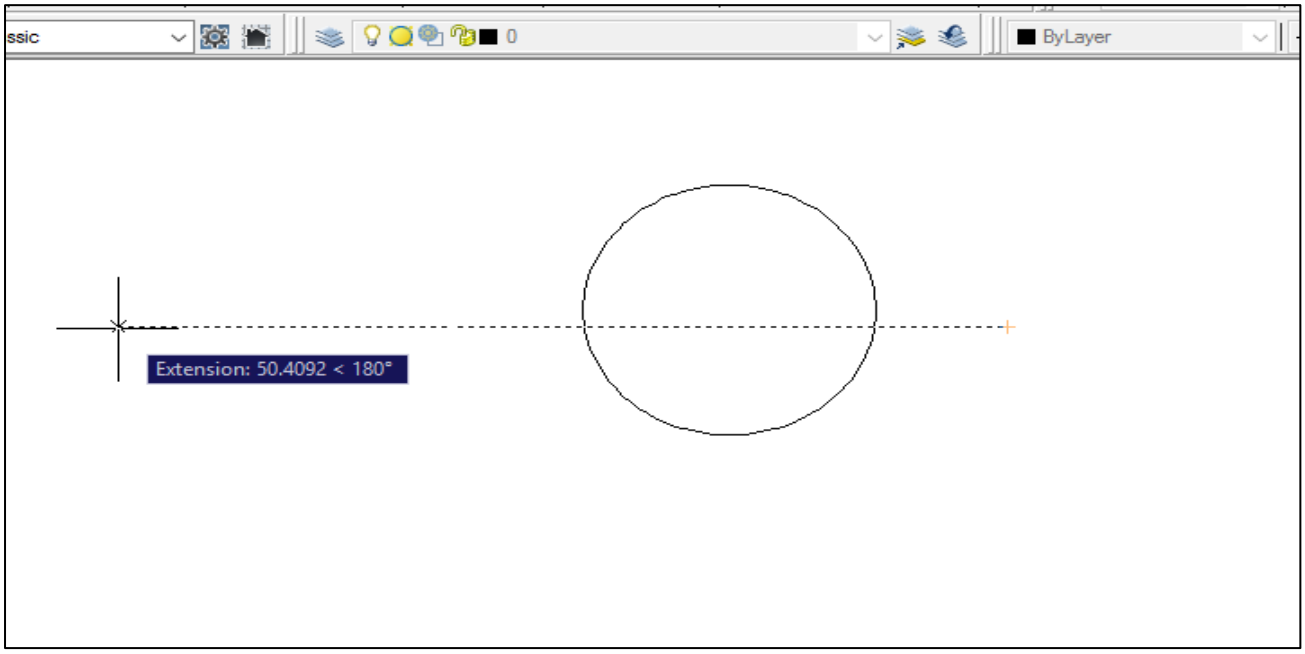


Fig: Extended line

Tangent mode: Snaps the tangent of any arc, circle, ellipse or ellipsoidal arc.

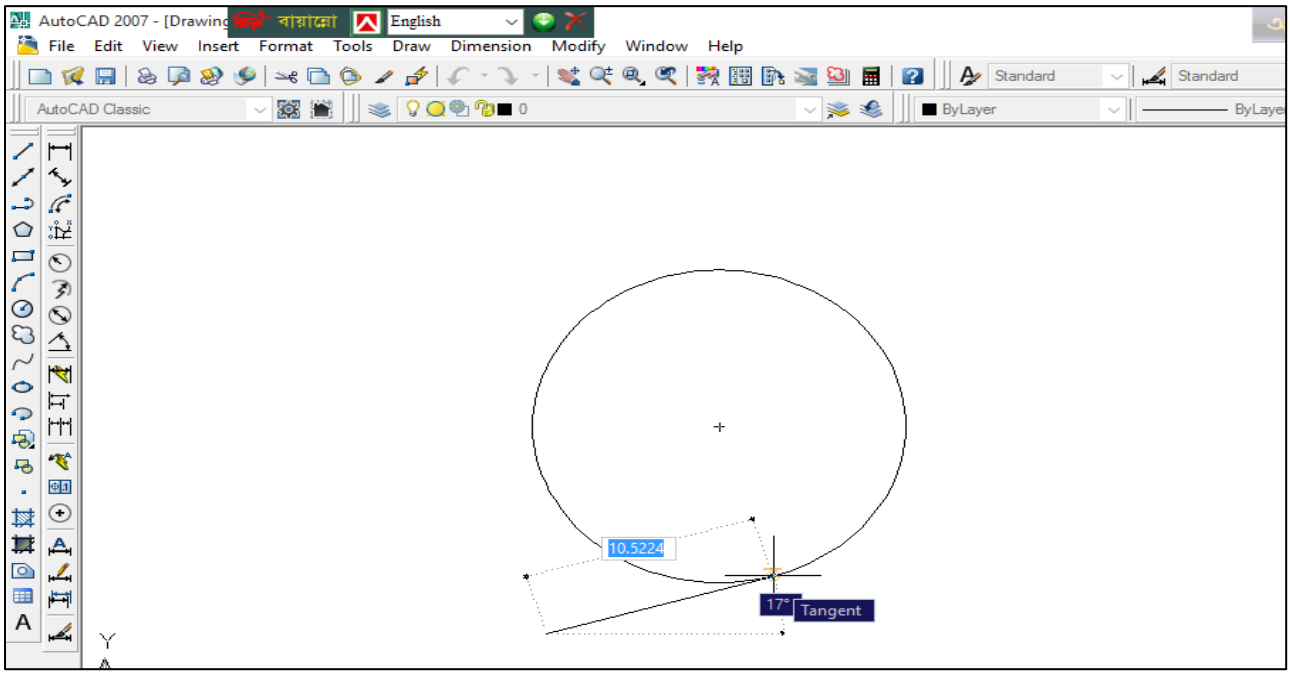


Fig: Tangent of a circle

Nearest mode: Snaps at the nearest point of arc, ellipsoidal arc, line, multiline, polyline segment, solid polyline or x-line.

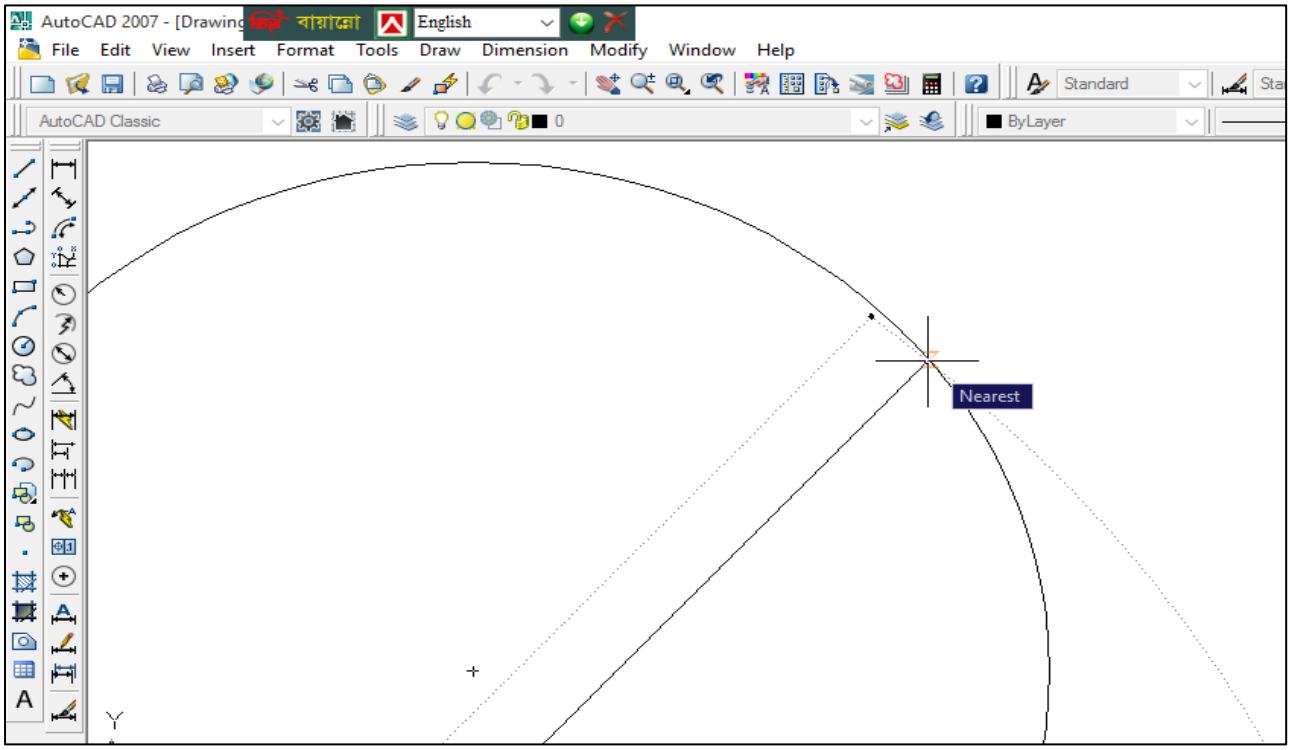


Fig: Nearest of a circle

DRAW TOOLBAR

Draw toolbar contains various commands which are used for drawing lines, polylines, circles, arcs, rectangles etc. It can be accessed by going to the Draw menu or by right clicking the upper right portion of AutoCAD to access ACAD from where draw toolbar can be enabled. . For ease of use pull the toolbar at one side of the screen to pin the toolbar.

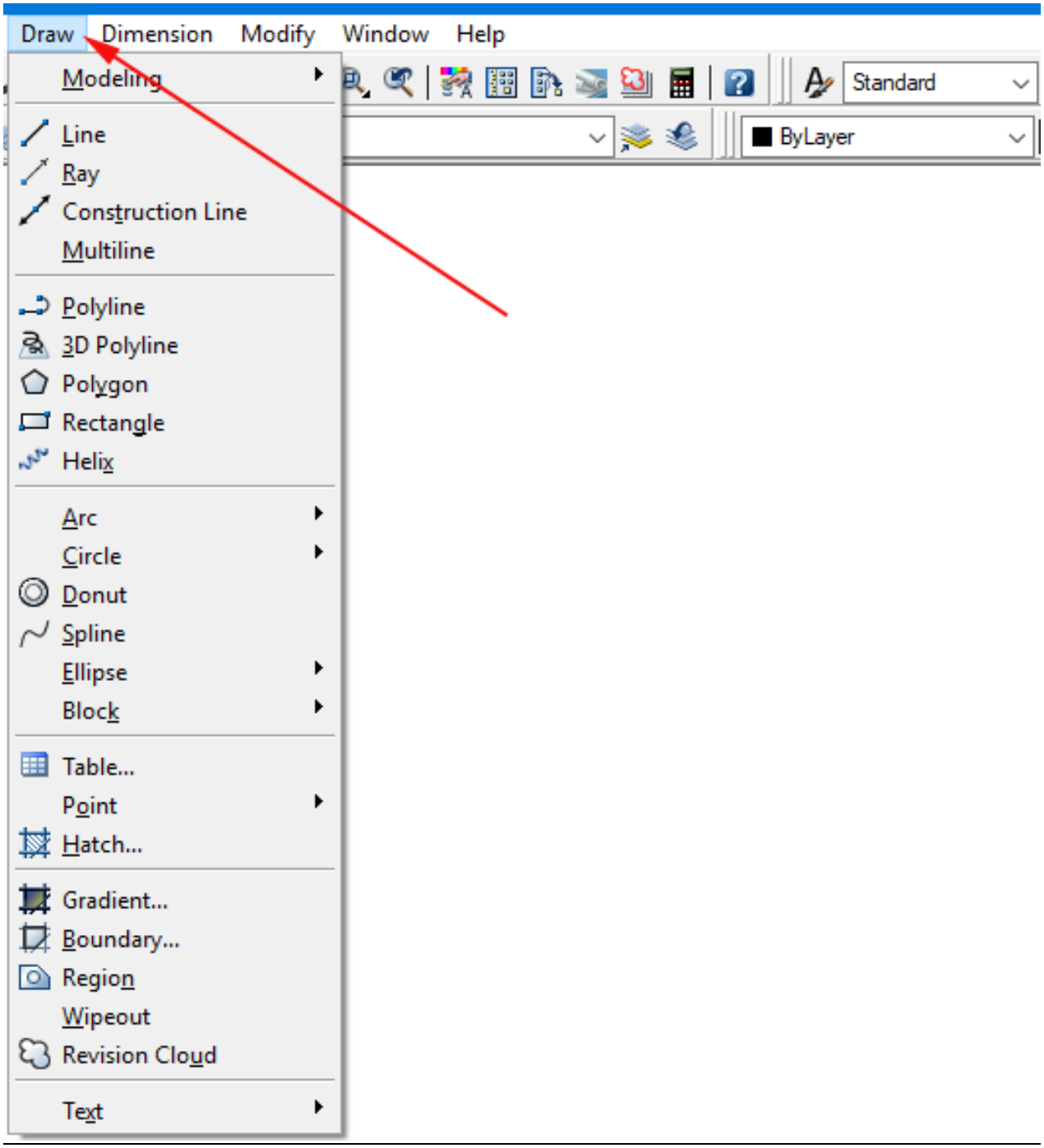


Fig: Draw menu and list of commands

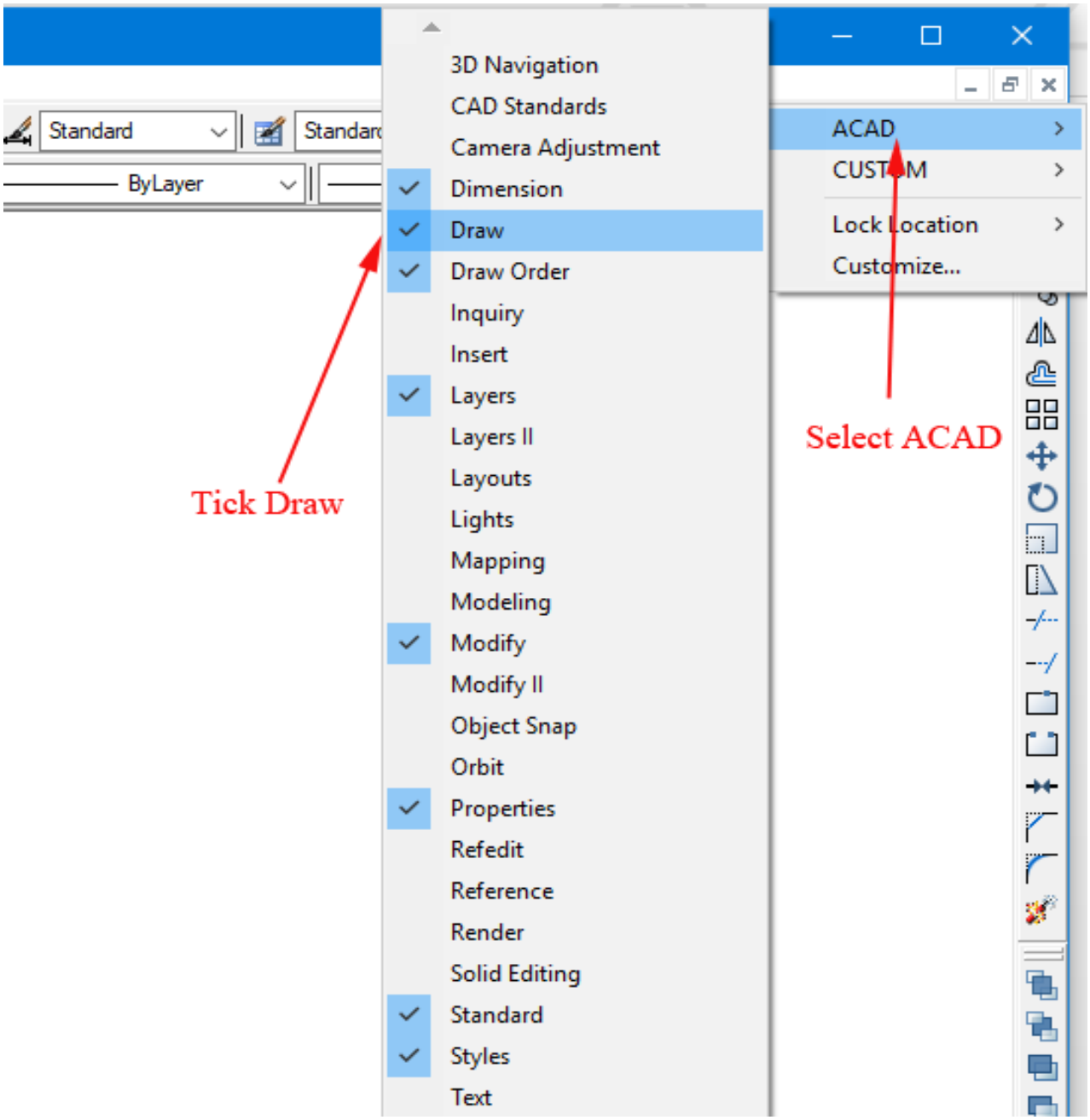


Fig: Enabling draw toolbar

Fig: Draw toolbar

Command Name: Line

Command Shortcut: L

Draw Toolbar Icon: 

Draw Menu Name: Line

Description: As the name suggests line command is used for drawing lines. By activating this command once a group of connected lines can be drawn. Lines can be drawn with precision with the help of point coordinates, object snap, grid snap etc.

Use:

How to draw lines without angle:

1. First the Line command is selected by either selecting from the toolbar or by entering **L**
2. The first point is selected by left clicking, then the end point is selected by left clicking or by entering the desired length.

How to draw lines with angle:

1. First the Line command is selected by either selecting from the toolbar or by entering **L**
2. The first point is selected by left clicking.
3. **@length<angle** is typed for creating a line with length.

How to draw with measurements:

1. Enter the line command by typing **L** and then pressing **enter.**
2. Select the first point
3. Now type the desired length and press enter. The line will have the specified length.

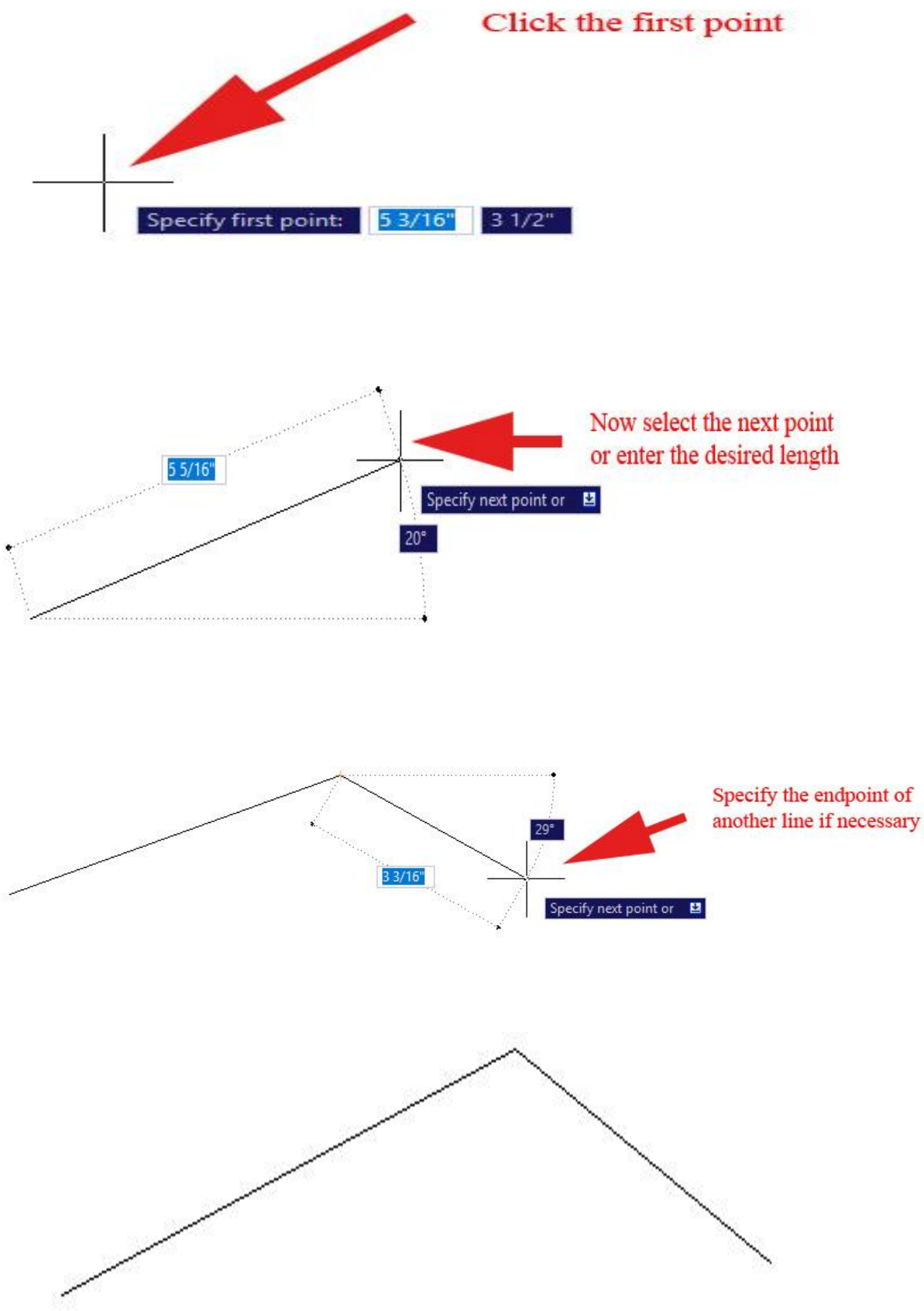


Fig: Drawing Lines

Command Name: Xline

Command Icon: 

Command Shortcut: XL

Description: This command is used to draw construction lines. Construction lines are either straight lines that extend to infinity in both direction. Construction lines are temporary line work entities that can be used as references when creating and positioning other objects or line work.

Use:

To draw a construction line through a point:

1. Enter the command **xl**
2. Specify the first point
3. Specify the second point then press enter

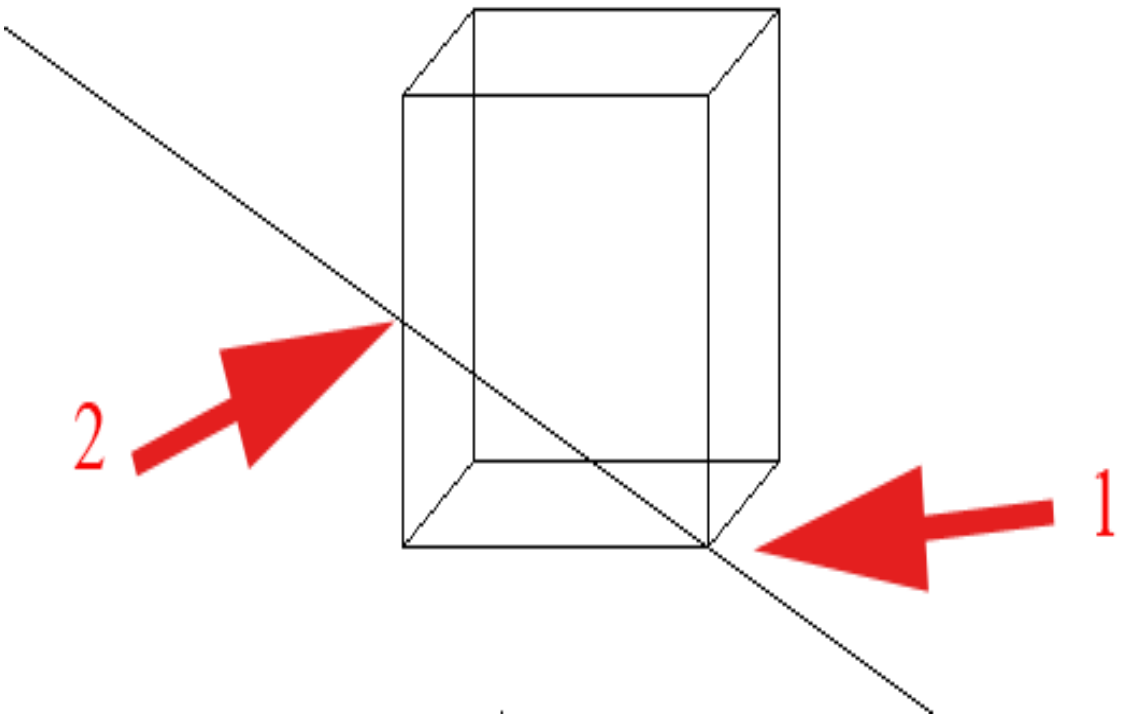


Fig: Construction line through a point

For a horizontal or vertical construction line:

1. Enter the command **xl**
2. Enter **H** for horizontal construction line or **V** for vertical construction line
3. Specify the point.

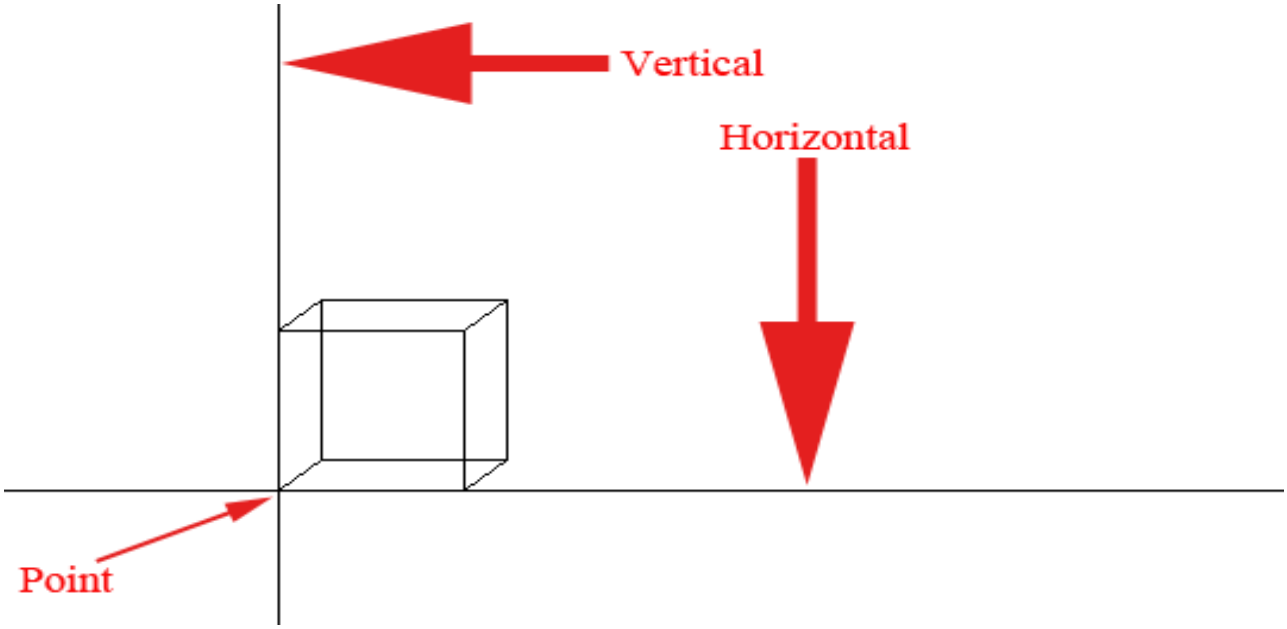


Fig: Vertical and Horizontal construction line

For a parallel construction line:

1. Enter the command **xl**
2. Type **O** press Enter
3. Specify the required distance
4. Select the line that you want to offset
5. Specify the side on which you want the line to be created

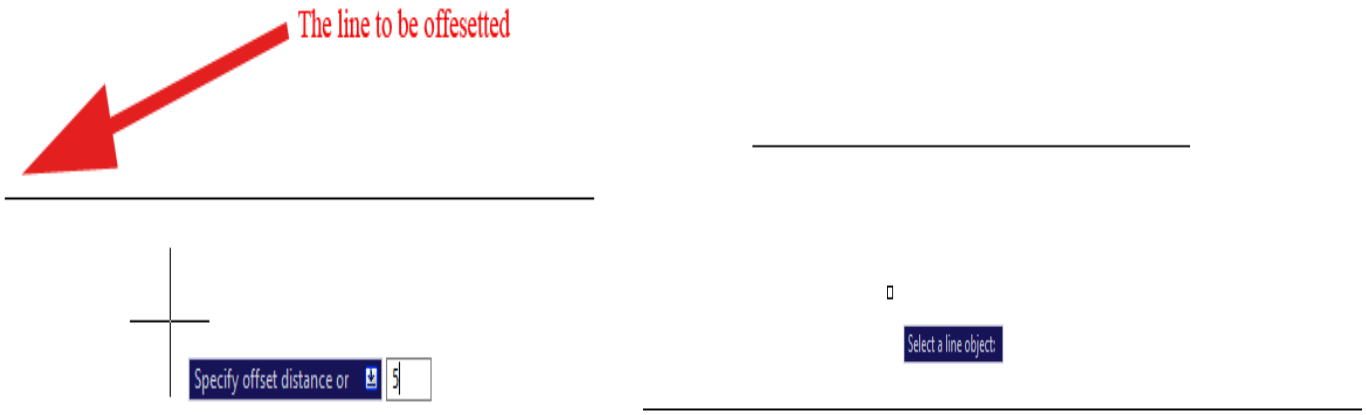


Fig: Drawing parallel construction line

Command Name: Polygon

Command Shortcut: pol

Draw Toolbar Icon: 

Draw Menu Name: Polygon

Description: Polygon is a collection of closed polylines. Polygon is used for drawing triangles, rectangles, pentagons, hexagons etc. This command can be used to draw a polygon with up to 1024 straight sides. It is most useful in mechanical drawings.

Use:

How to draw a polygon:

1. Select the polygon command and enter the numbers of sides.
2. Now select the center of the polygon
3. Choose **inscribed in circle** by pressing enter or choose **circumscribed about circle** by pressing **c** then enter.
4. Enter the desired radius of the circle

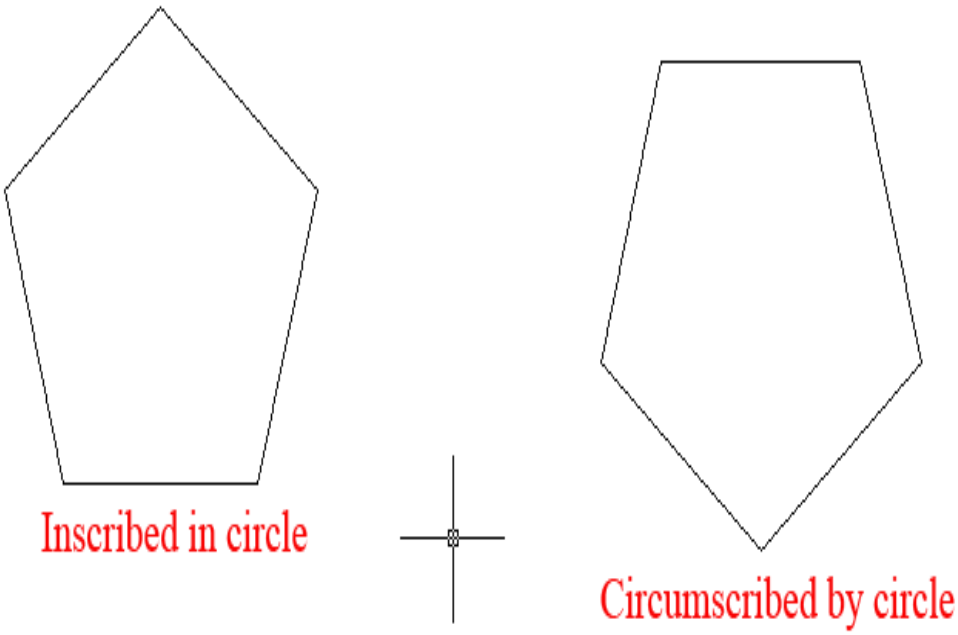


Fig: Polygons

Command Name: Rectangle

Command Shortcut: rectang or rec

Draw Toolbar Icon: 

Draw Menu Name: Rectangle

Description: Rectangle command is used for creating rectangles. It is mainly used in geometrical, architectural and civil engineering drawings.

Use:

How to use the rectangle command and its various options:

1. Select the rectangle command.
2. Specify the first point to create a normal rectangle or press **C** for **chamfer** or press **F** for **Fillet.**
3. Follow the on-screen commands and finish drawing.

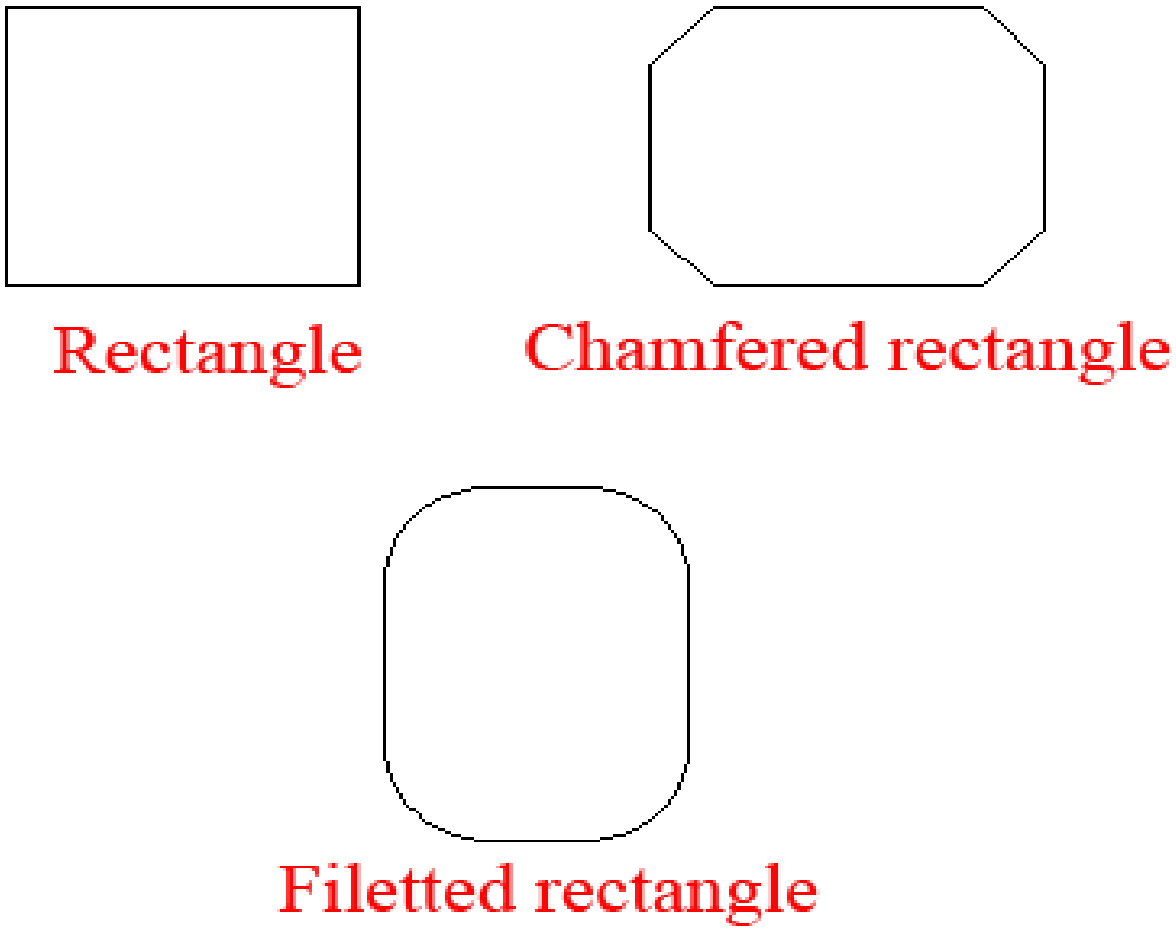


Fig: Different rectangles

Command Name: Circle.

Command Shortcut: Circle or c⏎

Draw Toolbar:

Draw Menu Name: Circle.

Description: To create a whole circle we can use circle command in auto cad. Circle command is mostly use in mechanical drawing. We can create a circle by using three points which are not in same line. There are some other methods to create a circle.

Use:

Using three points to create a circle:

1. In draw toolbar click the circle icon or enter **C** in the command line.
2. Write 3P & then press Enter.
3. Select the first point of the circle.
4. Select the second point of the circle.
5. Select the third point of the circle & then draw the circle.

Using two points to create a circle:

1. In draw toolbar click the circle icon.
2. Write 2P & then press Enter.
3. Select the first corner point of diameter of circle.
4. Select second corner point of diameter of a circle.

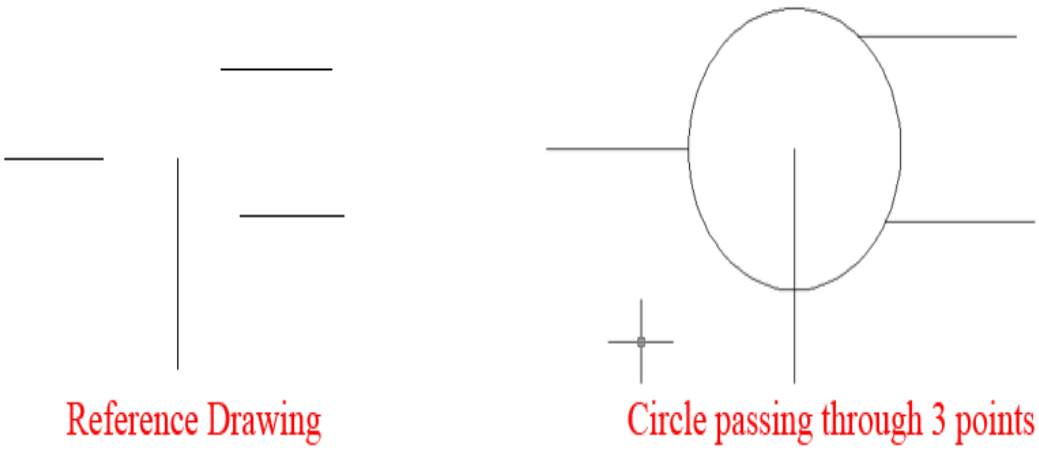


Fig: Drawing a circle passing through 3 points

Command Name: Arc.

Command Shortcut: arc or a⏎

Draw Toolbar: 

Draw Menu Name: Arc

Description: To create an arc in AutoCAD we can use the command arc. At first we can create an arc by specifying three points. We can create an arc using a start point, center & third point that determines the end point. Using different options, we can specify either the start point first or the center point first then arcs are drawn in a counter clockwise direction.

Use:

Using three points to draw an arc:

1. In draw toolbar click the Arc icon.
2. Select the starting point.
3. Select the second point of the arc.
4. Select the last point of the arc.

Using first point, center point & last point to draw an arc:

1. In draw toolbar click the arc icon.
2. Now select the first point.
3. Write **C** & press Enter.
4. Select the center point.
5. Select the last point.

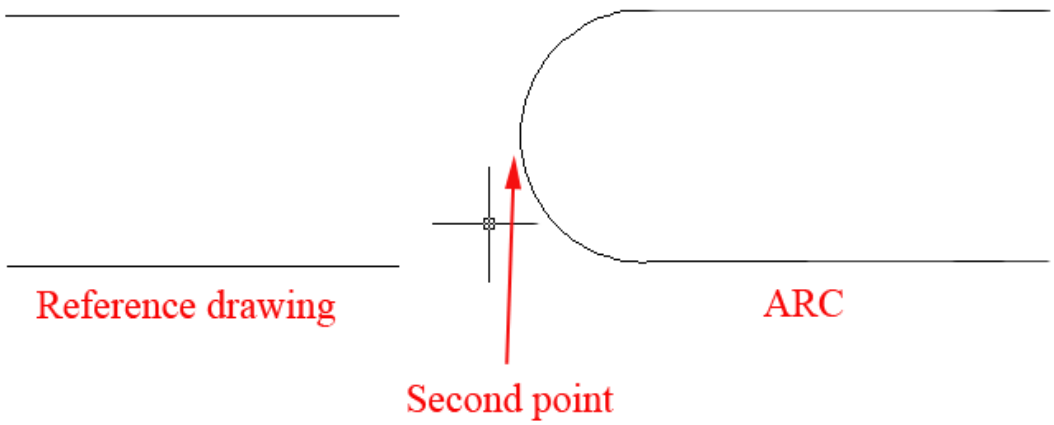


Fig: An arc passing through three points

Command Name: Multiline

Command Shortcut: mline or ml⏎

Draw Menu Name: Multiline

Description: Multiline command is used for drawing multiple parallel lines. Multiline style is set to Standard by default. New multiline style can be loaded from Format → Multiline.

Use:

1. Select multiline from the draw menu or type **ML** or **Mline** in the command line.
2. Specify the start point or enter **J** for Justification, **S** for scale, **ST** for style.

   Justification Option: Used for controlling the double line

   Scale Option: Used for setting the distance between the two lines

   Style Option: Used for editing the style of the lines

3. Specify the second point and other points as necessary.

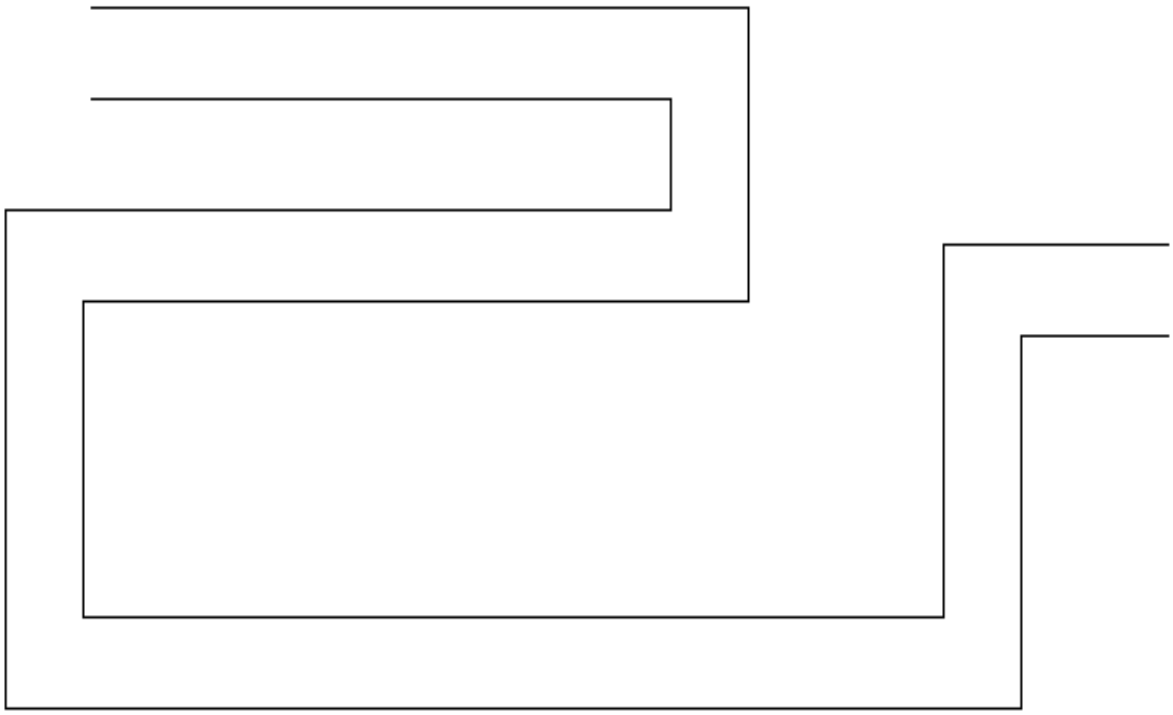


Fig: Multiline

Command Name: Point

Command Shortcut: po ↩

Draw Toolbar Icon:

Draw Menu Name: Point

Description: This command is used to draw a point. A point has no height, breadth and width. It can only be selected, moved and copied.

Use:

Procedure for single point:

1. Select **Point → multiple point** from the draw toolbar or by entering the command **PO** and press enter.
2. Click the point where you want a single point.

Procedure for multiple points:

1. Selecting Point icon from draw toolbar or by **Draw menu → Point → Multiple point**
2. To draw necessary points.
3. Pressing Esc button the command is terminated.
4. If the points are **not visible** enter the command **Blipmode** then click **ON.** The points will become visible and take a "+" shape.

Fig: Single command and multiple command

Procedure for changing the point style:

1. By default point is a dot but AutoCAD has various other types of points.
2. Go to **Format Menu→ Point Style** and select the desired type and size from the dialog box
3. Point size can be changed by entering command **pdsize** and entering the value. The style of the point can also be changed by entering the command **pdmode** and then entering the desire value for the style from the list given below-

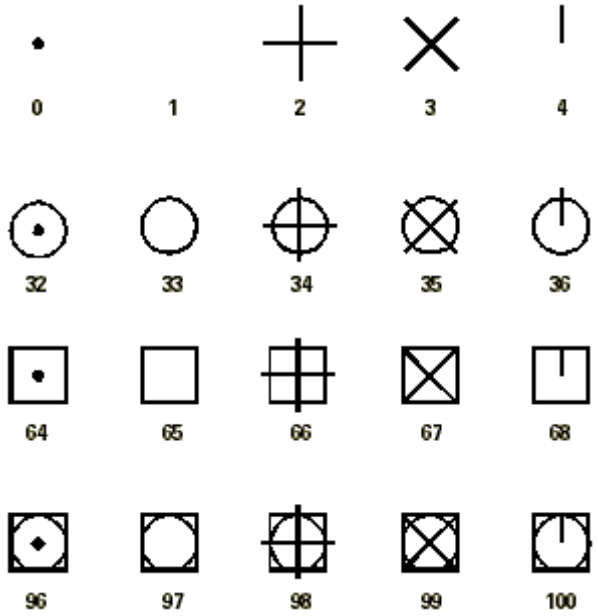


Fig: Values for PDMODE command

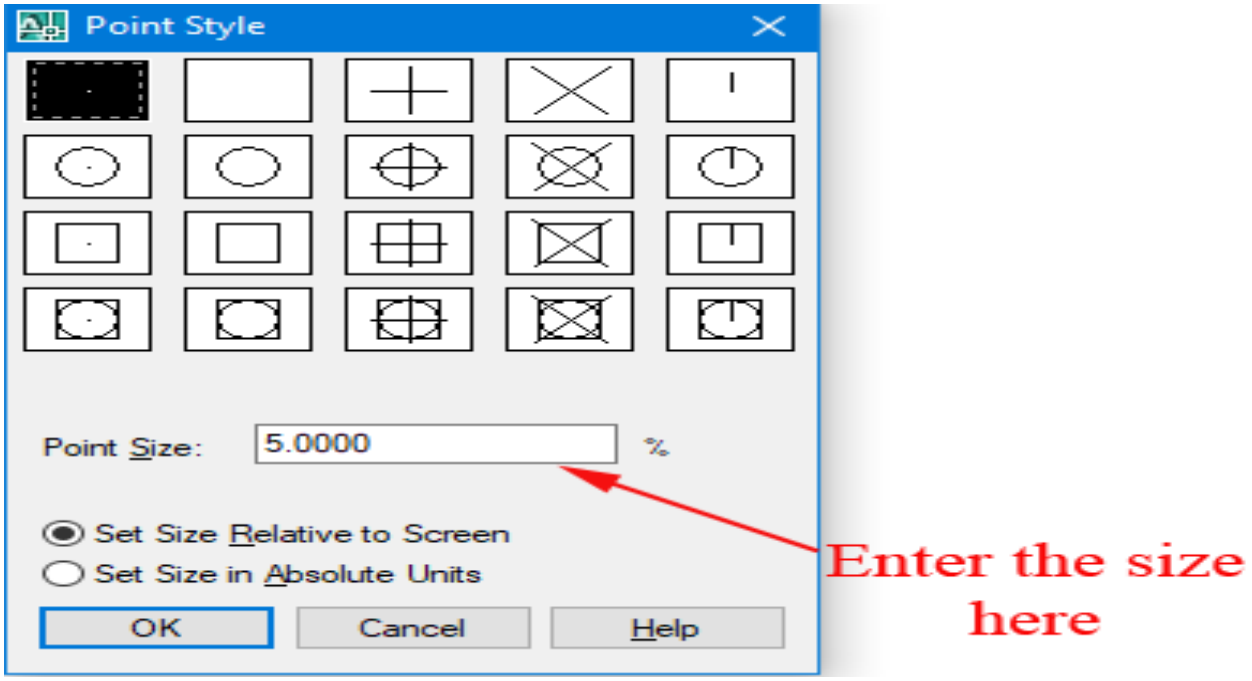


Fig: Point Style Window

Command Name: **Measure**

Command Shortcut: **Me ↩**

Draw Menu Name: Point → Measure

Description: Measure (Command) Creates point objects or blocks at **measured** intervals along the length or perimeter of an object. The points or blocks are placed in the previous selection. You can use the Node object snap to draw an object by snapping to the point object.

Use:

1. Specify interval distance.
2. Type Me in command line and click enter
3. Select Line, Arc, Circle, Polyline etc.

Example:

- **Command Line:** Me ↩
- **Select object to measure:** select the line
- **Specify length of segment:** click point A
- **Specify second point:** click point B

Fig: Reference Drawing

Fig: After using measure command

Command name: Block making

Command Shortcut: B ↩

Draw Toolbar Icon:

Draw Menu Name: Block → Make

Description: Blocks are objects that made using multiple lines. In architectural drawing door, window, furniture etc. are converted into blocks as they are used multiple times. AutoCAD has some preinstalled blocks which are described later in **DesignCenter** command.

Use:

1. Selecting Make block icon from draw toolbar.
2. Writing name on **'Block definition'** dialogue box.
3. By clicking Select objects to select the object and right click on mouse.
4. By cling Pick point we take the base point.
5. And then clicking Ok.

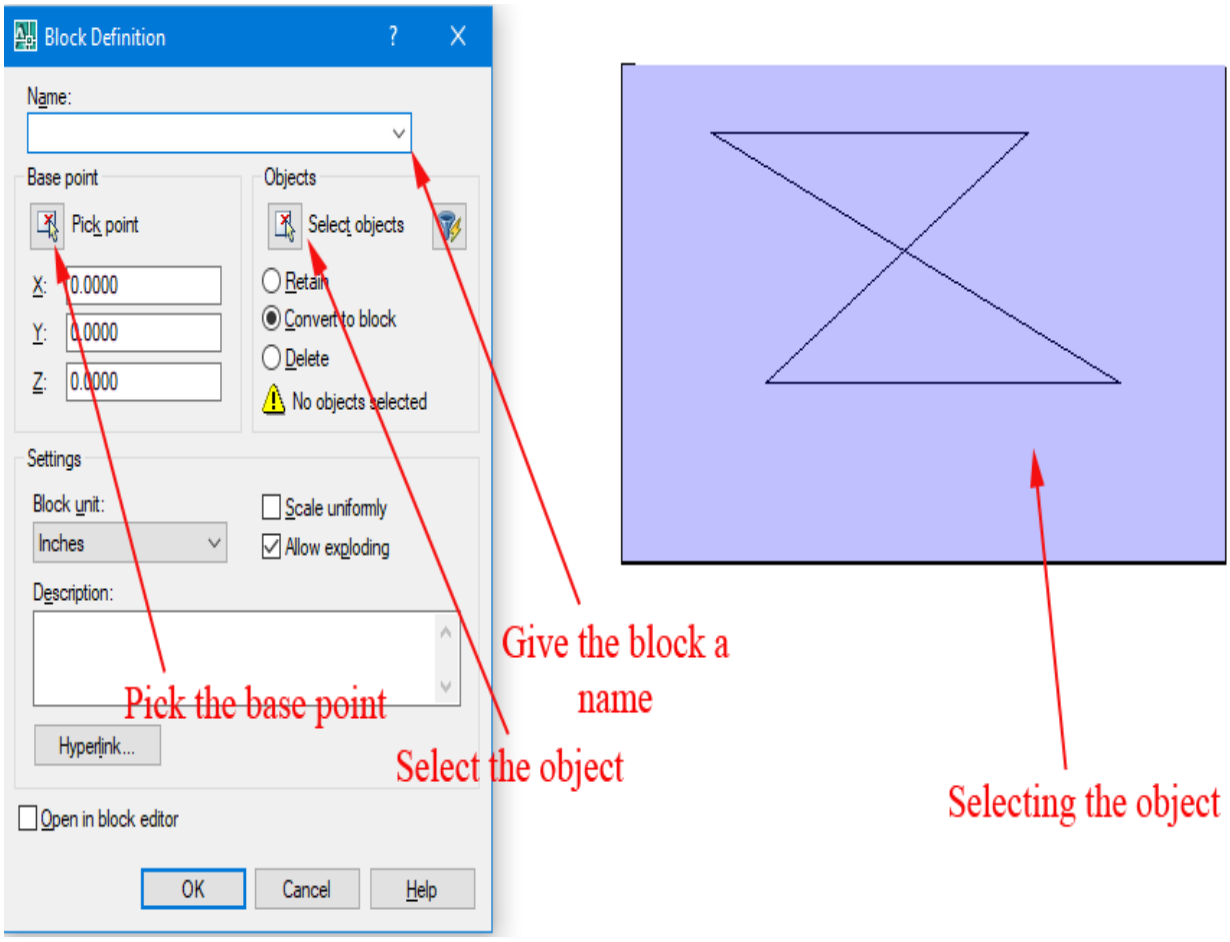


Fig: Making a block

Command Name: Block insert

Command Shortcut: I↩

Draw Toolbar Icon: 

Draw Menu Name: Insert

Description: This command is used for insert the created block.

Use:

1. We click the icon **'Insert block'** or enter the command **I.**
2. In the Name option, select the specific block or enter the name of the block.
3. In the dialogue box there are three specific zones called Insertion point, Scale and rotation. We can add value and angle on these. If left checked the scale can later be inserted by following the on screen commands.
4. If we want to break the object in case of inserting we will check the explode option.
5. And finally we click Ok.

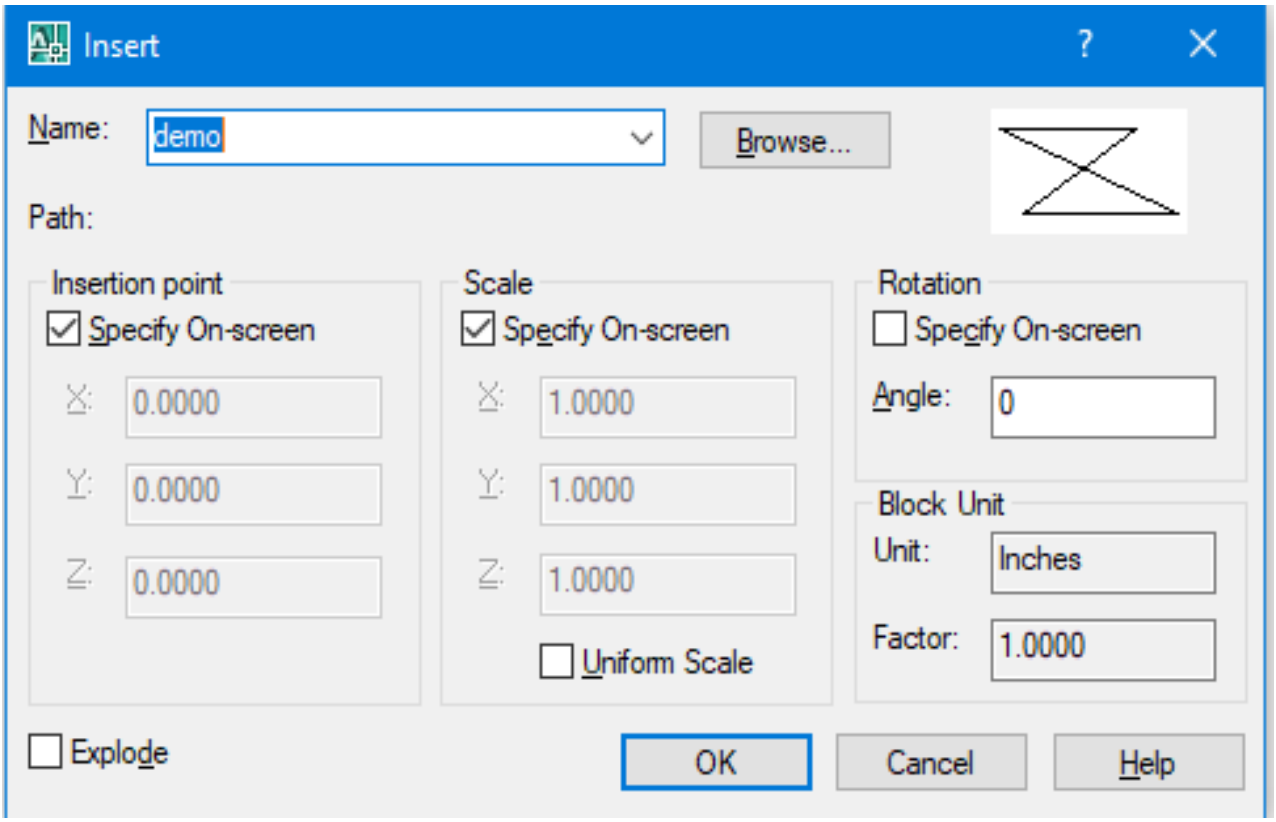


Fig: Insert Block window

Name: DesignCenter

Command Shortcut: ctrl + 2

Toolbar Icon: 

Draw Menu Name: DesignCenter

Description: DesignCenter is used for-

- Browsing for drawing and symbol library
- Updating block definitions
- Creating shortcuts for frequently used blocks
- Adding Furniture

Use:

1. Open the DesignCenter by pressing **CTRL & 2** at the same time
2. From the left of the panel the user can browse file directory
3. To add pre-installed furniture, which came with AutoCAD 2007 go to **Autocad2007 directory → Sample → DesignCenter**. Furniture can be found in **Home – Space Planner.dwg, House Designer.dwg, Kitechens.dwg** etc. folder inside Blocks

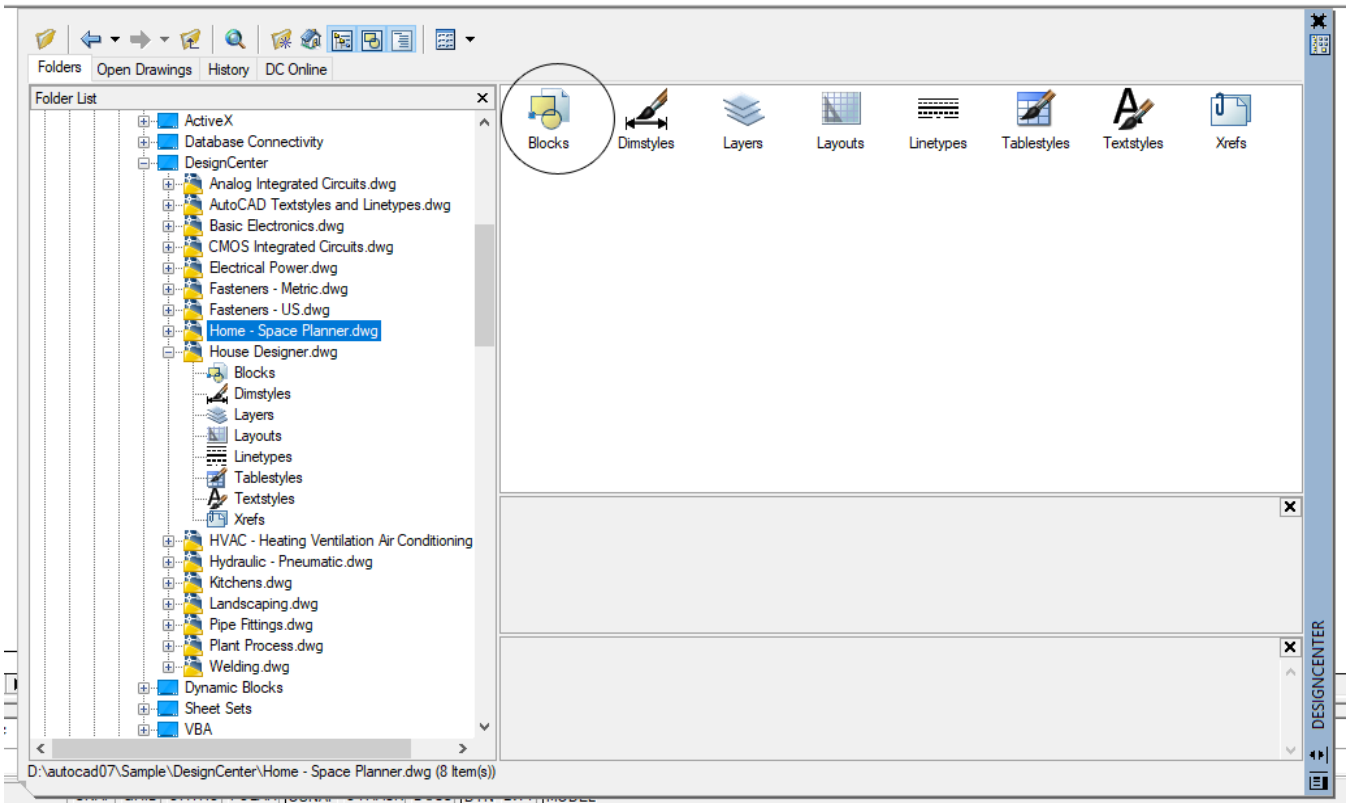


Fig: DesignCenter

Command Name: Hatch

Command Shortcut: h⏎

Draw Toolbar: 

Draw Menu Name: Hatch

Description: The process of filling a empty space in a drawing using patterns or solid color is known as hatching. When drawing a plan layout or an elevation of a building various objects such as- doors, windows, roof, stairs etc. are hatched to make the plan more legible.

Use:

1. Type h in the command line.
2. The **Hatch and Gradient** window will open. Click the swatch box and select a pattern.
3. Click the **Pick points** button and select the area that you want to hatch or click **Add: select objects** button to select individual objects.
4. Clicking the **Preview** button to see the preview then adjust the angle and scale as necessary. Finish hatching by clicking **OK.**

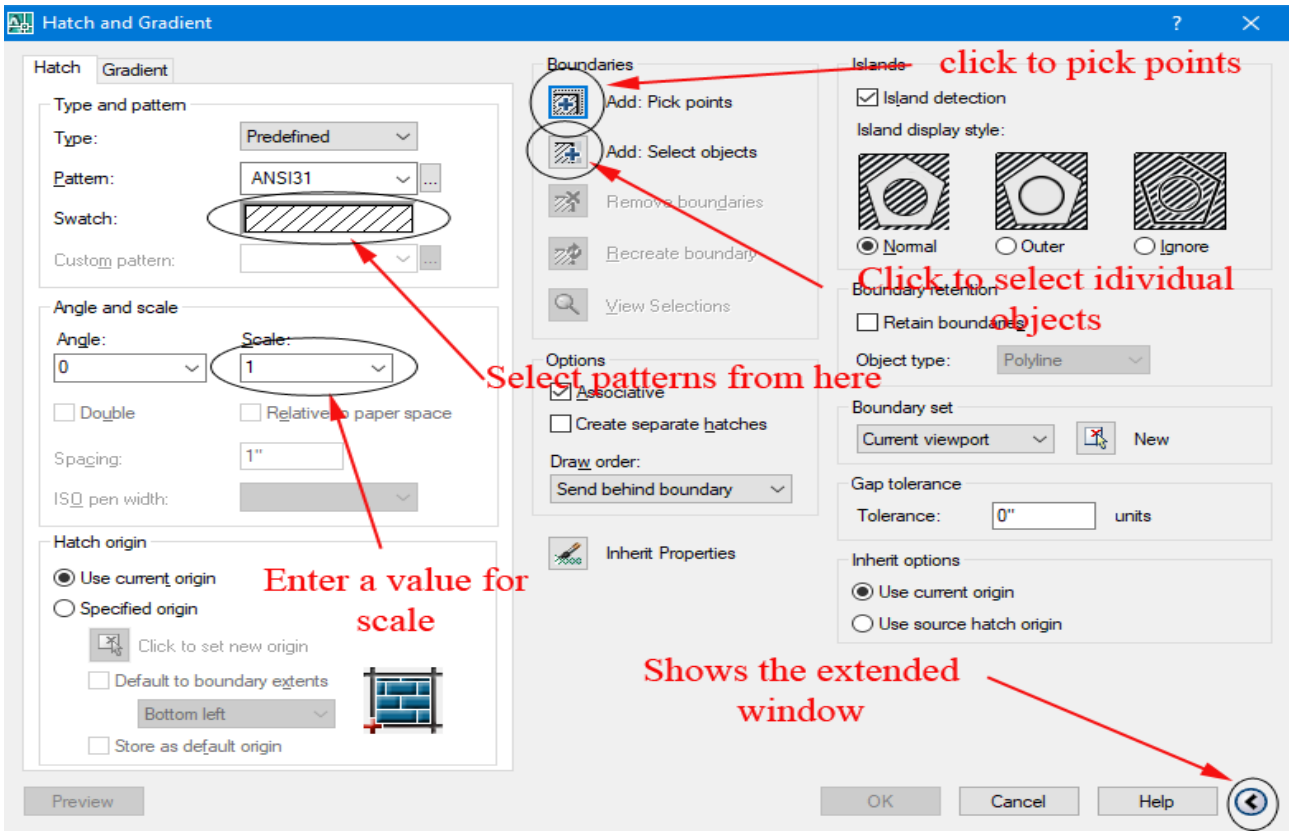


Fig: Hatch and Gradient window

Command Name: Gradient Filling

Command Shortcut: h↵

Draw Toolbar:

Draw Menu Name: Gradient

Description: Gradient filling or simply Gradient is a type of filling which is used for showing reflection of light. It is used for giving object metallic look or glassy texture in drawing. One or two colors are used in gradient.

Use:

1. Type h in the command line.
2. The **Hatch and Gradient** window will open. Now click the **Gradient** tab on the upper left side of the window.
3. Click the **Pick points** button and select the area that you want to hatch or click **Add: select objects** button to select individual objects.
4. In **color** there is option for select **one color** or **two color,** a dropdown list for selecting color and a slider for adjusting the amount of tint
5. Clicking the **Preview** button to see the preview then adjust the angle and orientation as necessary. Apply gradient by clicking **OK.**

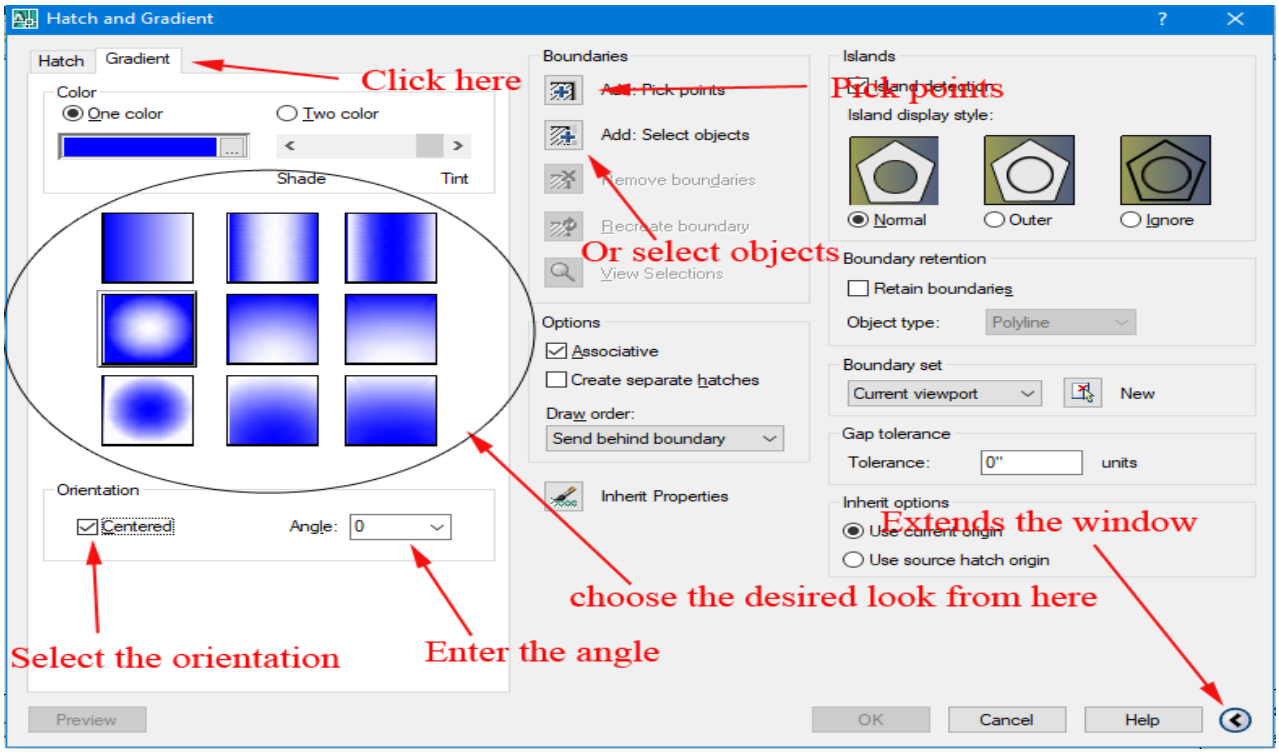


Fig: Gradient tab

MODIFY TOOLBAR

Modify toolbar contains various commands which are used for carrying out various modification on drawings. Some examples are- Erase, Mirror, Array, Trim etc. commands. It can be accessed by going to the Modify menu or by right clicking the upper right portion of AutoCAD to access ACAD from where modify toolbar can be enabled. For ease of use pull the toolbar at one side of the screen to pin the toolbar.

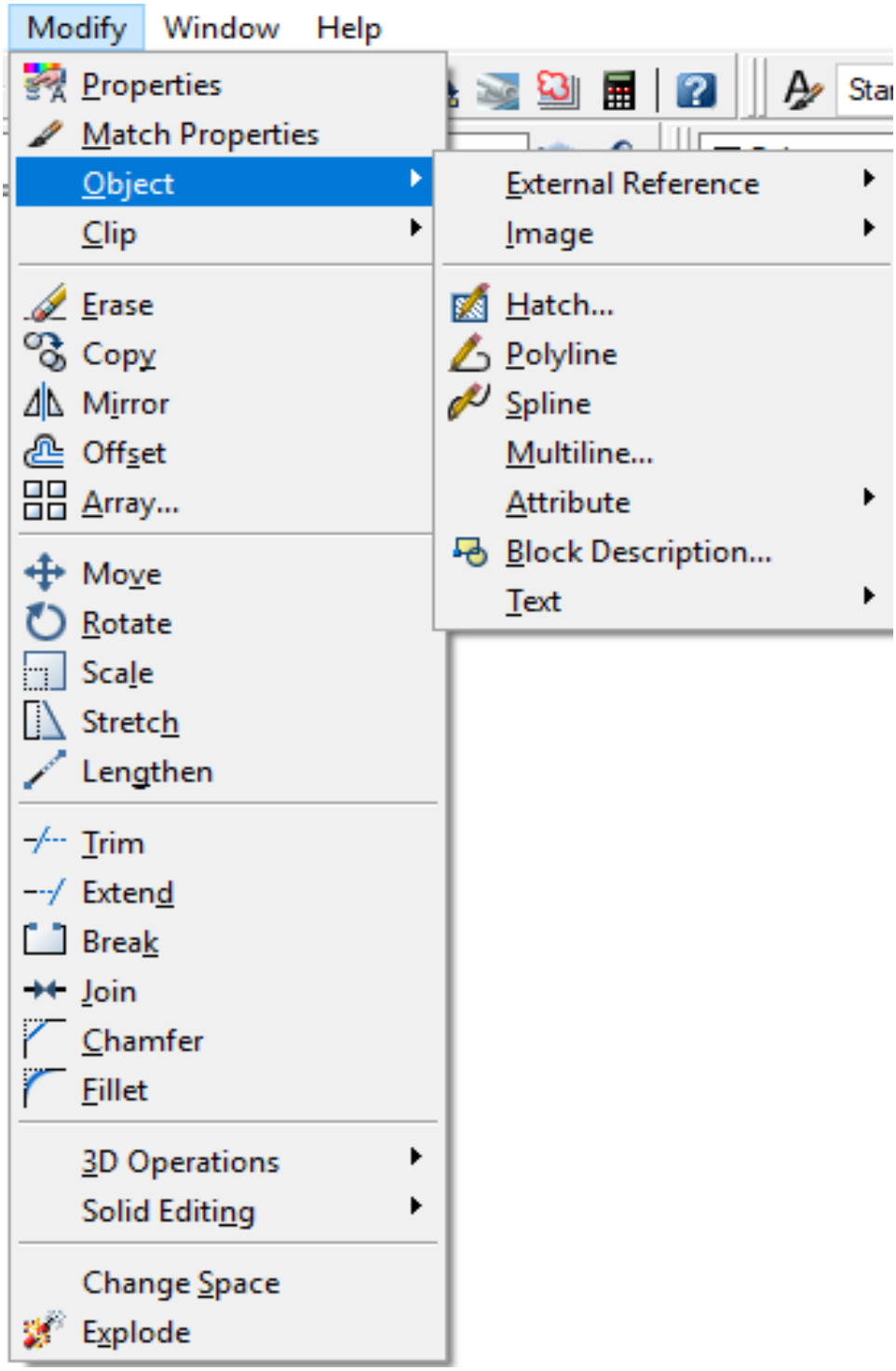


Fig: Modify Menu

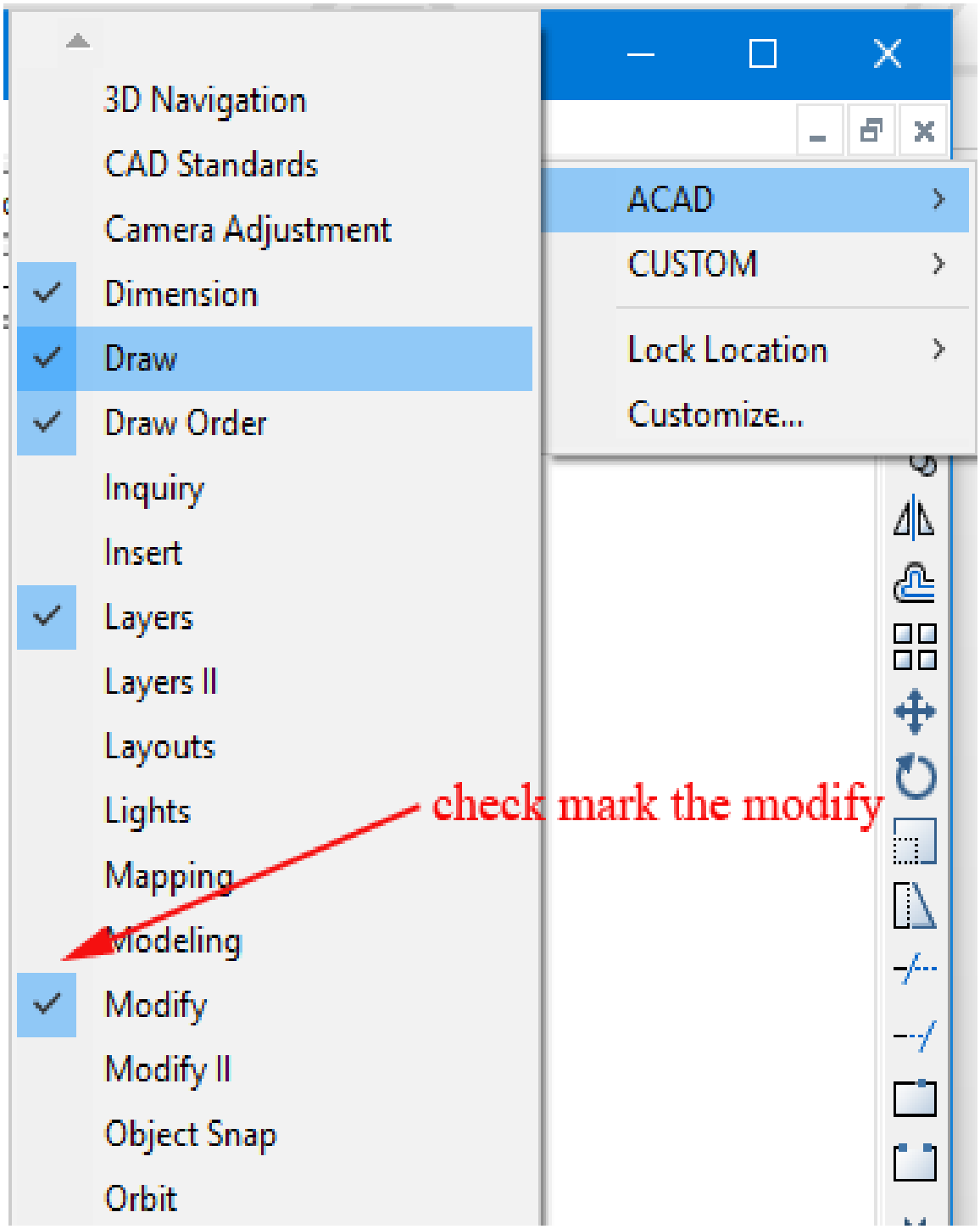


Fig: Enabling the modify toolbar

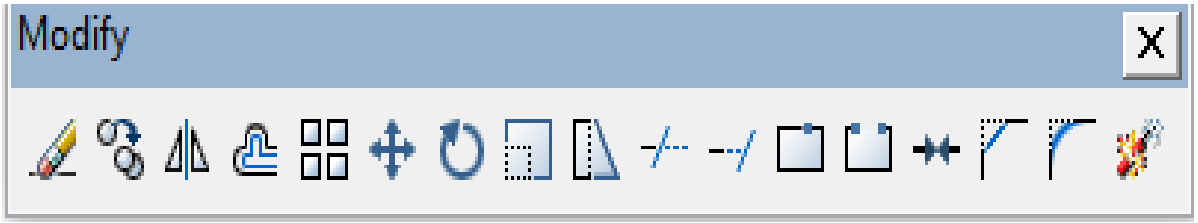


Fig: Modify Toolbar

Command Name: Erase

Command Shortcut: E ⏎

Modify Toolbar Icon: 

Modify Menu Name: Erase

Description: Erase is the command that allows to erase objects in AutoCAD. But, this command will erase the whole object instead of removing parts of it. This command removes a selected group of entities, which may be entered before or after the command itself is entered.

Use:

1. Click on the **ERASE** icon.
2. Select the object to erase.
3. Press the **ENTER** key.

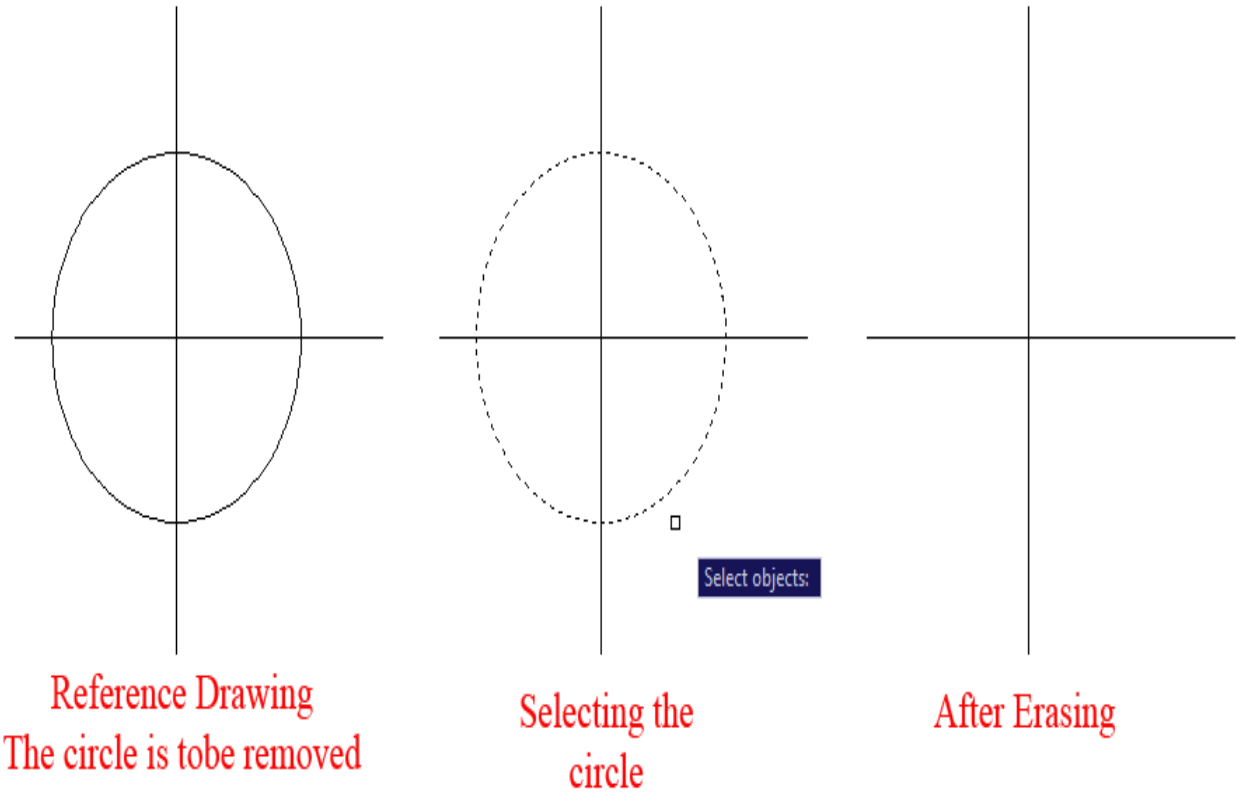


Fig: Erase Command Demonstration

Command Name: Copy

Command Shortcut: Co ⏎

Modify Toolbar Icon: 

Modify Menu Name: Copy

Description: Copy is the one of most useful command in AutoCAD. As the name suggest it is used making multiple copies of selected objects.

Use:

1. Click on the **COPY** icon.
2. Select the object to copy.
3. Select the base point.
4. Finally select the position where you want to copy the object. Multiple positions can be selected to make multiple copies.

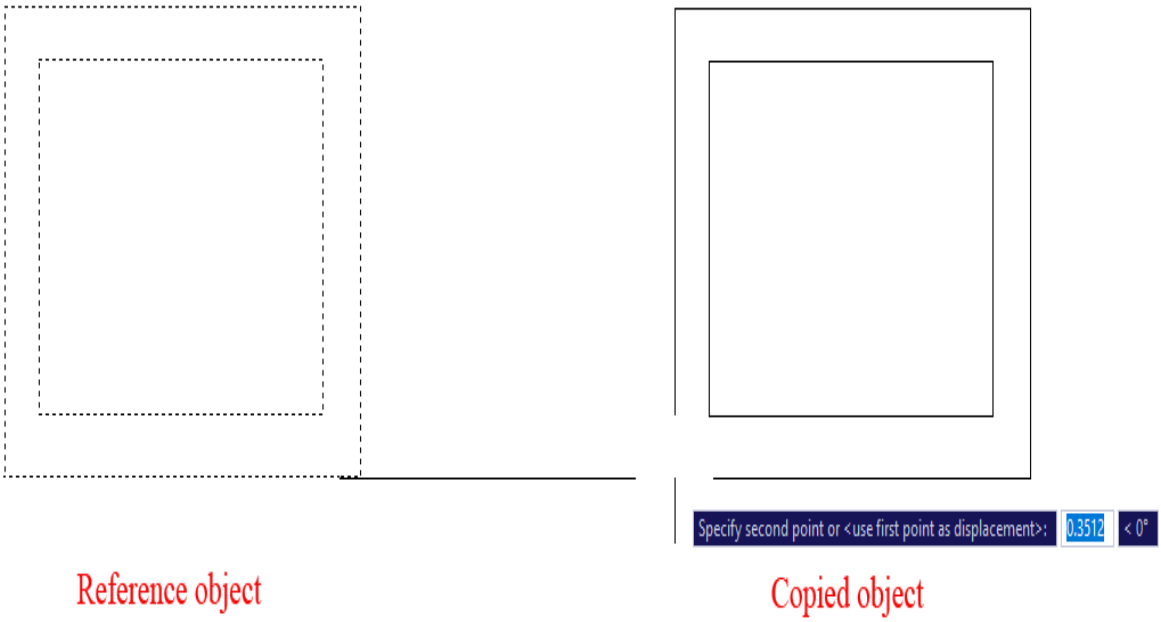


Fig: Copy command

Command Name: Mirror

Command Shortcut: MI ⏎

Modify Toolbar Icon: 

Modify Menu Name: Mirror

Description: Mirror command is used for creating mirror objects of an object or drawing. It is most commonly is used in architectural, mechanical and furniture drawings. Door is one of the most mirrored objects in AutoCAD as some doors open towards the right side while others open towards the left side.

Use:

1. Select objects
2. Click on the **MIRROR** icon
3. Specify the first point of mirror line
4. Specify the second point of mirror line
5. Press **Enter** to keep the original drawing or **Y** to erase the original drawing.

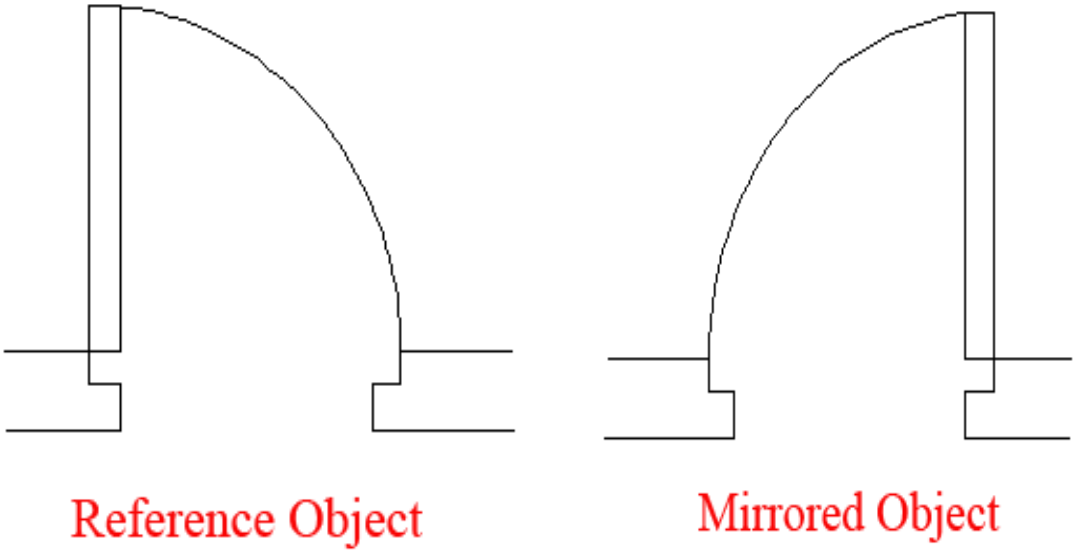


Fig: Mirror Command

Command Name: Offset

Command Shortcut: O ⏎

Modify Toolbar Icon: 

Modify Menu Name: Offset

Description: Offset command is used to create a line parallel to another at a certain distant or to create circles with variable radius while having same center point. A 10 inch thick wall can be easily drawn using offset command. In offset command distance between the lines and direction are to be specified.

Use:

1. From **Modify** toolbar click on the **Offset** icon.
2. Type offset distance and press enter key (for distance).
3. Move the mouse (pick box) over the line and click on it.
4. Move the mouse (cross hair) on right side of line and click.
5. Press **Enter Key** to terminate the command.

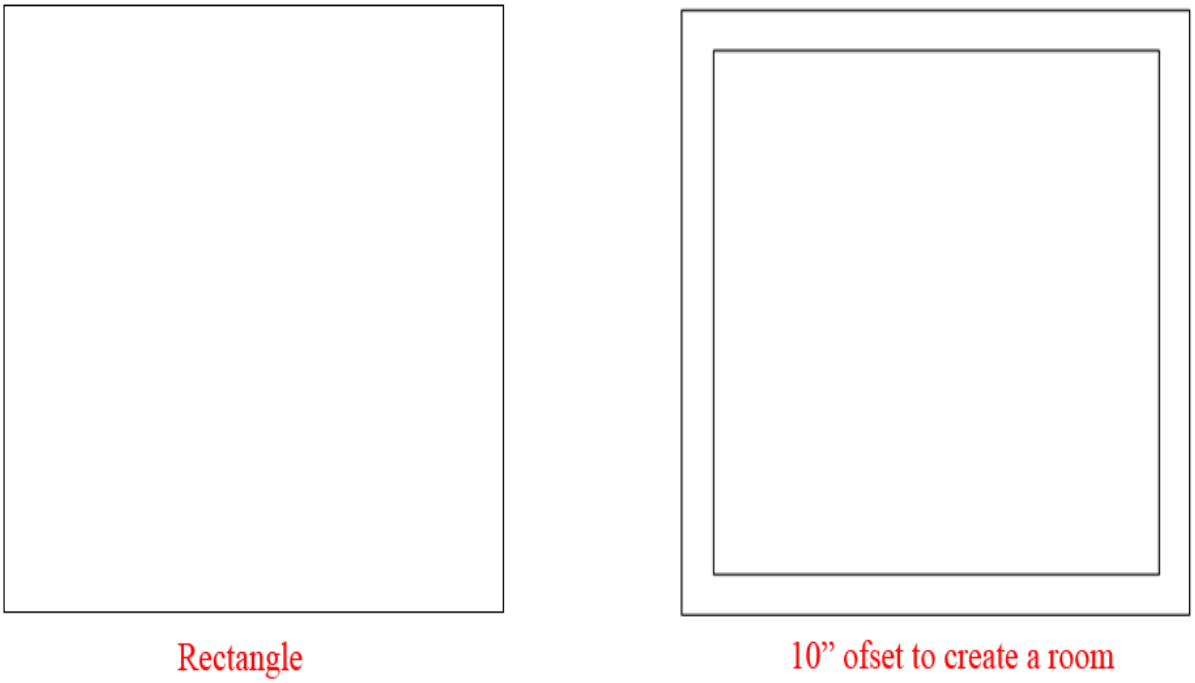


Fig: Creating a room using offset command

Command Name: Array

Command Shortcut: AR ↩

Modify Toolbar Icon:

Modify Menu Name: Array

Description: Array means arrangement. By the help of array command we can arrange an object in a specific arrangement and can take as many as copy we want. An object can be arranged in rectangular or circular form.

Use:

Rectangular array:

1. Click on the Array icon in the modify toolbar or enter the command **AR.**
2. In array dialogue box, click on the Rectangular array radio button. (Here it is elected by default)
3. Click on Select objects button and then select the object which you want to array.
4. Enter the number of rows and columns that you want in the respective boxes.
5. In the box of Row offset and Column offset, the distance between two rows and two columns should be written.
6. In the box of Angle of array, the specific angle should be written in which angle we want to build array the object.
7. Click on the **Preview** button to see whether the array is built properly or not. If it is not built properly, click on the modify button and come back to Array dialogue and change the value. If it is built properly, click on the **Accept** button.

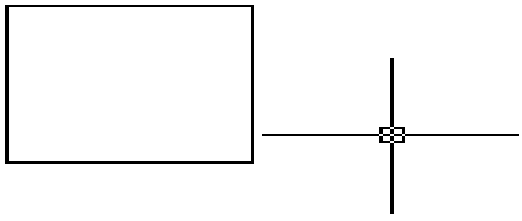


Fig: Reference Drawing

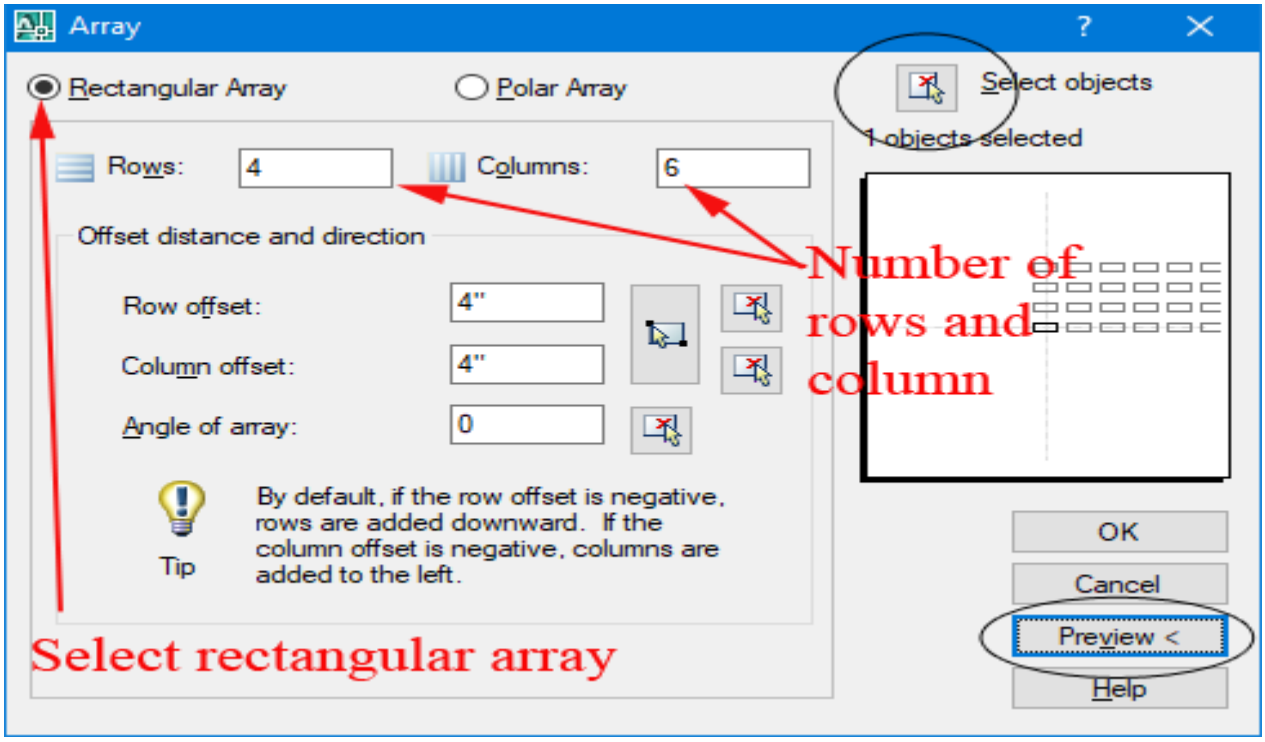


Fig: Array Dialogue box

Fig: Array Preview Dialogue box

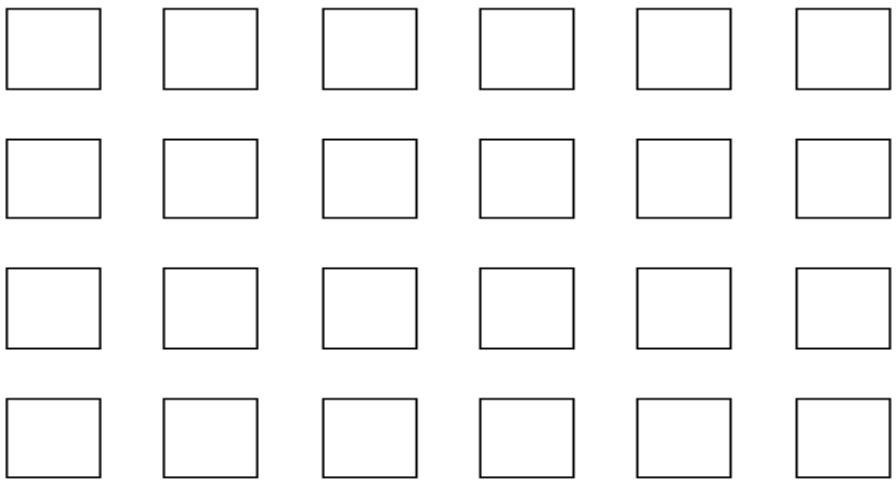


Fig: After Using Rectangular Array

Polar array:

1. Click on the Array icon in modify toolbar.
2. In array dialogue box, click on the Polar array radio button.
3. Click on the select objects button and then select the object which you want to array.
4. Click on the Pick center point button and select a specific point. The arrangement will be created around the point. You can also uphold the value of X and Y.
5. In the box of Total number of items, the number of object should be written.
6. In the box of Angle to fill, the array size should be specified by specified angle. In the counter clock-wise rotation, give a negative value and in the clock-wise rotation it will be positive.

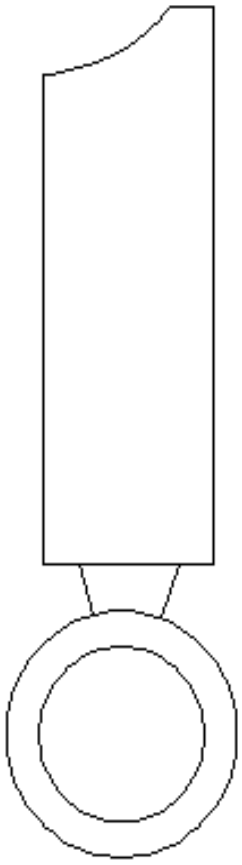


Fig: Reference Drawing

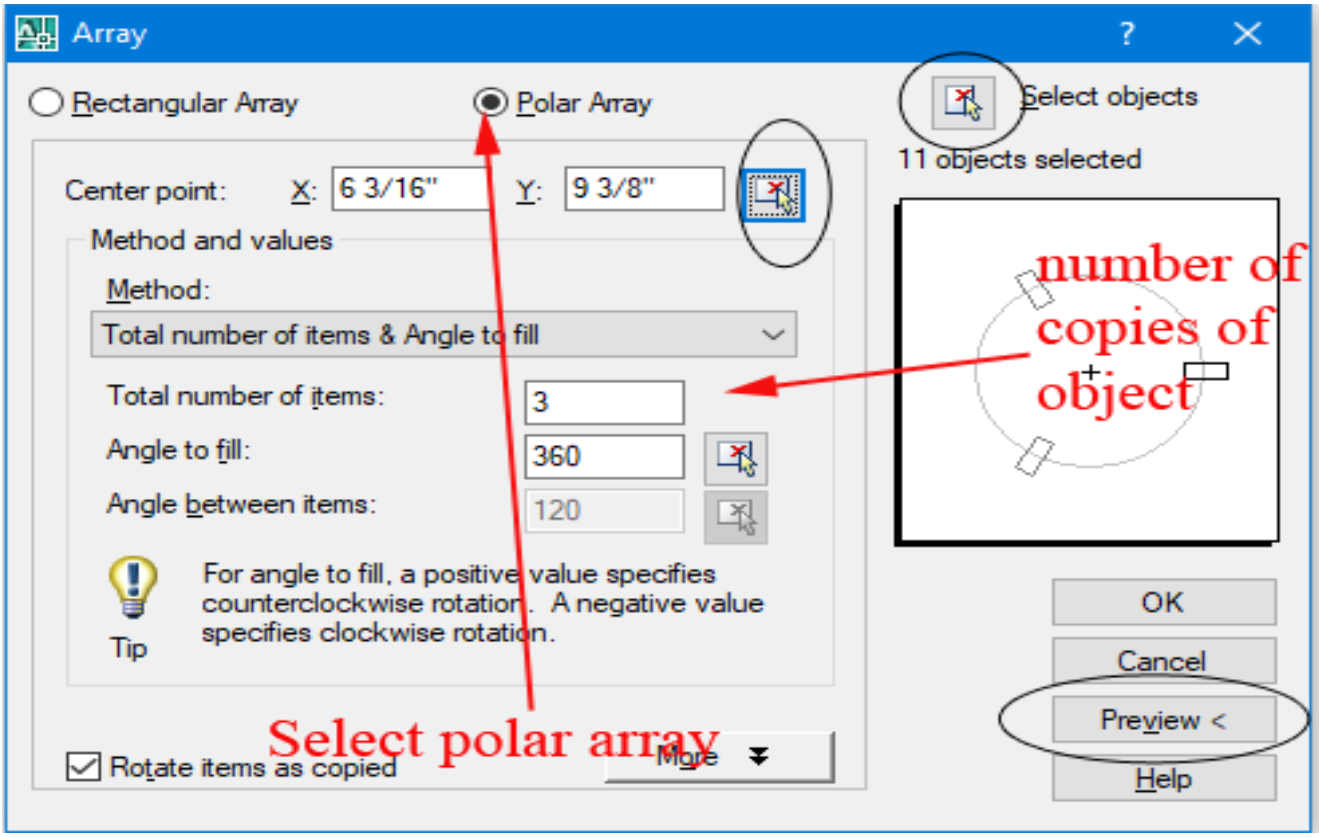


Fig: Array Dialogue box

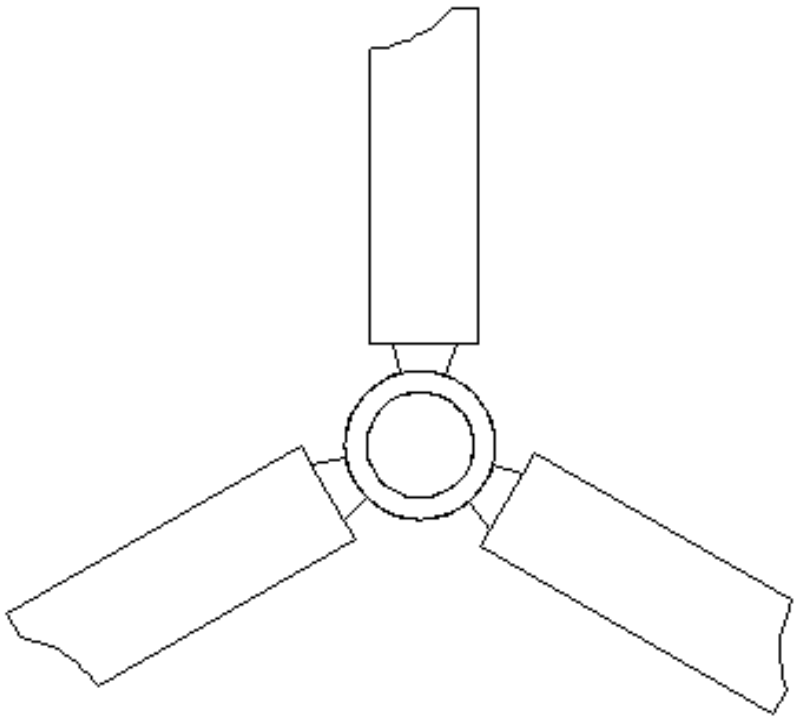


Fig: After Using Polar Array

Command name: Move

Command Shortcut: M ↩

Modify Toolbar Icon: 

Modify Menu Name: Move

Description: Move command is used to transfer an object one place to another. It is almost similar to copy command. You can click in any place to select a base point, then the second point is to be selected where you transfer the object. Specify the objects place, to move the object.

Use:

1. Click on the Move icon in modify toolbar or enter the command **M**
2. Select the object which you want to move.
3. Take the base point on the object or out of the object.
4. Drag the place where you want to move the object. Then by taking the object, click on that dragging place.

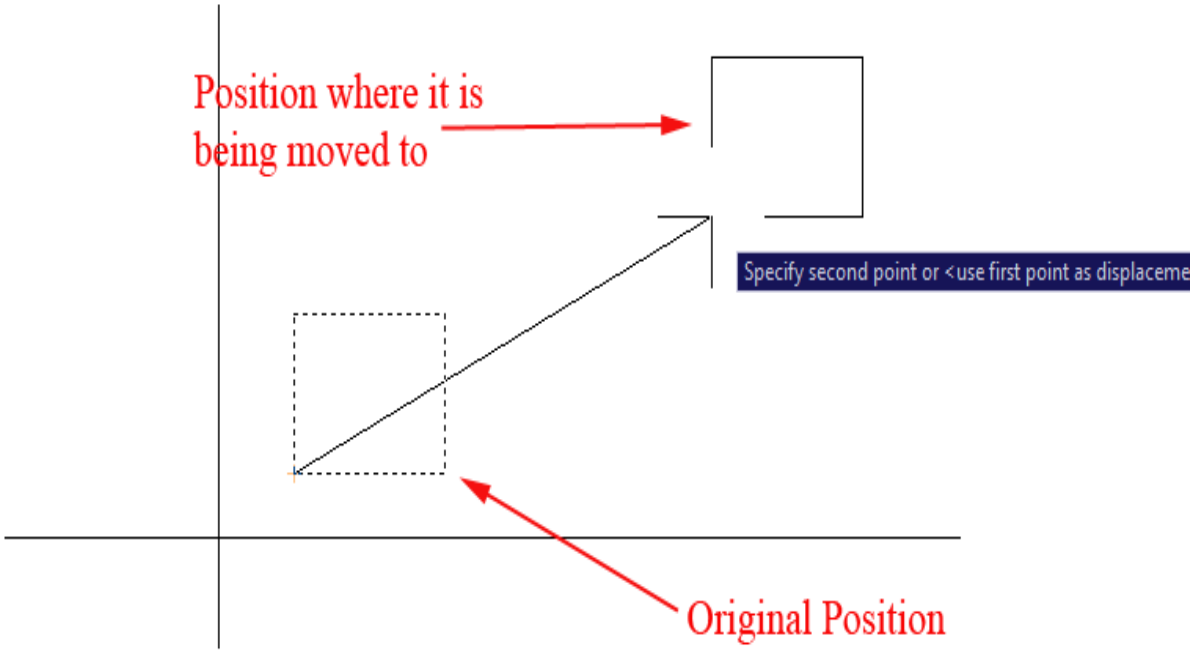


Fig: Move Command

Command Name: Rotate

Command Shortcut: Ro ↩

Modify Toolbar Icon: 

Modify Menu Name: Rotate

Description: The Rotate command is used to rotate an object with respect to a base point at a specific angle. Rotate command is used in architectural drawing, mechanical drawing etc.

Use:

1. Click on the rotate icon in modify toolbar or enter **ro.**
2. Select the object which you want to Rotate
3. Select the base point.
4. Wrote the rotation angle and press the enter button.

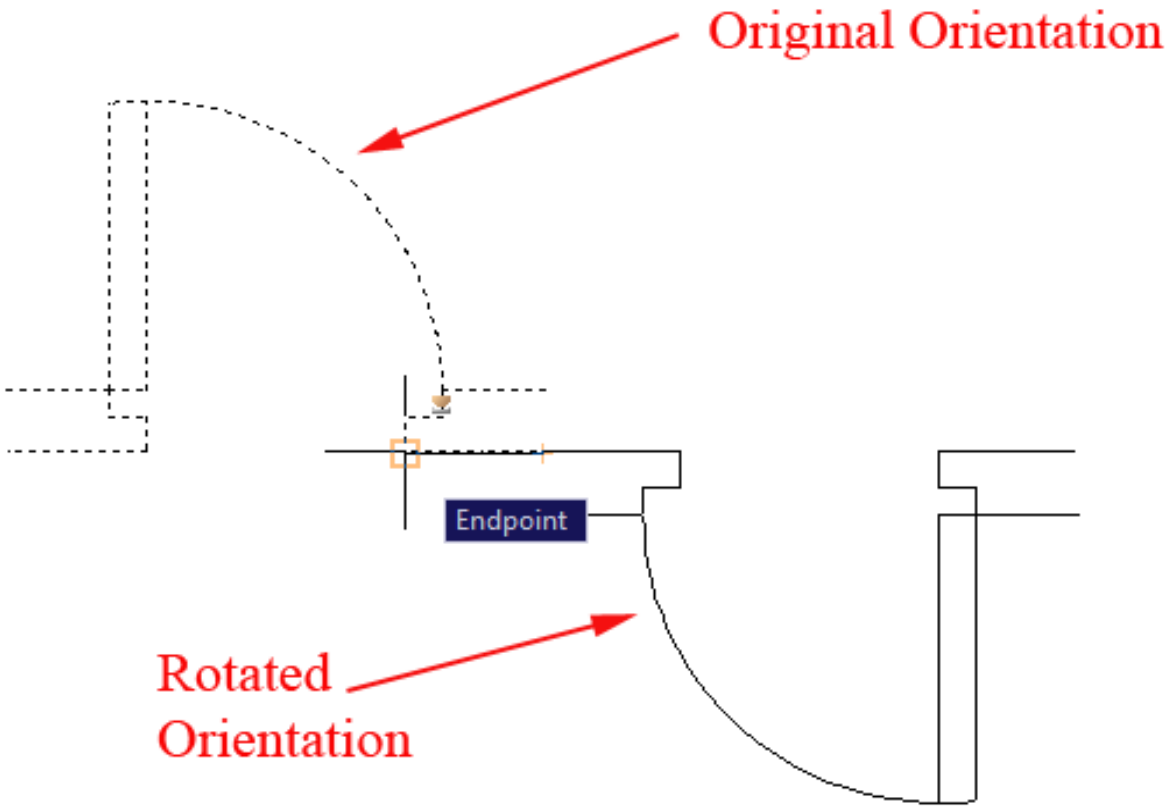


Fig: Rotate Command

Command Name: Scale

Command Shortcut: Sc ↩

Modify Toolbar Icon:

Modify Menu Name: Scale

Description: Scale command is used to make an object small or large. Scale command can be used in a specific ratio. For scaling, a base point should be specified after select an object. Then you should select the scale factor / reference length/ new length.

Use:

1. Click on the scale icon in modify toolbar.
2. Select the object.
3. Select the base point.
4. Specified the scale factor or entry the reference length or new length.

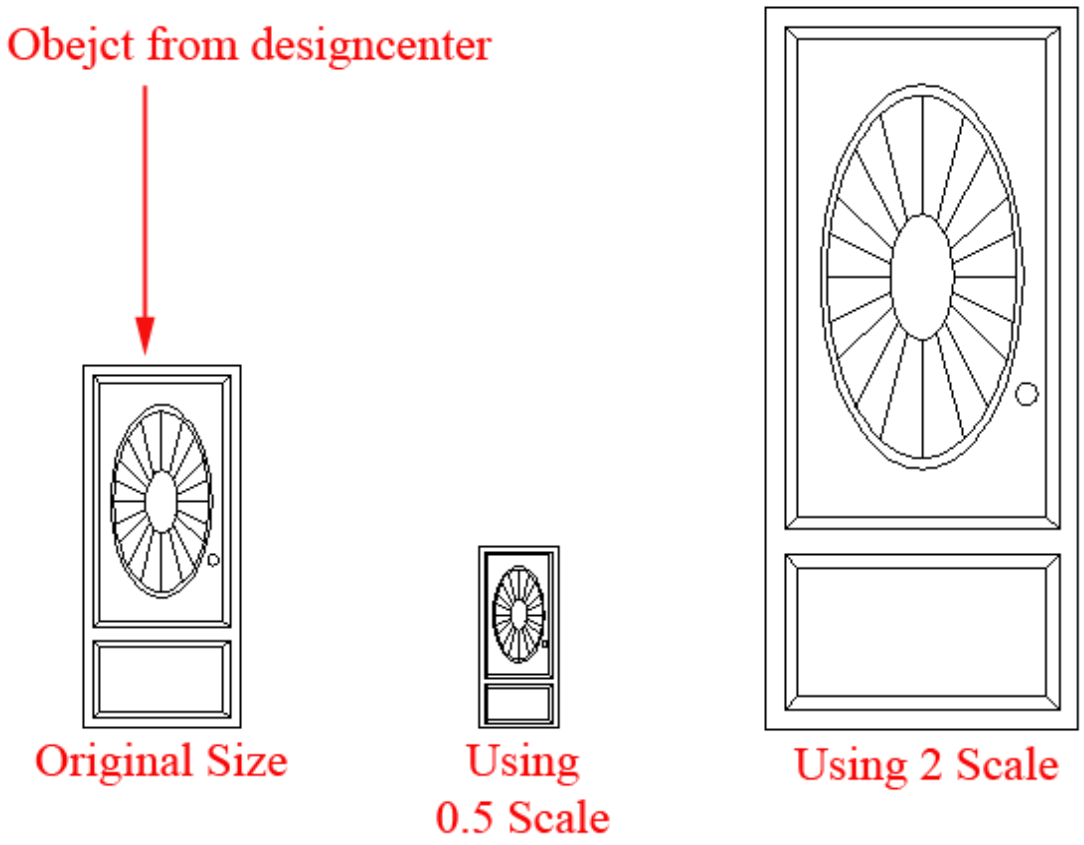


Fig: Scale Command

Command Name: Stretch

Command Shortcut: S↩

Modify Toolbar Icon: 

Modify Menu Name: Stretch

Description: The **Stretch command** in **AutoCAD** is a powerful tool. It can **stretch** or move objects, or **stretch** some and move others at the same time, depending on how someone selects them. The key to using **Stretch** is specifying a crossing selection box properly. The **Stretch command** starts, and **AutoCAD** prompts to select objects.

Use:

1. Select STRETCH Icon
2. Select object to stretch
3. Specify base point
4. Select second point

Example:

Stretch the table top 1feet.

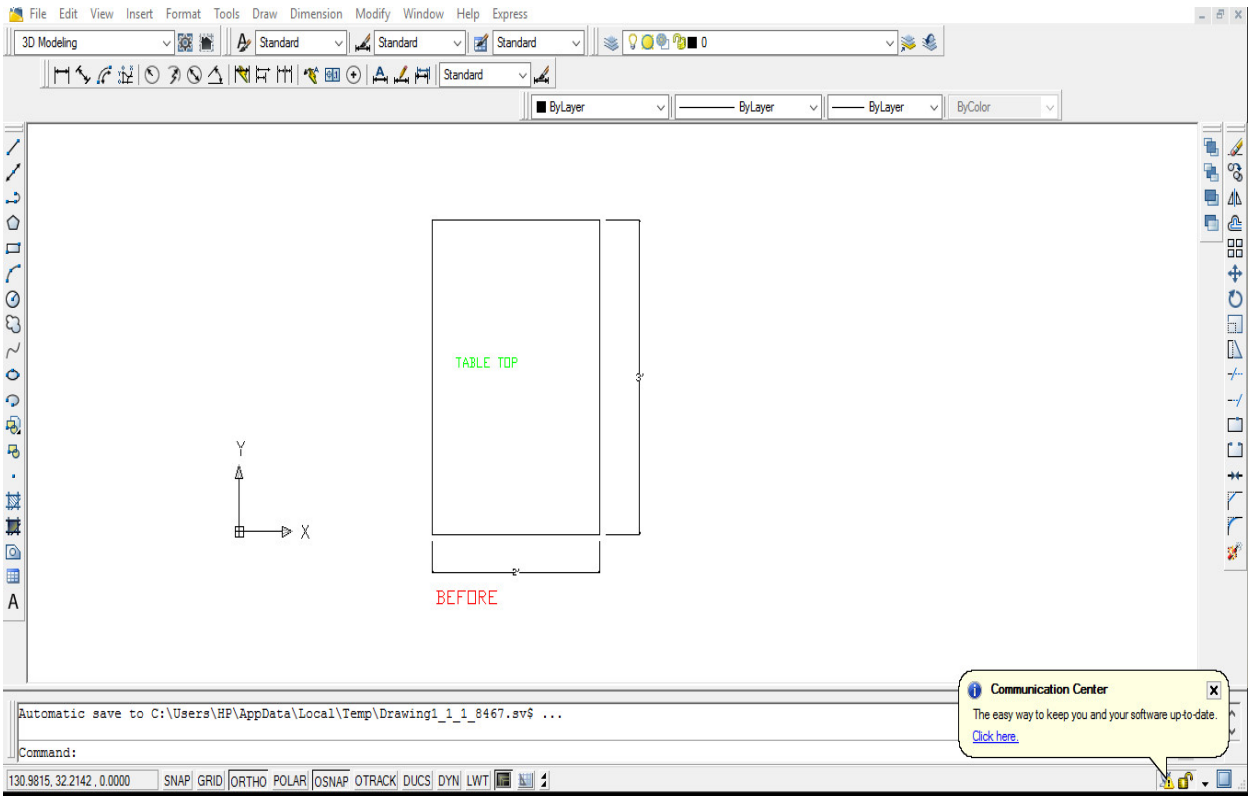


Fig: Reference Drawing

- **Command:** S↩
- **Select Objects:** Select the right side of the table top (CD) by the crossing window
- **Select Objects:** Click the right mouse button
- **Specify Base point:** Click the C point
- **Specify second point:** Drive the c to the right and write 12 then enter on ORTHO MOOD

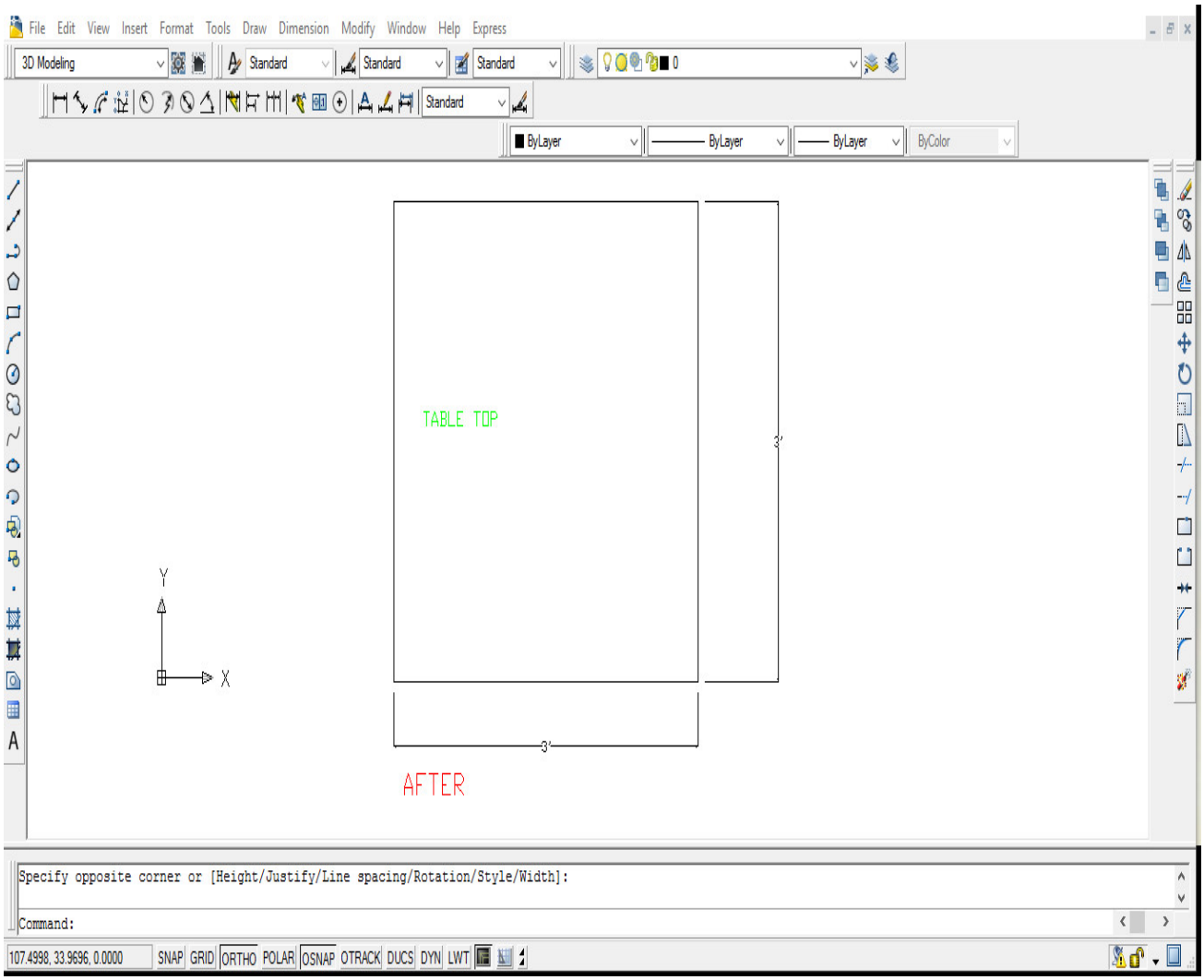


Fig: After using Stretch command

Command Name: Extend

Command Shortcut: EX↩

Modify Toolbar Icon: 

Modify Menu Name: Extend

Description: Extend command can be accessed from the toolbar or from the modify menu, text = Modify> Extend, or the command line prompt, type the command. Extend the command we can extend a line from its end (Endpoint).

Use:

- Select extend icon from the toolbar
- Select Boundary edge
- Select the object to extend

Example:

Connect the lines with the boundary edges.

Fig: Reference Drawing

- **Command:** EX↩
- **Select Object:** Select the boundary edge
- **Select objects:** click enter
- **Specify object to extend:** Select the lines to extend

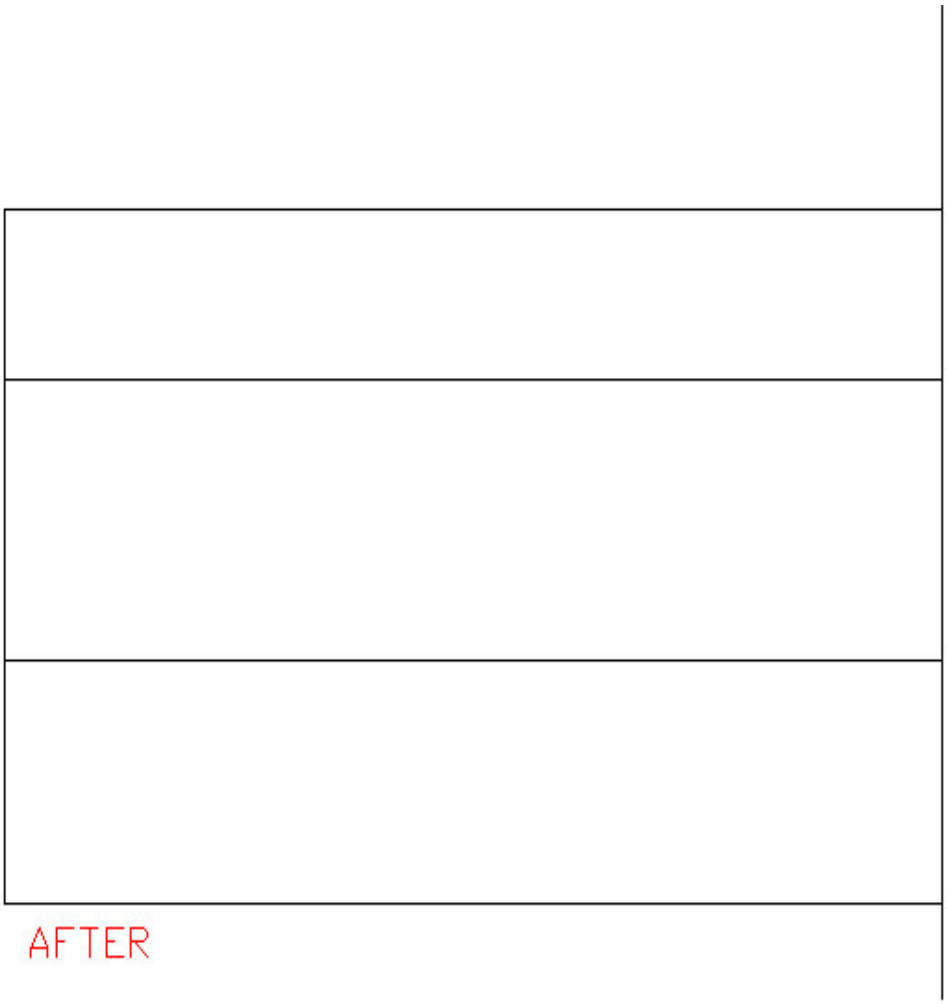


Fig: After using Extend command

Command Name: Trim

Command Shortcut: TR↩

Draw Toolbar Icon: 

Draw Menu Name: Trim

Description: Trim command can be accessed from the modify toolbar toolbar or from the modify menu, text = Modify> Trim, or the command line prompt, type the command. Trim command is used for cutting off parts of the drawing.

Use:

- Click on the **TRIM** icon
- Select objects that intersect each other and hit **ENTER**
- Select the part (or parts) to be removed

Example:

Trim the objects of the pic.

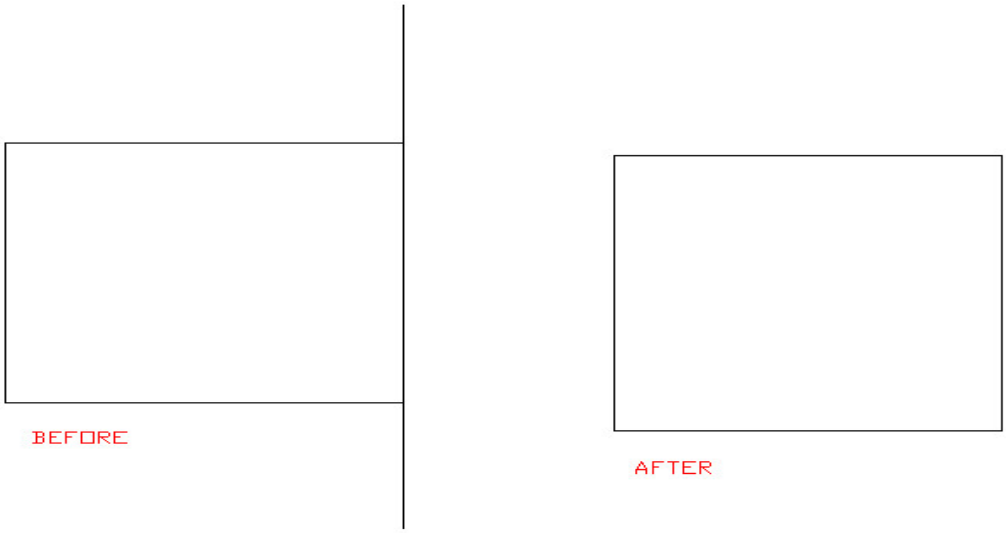


Fig: Trim command demonstration

Command: TR↩

Select objects: select the cutting edges

Select object to trim: Select the lines to trim.

Command Name: Break

Command Shortcut: BR ↩

Modify Toolbar Icon: 

Draw Menu Name: Break

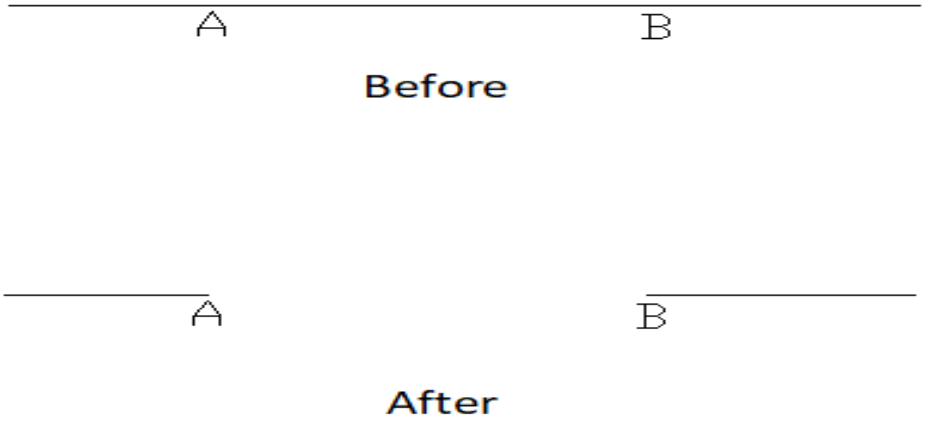
Description: The break command can create a gap between two specified points of an object, breaking it into two objects. If the points are off of an object, they are automatically projected on to the objected. Break is often used to create a space for a block or a text.

Use:

- Select break icon or type **BR** in command line and press enter.
- Select the first point from where you want to create a break when prompted to select object. Then select the second point of the break.
- If you have first selected the object you want to break and want to enter the first point then type F and enter first point of the break and continue.

Example:

Breaking the line from a specific point:



Command: BR ↩

Select object: Click at A point

Specify second break point or [first point]: Click at B point.

There is a gap created between A and B point.

Command Name: Join

Command Shortcut: J↩

Modify Toolbar Icon: 

Modify Menu Name: Join

Description: As the name suggests it is used for joining lines. It is mostly used in architectural, mechanical and furniture drawing. This command does not work on blocks so they need to be exploded first before applying the join command.

Use:

1. Enter the command **J** or click the join icon
2. Select the lines that you want to join then press enter the lines will be joined

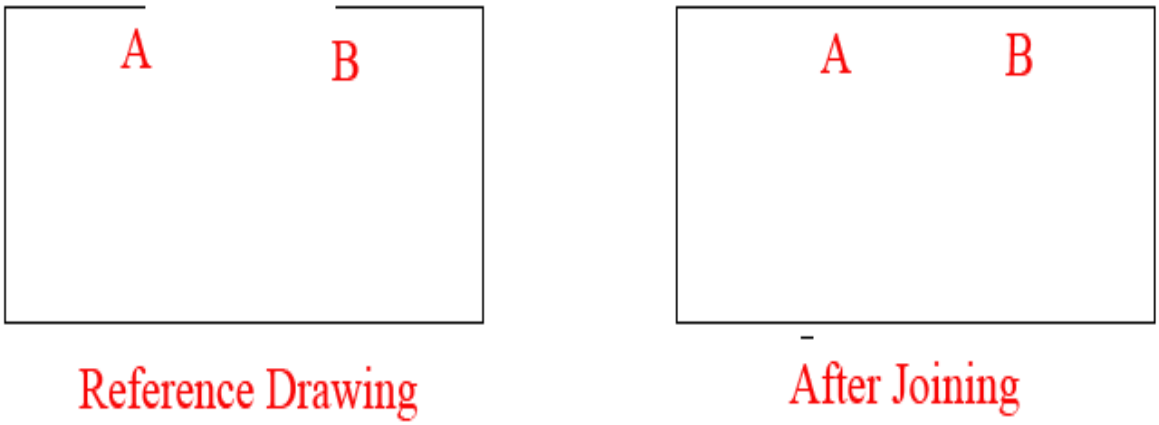


Fig: Join command

Command Name: Divide

Command Shortcut: Divide or DIV ↩

Modify Toolbar Icon: 

Modify Menu: Point → Divide

Description: Divide command is used to divide any object in definite portion. We can use this command for point marking and block marking on the object. By using this command, we can divide a linear line more than one line in same portion. In divide command, the marking is a reference point.

Use: To divide an object in same portion and for point marking, we can apply the following steps:

1. In command line write DIV and then press enter.
2. Select the line, circle, ellipse, polyline, arc or spline.
3. Write how much we want to divide object and press enter.
4. To divide in same portion and for block marking we can apply the following steps:
    i. Make the block, if we want.
    ii. In command line write **DIV** and then press enter.
    iii. Select the line, arc, circle, ellipse, polyline or spline.
    iv. Write **B** and then press enter.
    v. Write the name of the definite block and then press enter.
    vi. For block alignment with divide object, write **y** and then press enter or write **N** and then press enter.
    vii. Write the number of divided parts we want and then enter press.

Example: To divide the linear line in 8 portion, we can apply the following steps.

- **Command: DIV** ↩
- **Select object to divide:** select the line.
- **Enter the number of segment or (Block):** write 8 and then press enter.

The definite line is divided into 8 portion.

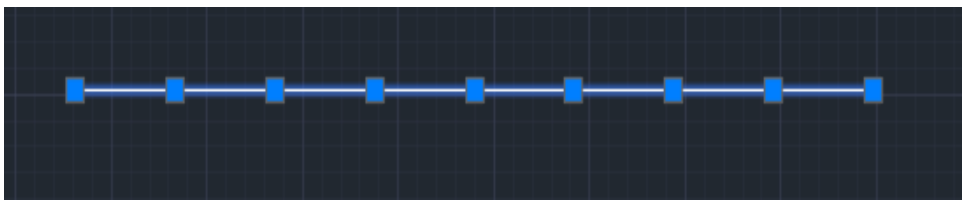


Fig: Divide Command

Command Name: Break

Command shortcut: br↩

Modify Toolbar Icon: 

Modify Menu: Modify → Break

Description: Break at point is used to break the selected object at a single point.

Use:

1. In the modify menu select the option of break at point.
2. **Command:** select the single point.
3. **Command:** specify first break point: select the break point of the object. Then see the definite portion of the object is removed.

Example: We can use break at point command to break a line at a single point.

- **Command:** Press break at point on modify tool.
- **Select object to divide**: Click the line.
- **Command**: specify first break point or [first point]: click at any point of the line.

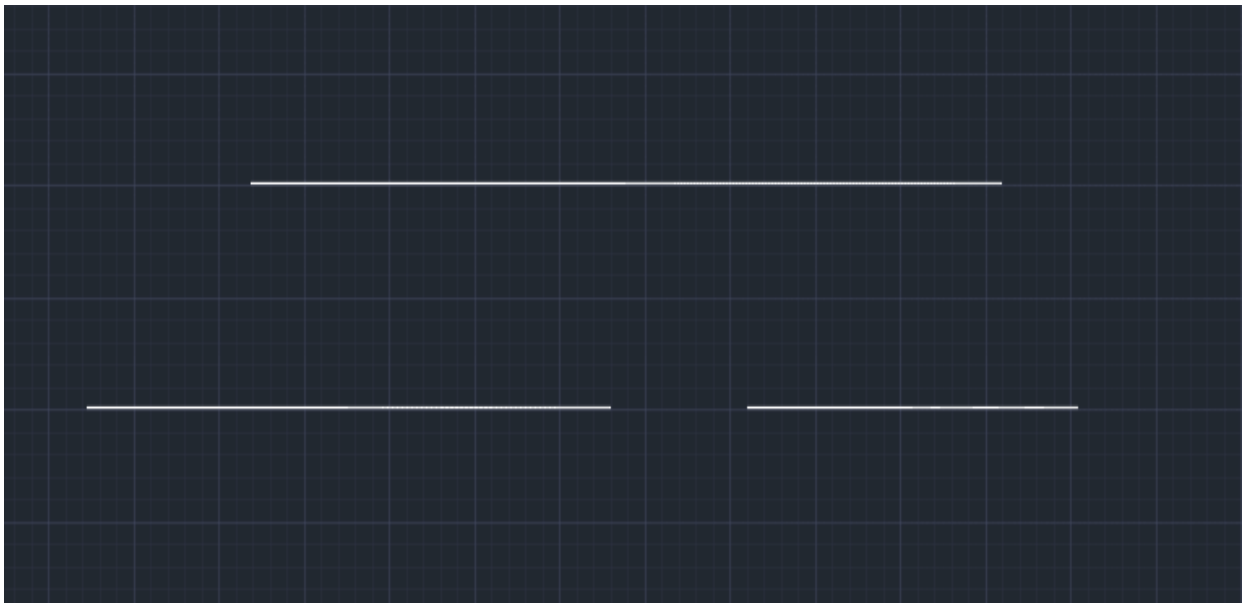


Fig: Break Command

Command name: Chamfer

Command shortcut: CHAMFER ↩

Modify toolbar icon: 

Modify menu name: Chamfer

Description: A chamfer is an angled line that meets the endpoint of two straight 2D objects. By using the command chamfer a bevel or chamfer can be created on an object.

Use:

1. Select the chamfer icon or type **D** and press enter.
2. Specify first chamfer distance and press enter.
3. Specify second chamfer distance and press enter.
4. Select the first line.
5. Select the second line.

Example:

Adding chamfered corner on a rectangle:

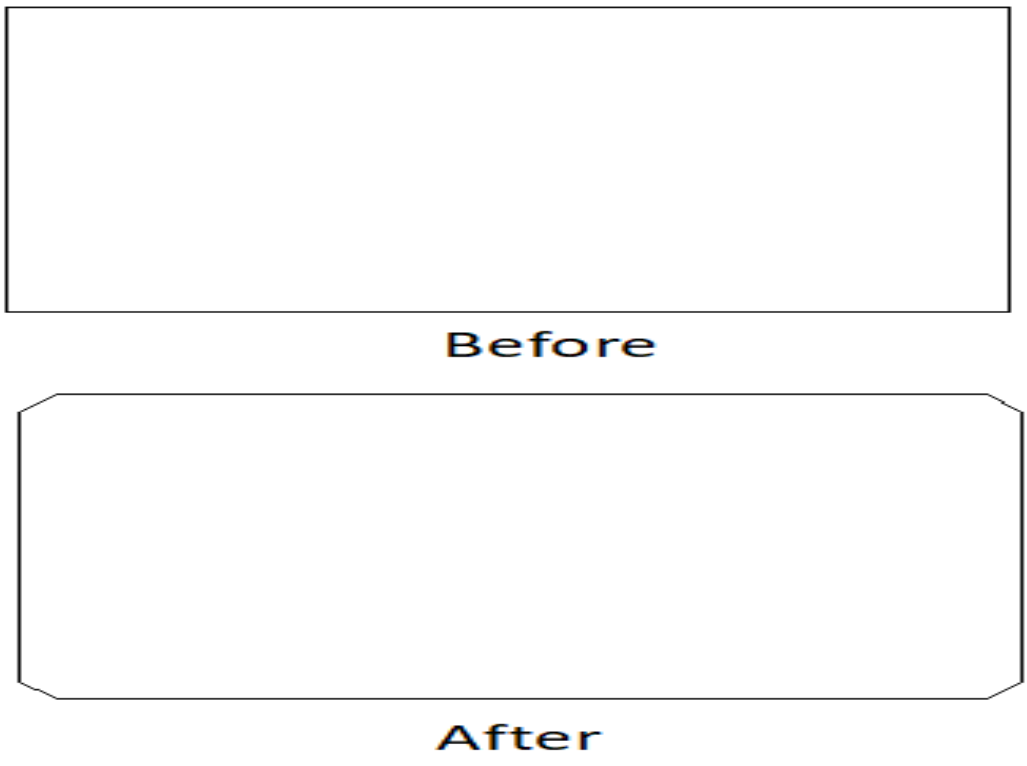


Fig: Chamfer Command

- Type **REC** and press enter.
- Draw a rectangular.
- Type **CHAMFER** and press enter.
- Type **D** and press enter.
- **Command:** Specify first chamfer distance<0.5000>:
  Type 1 and press enter.
- **Command:** Specify second chamfer distance<1.000>:
  Type 1 and press enter.
- **Command:** Press enter
- **Command:** Select first line: Type P and enter.
- **Command:** Select 2D polyline: Select the rectangular.

Command name: Fillet

Command shortcut: FILLET ↩

Draw toolbar icon: 

Draw menu name: Fillet

Description: The fillet helps convert sharp edges to round edges. Even though this command can be bypassed using circles and trim, its use will still be appreciated seeing how easy getting to the result is while using it.

Use:

- Select the fillet command icon.
- Type R and press enter.
- Specify the fillet radius and press enter.
- Select the first line.
- Select the second line.

Example:

Fillet the corners of a square:

- Type **REC** and press enter.
- **Command:** Specify the first corner point or specify a corner point of the square by clicking at a place.
- **Command:** Specify other corner point: Type @4, 4 and press enter. The square will be drawn.
- Select **fillet** icon from the modify toolbar.
- **Command:** Select the first object or: Type **R** and press enter.
- **Command:** Specify fillet radius<0.5000>: Type 1 and press enter.
- **Command:** Select the first object or: Type **R** and press enter.
- **Command:** Select 2D polyline: Select the square.

Command name: Explode

Command shortcut: X ↩

Draw toolbar icon: 

Draw menu name: Explode

Description: The explode command breaks a compound into its component objects. This command explodes a compound object when you want to modify its components separately. This command is used on objects that can be exploded include blocks, polylines and regions among others.

Use:

1. Select explode option from the modify toolbar.
2. Select objects that you want to explode.
3. Press enter to end the command.

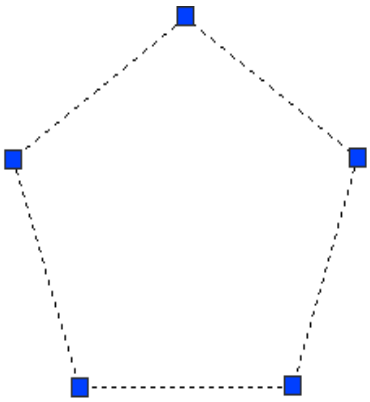


Fig:1-Selection before explode

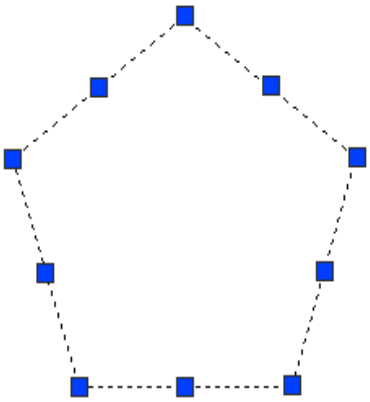


Fig:2-Selection after explode

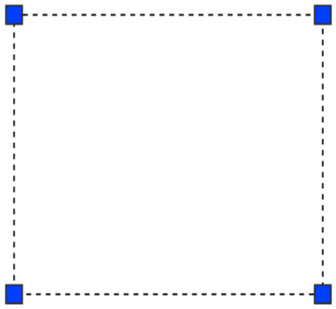


Fig:3-Selection before explode

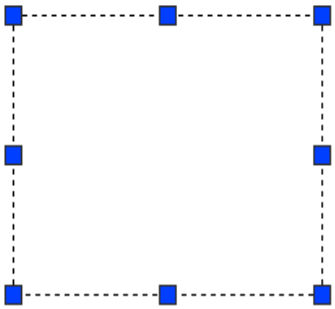


Fig:4-Selection after explode

Fig: Using Explode on polygons and rectangle

Command Name: Match Property

Command Shortcut: MATCHPROP or MA or Painter ↩

Standard Toolbar Icon: 

Modify Menu: Match Properties

Description: We can use properties to copy an object to other object. For example, we can change an object's color, rear, line type, line, weight, light type scan, plot style,3D thickness etc. to other object. For matching of the properties, we can use MATCHPROP command. The icon is located on top portion of the screen just below the draw menu.

Use: For copying one objects properties to other, we can apply the following steps:

1. In command line, write **MA** and then press enter.
2. Select the object which we want to copy the object's properties.
3. If we want to copy a particular properties, then for settings we will press S and then enter.
4. Put tick mark that we don't want to copy in the property settings dialogue box.
5. Select the object of the object properties which we want to copy.

Example: To copy an object properties for other object, we can apply the following steps:

- **Command:** MA
- **Select source object:** select the source object
- **Select destination object (S) (settings):** select destination object
- **Select destination object (S) or (settings):** select the text which is given below
- **Select destination object (s) or (settings):** press enter.

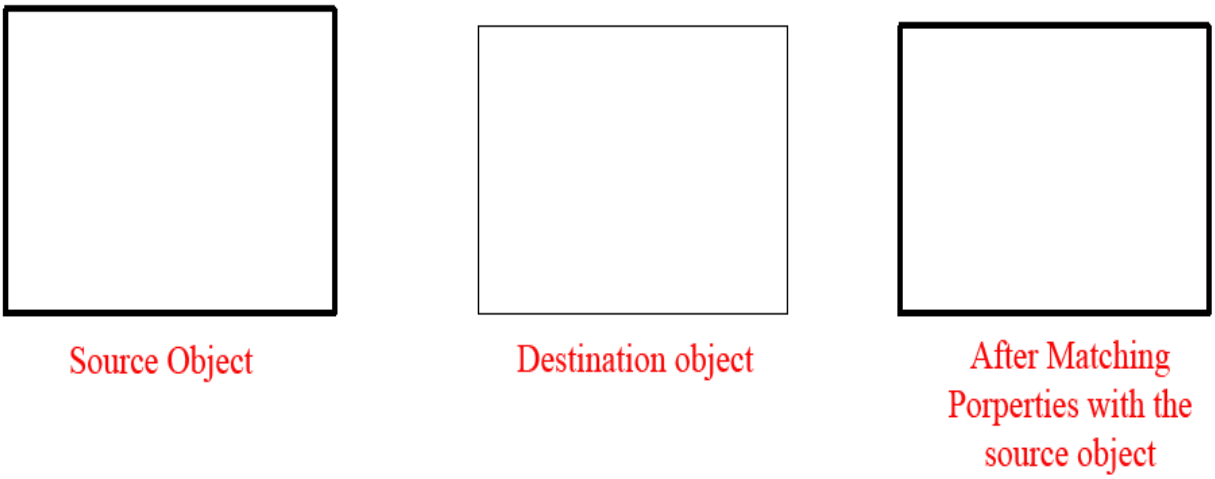


Fig: Match properties command

Command Name: Properties

Command Line: PROPERTIES or PR or CH ↩

Standard Toolbar Icon: 

Modify Menu: Properties

Short Menu: Click the right side of the mouse and click the properties

Pointing Device: Double click on the object

Description: Properties command is used to display the properties palette. In the palette, we can see the properties of any selected object. In command line, we can change color, layer, line type, line type scale, line weight and thickness, this 6 types of properties of any object.

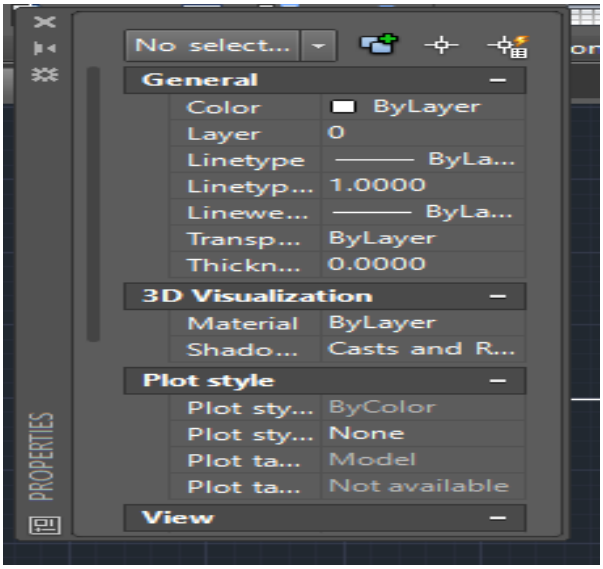
Use: For displaying properties palette and change the properties, we can apply the following steps:

1. In command line, write PR and then press enter or click the properties icon in standard toolbar.
2. Select the object which we want to change the properties
3. Select the value we want to change and entry the new one.
4. To remove the selection press Esc.

Example: To find out the properties of any object we have to follow the following steps:

Command: PR Or click right button on the object and click on properties.

In this way, we can see the properties of an object and change them if we want.



DIMESNION TOOLBAR

Dimension is used for showing the values of height and weight in a drawing. A drawing cannot be complete without specifying its dimension. Using dimension one can learn about the length, width & height of a room, thickness of a wall, width of the window and door, radius of the pipe etc. Using AutoCAD, one can easily draw dimensions flawlessly.

Dimension toolbar can be accessed by going to the dimension menu or by right clicking the upper right portion of AutoCAD to access ACAD and then ticking dimension.

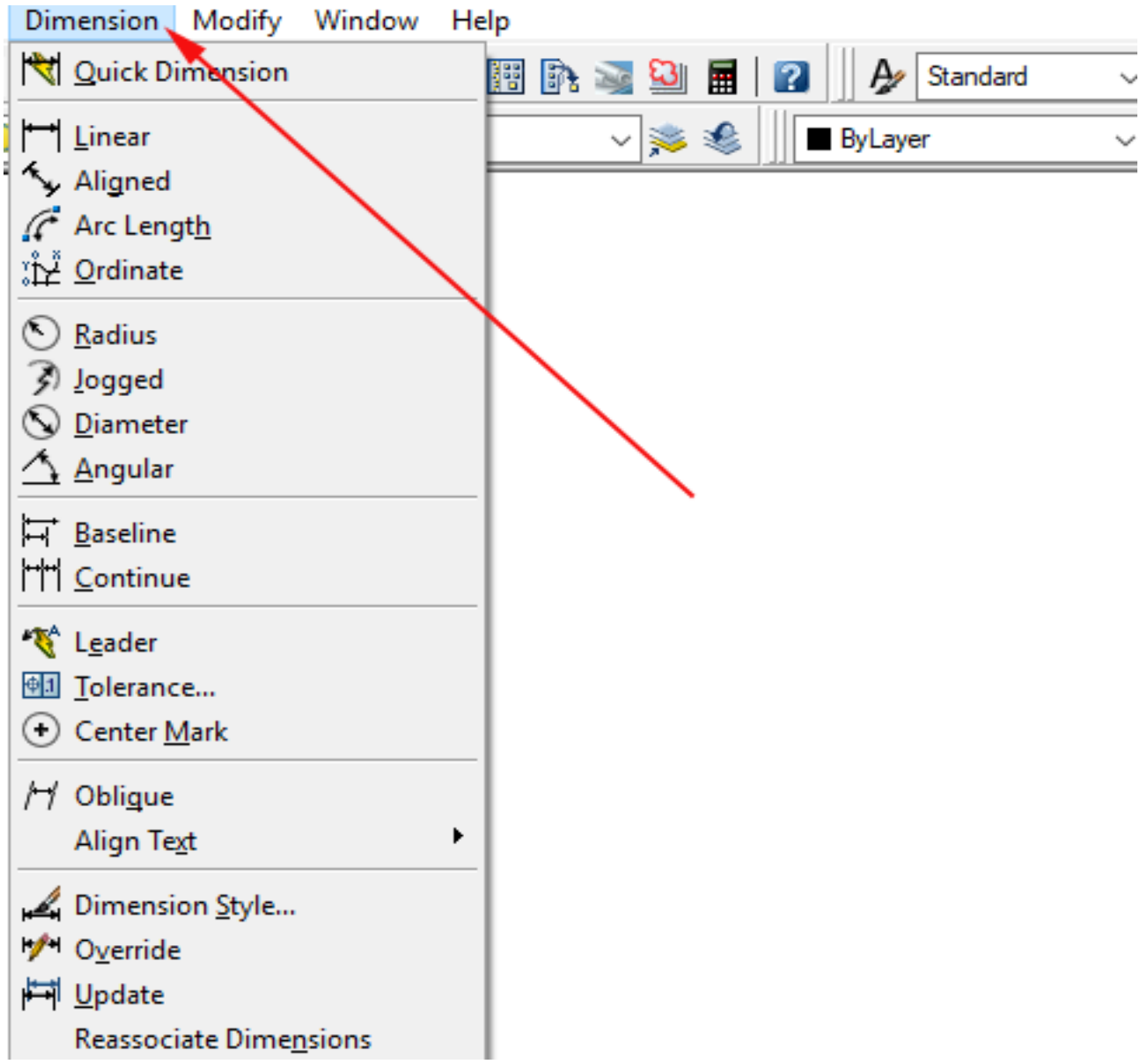


Fig: The dimension window and list of available dimension types

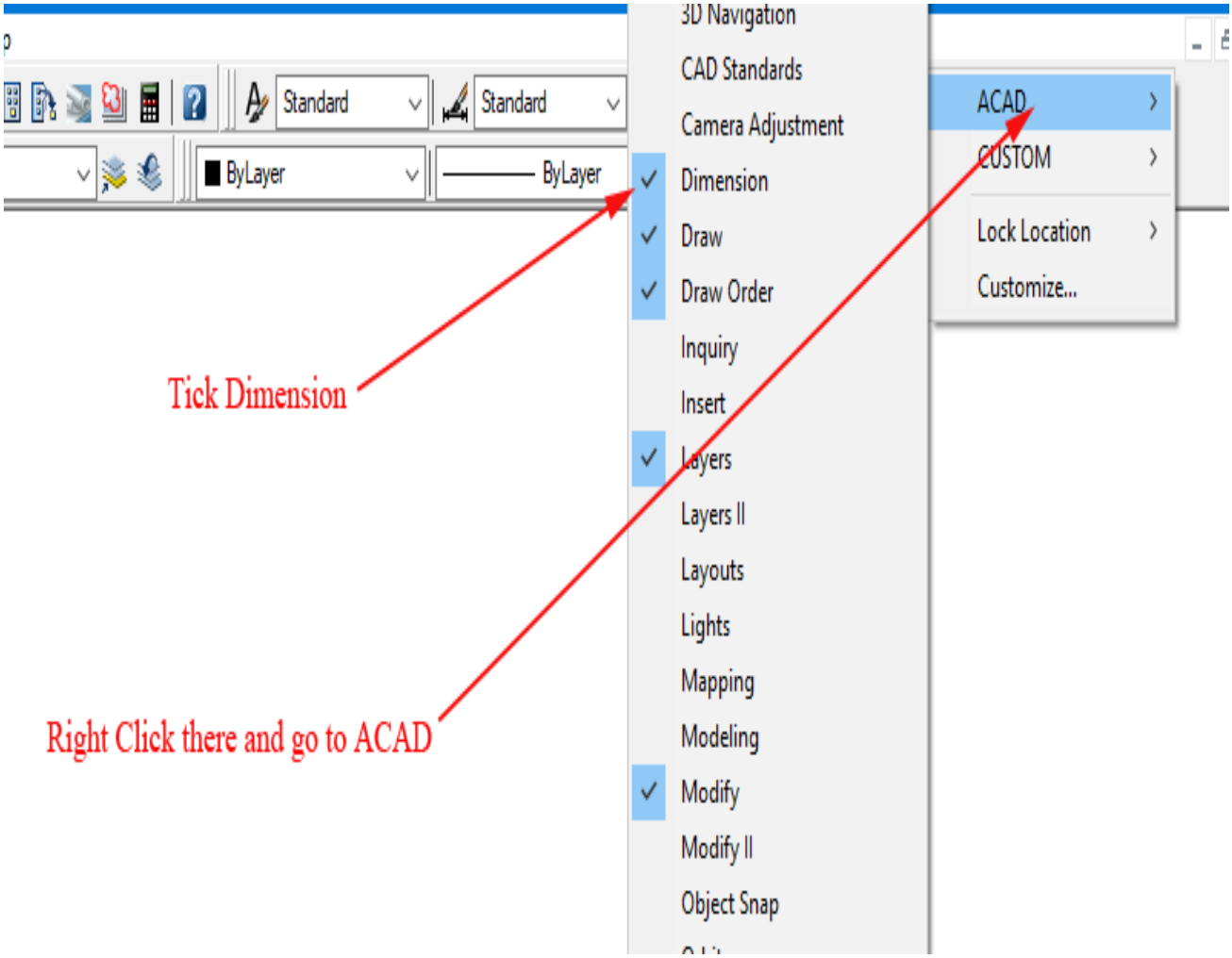


Fig: Enabling the dimension toolbar

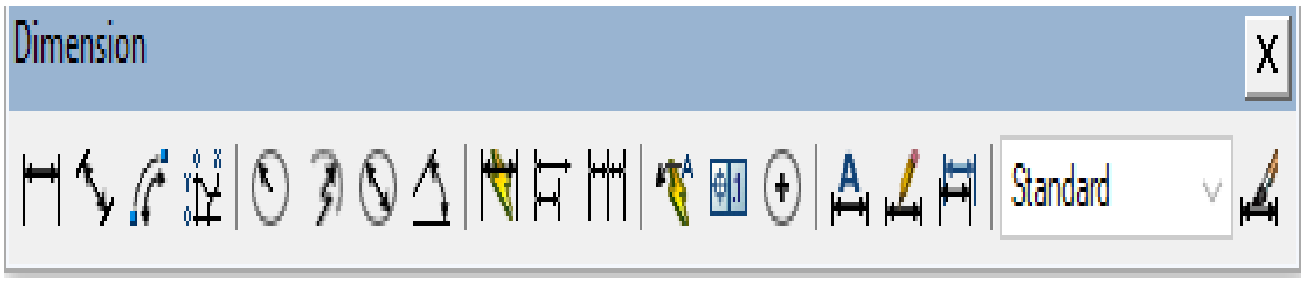


Fig: Dimension toolbar

Command Name: Linear Dimension

Command Shortcut: dli

Dimension Toolbar Icon: 

Dimension Menu Name: Linear

Description: Linear dimension is used for measuring the linear distance between two points. Linear dimension can be used to measure horizontal, vertical and rotated dimension.

Use:

1. Enter the command **dli** or select form the toolbar.
2. Specify the first point, then specify the second point. Using the mouse cursor to move the dimension at a distance away from the drawing. If the unit does not match your desired unit the process is shown at the topic “Dimension Style”.

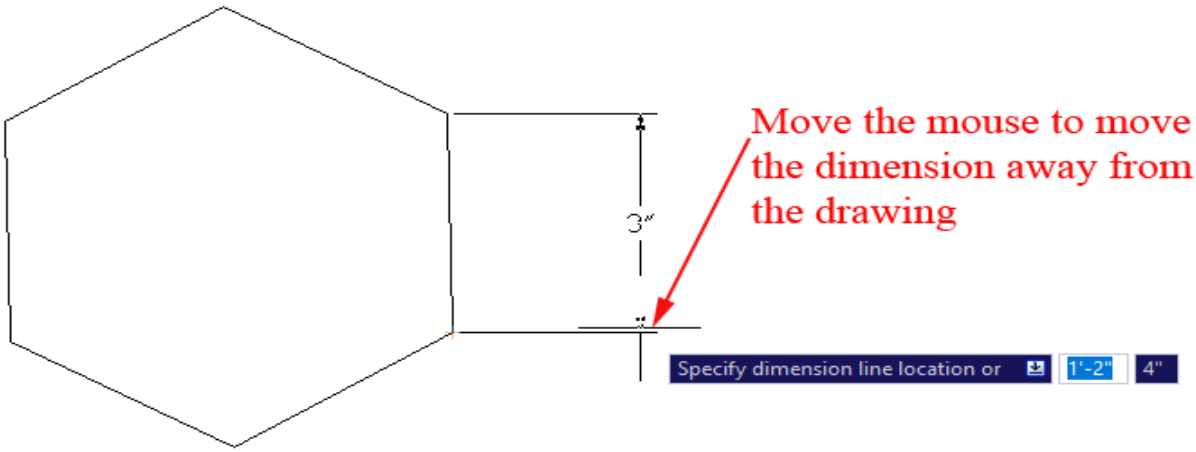


Fig: Moving the dimension line

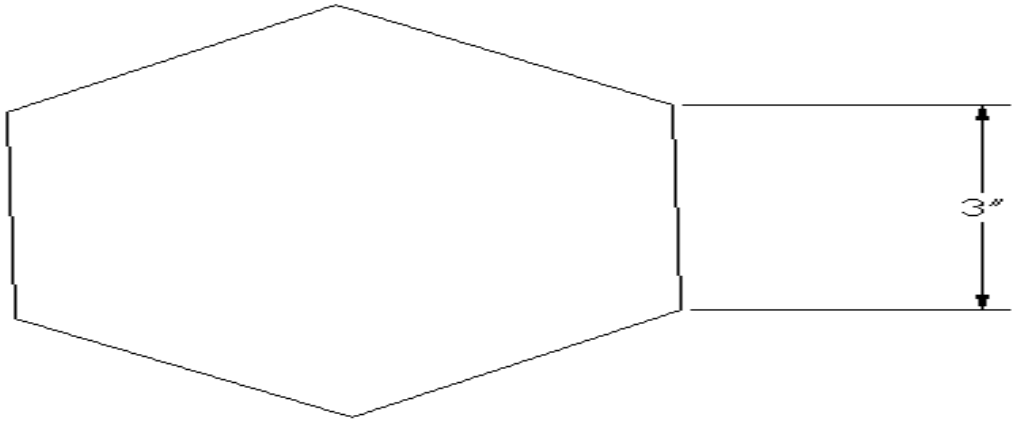


Fig: Linear dimension

Command Name: Aligned Dimension

Command Shortcut: dal

Dimension Toolbar Icon: 

Dimension Menu Name: Aligned

Description: Aligned dimension is also called true length dimension. Aligned dimension is parallel to the line going between two points at any angle. As a result it measures true length between two points at an angle.

Use:

1. Enter the command **dal** or select form the toolbar.
2. Specify the first point, then specify the second point. Using the mouse cursor to move the dimension at a distance away from the drawing. If the unit does not match your desired unit the process is shown at the topic “Dimension Style”.
3. (optional) At step 2, after choosing the two point type **M** for multi-line text**, T** for single-line text, **A** for text angle if you want to change these.

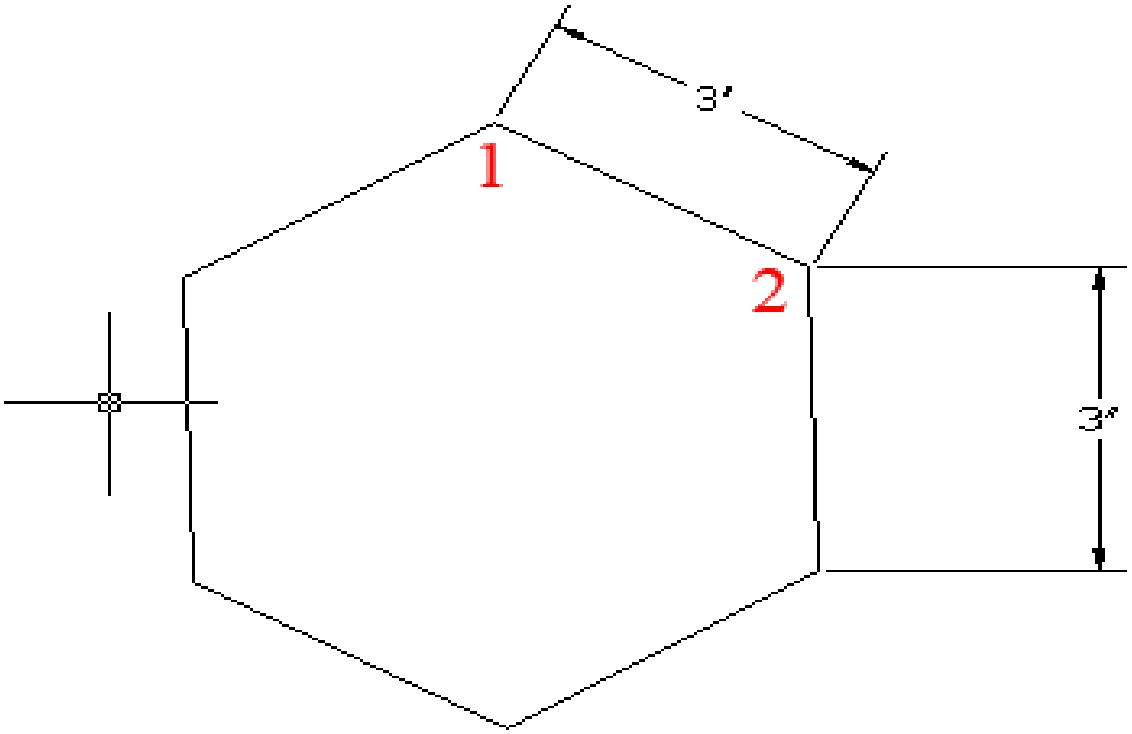


Fig: Aligned Dimension

Command Name: Angular Dimension

Command Shortcut: dan

Dimension Toolbar Icon: 

Dimension Menu Name: Angular

Description: Angular dimension is used for measuring the angle created by an arc, circle, lines etc.

Use:

1. Enter the command **dan** or select form the toolbar.
2. Specify the first line or point then specify the second line. Use the mouse to get the angle from the direction that you want. If the unit does not match your desired unit the process is shown at the topic “Dimension Style”.
3. (optional) At step 2, after choosing the two point type **M** for multi-line text, **T** for single-line text, **A** for text angle if you want to change these.

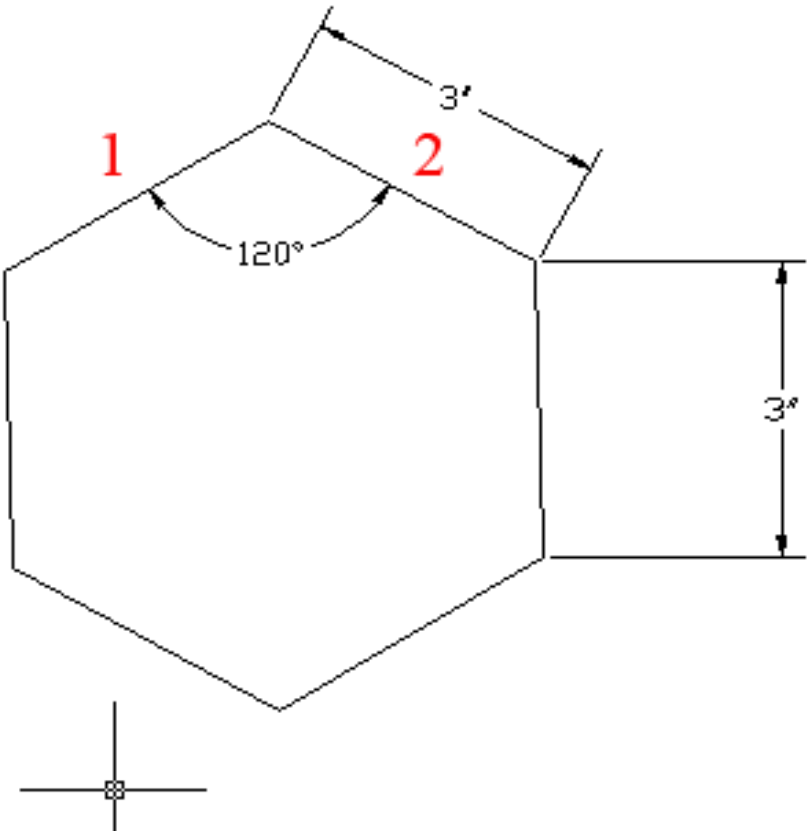


Fig: Angular Dimension

Command Name: Radius Dimension

Command Shortcut: dra

Dimension Toolbar Icon: 

Dimension Menu Name: Radius

Description: Radius dimension is used for showing the radius of a circle or an arc

Use:

1. Enter the command **dra** or select form the toolbar.
2. Specify the circle or the arc of which you want to show the radius. Next the point where you want to show the dimension. If the unit does not match your desired unit the process is shown at the topic “Dimension Style”.
3. (optional) At step 2, after choosing the two point type **M** for multi-line text, **T** for single-line text, **A** for text angle if you want to change these.

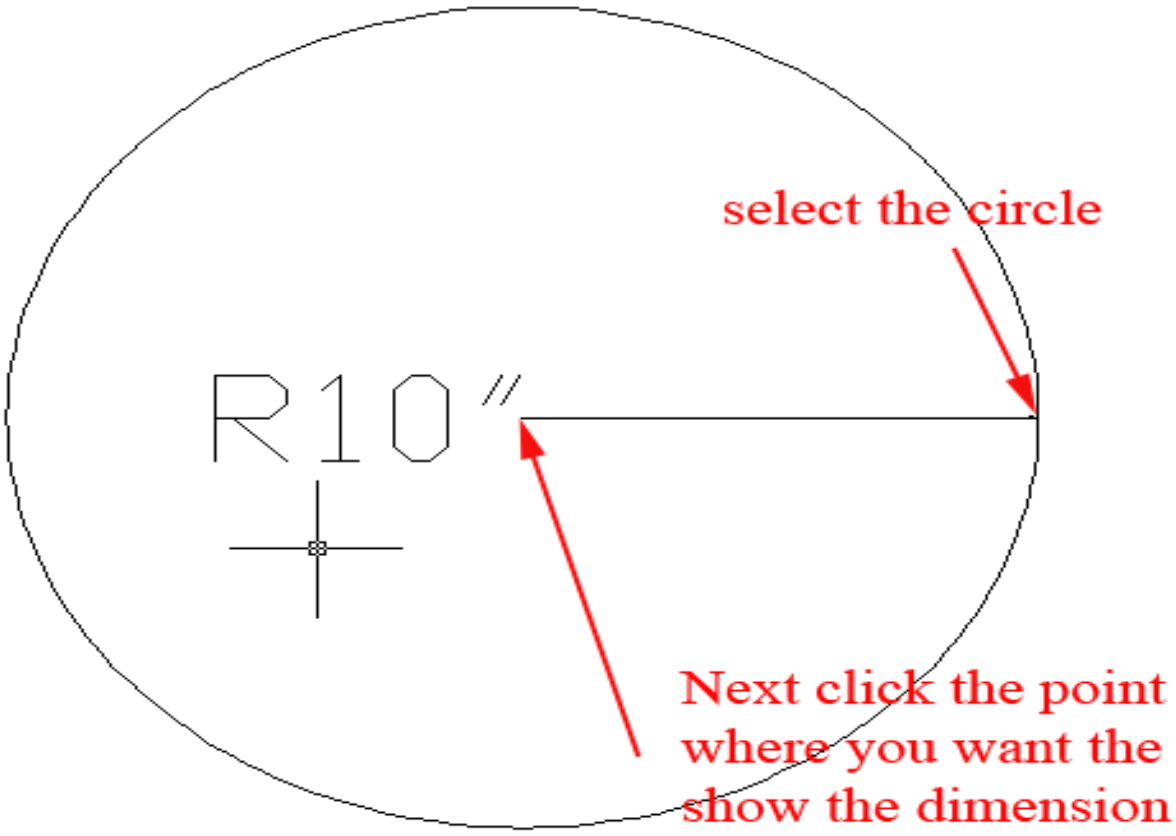


Fig: Radius Dimension

Command Name: Diameter Dimension

Command Shortcut: ddi

Dimension Toolbar Icon: 

Dimension Menu Name: Diameter

Description: Diameter dimension is used for showing the diameter of a circle or an arc

Use:

1. Enter the command **ddi** or select form the toolbar.
2. Specify the circle or the arc of which you want to show the diameter. Next the point where you want to show the dimension. If the unit does not match your desired unit the process is shown at the topic “Dimension Style”.
3. (optional) At step 2, after choosing the two point type **M** for multi-line text, **T** for single-line text, **A** for text angle if you want to change these

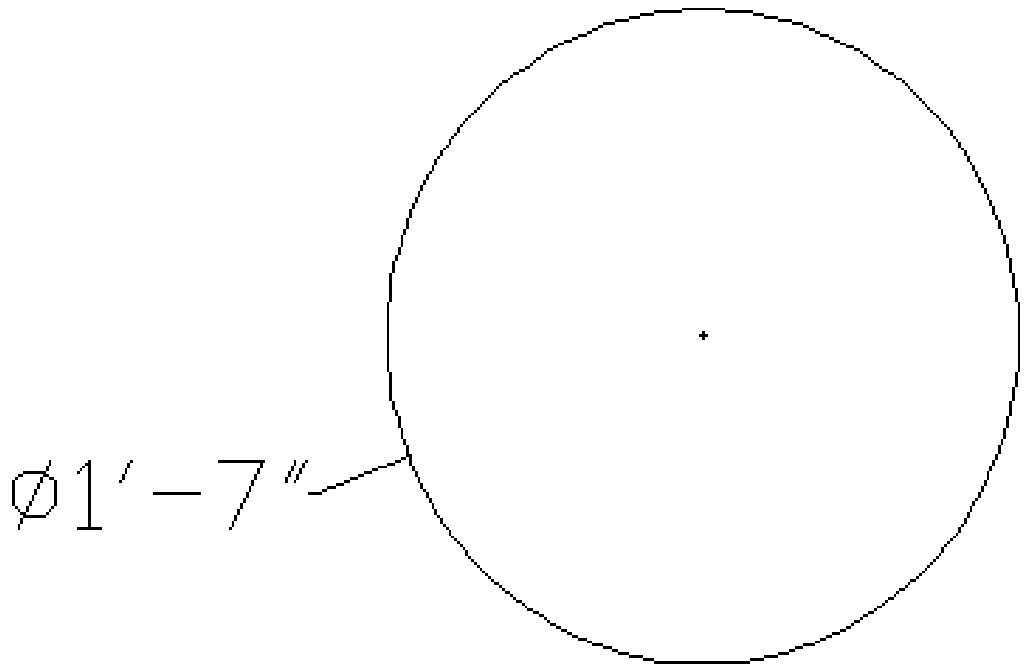


Fig: Diameter Dimension

Command Name: Continue Dimension

Command Shortcut: dco

Dimension Toolbar Icon: 

Dimension Menu Name: Continue

Description: Diameter dimension is used for showing the diameter of a circle or an arc

Use:

1. First measure the linear dimension of the first line.
2. Next Enter the command **dco** or select form the toolbar. The dimension will continue from the last point where the linear dimension ended.
3. Specify up to which point you want to measure the length. Continue repeating this 3rd step. Until the dimension is fully measured. You only need to click on the end points for continue dimension. Press Esc to exit.
4. If you want to continue any previous dimension enter **dco** then choose the dimension and follow step 3

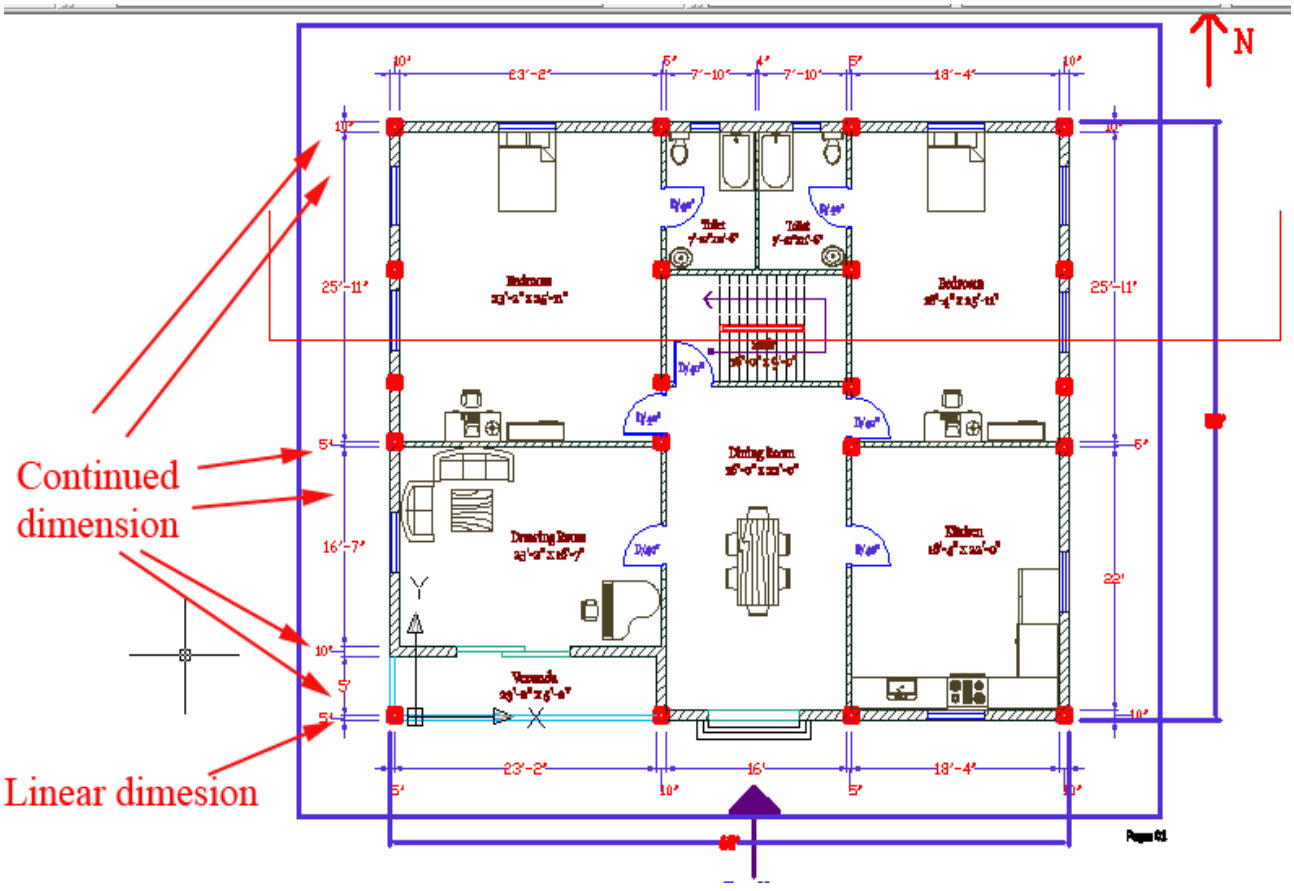


Fig: Continue

Command Name: Dimension style

Command Shortcut: d

Dimension Toolbar Icon: 

Dimension Menu Name: Dimension Style…

Description: Dimension style is used for changing various aspects of the dimension line an text such as Line color, line type, units, arrow size, text height, text spacing etc.

Use:

1. Enter the command **d** in the command bar or tap the icon. The dimension style manager will open.
2. Click **New** to create your own dimension style or **modify** to make changes on the default dimension style. (If you are using modify skip to step 4)
3. Clicking **New** will open the **Create New Dimension Style** window. Enter the name and press continue
4. A new window will open. From here you can change different attributes of dimension. It has the following tabs-

    - **Lines:** From here you can change the color of the lines, line type, line weight, use baseline spacing for creating distance between drawing line and the dimension line, suppress lines etc.
    - **Symbols and Arrows:** From here you can make changes to the shape of the arrow, size of the arrow etc.
    - **Text:** In the text tab you can change text style, text color and text height to make the text more visible from a distance
    - **Fit:** In this tab you can assign AutoCAD to give priority to either text or arrow when there is a deficit in space.
    - **Primary Units:** It is set to decimal by default but as we use architectural units it is best to change it to architectural.
    - **Secondary Units:** It is turned off by default. Can be enable if the user wants to display a secondary unit inside of a third bracket
    - **Tolerances:** In this tab you can enable the option to add tolerances which makes the amount of error that is negligible in the plan visible. You can also change how it is to be shown here. It is turned off by default.

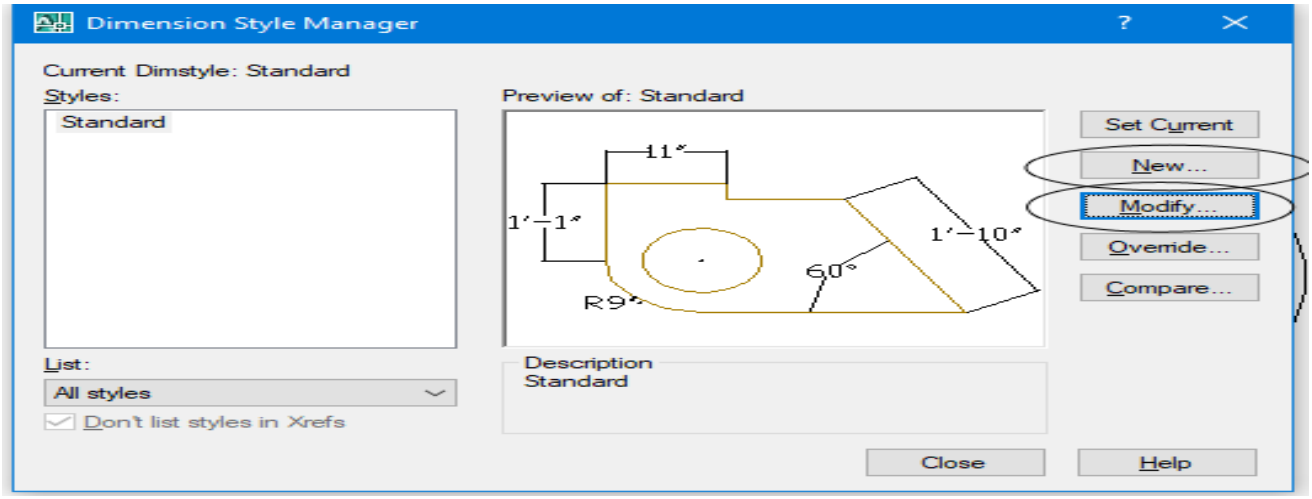


Fig: Dimension Style Manager

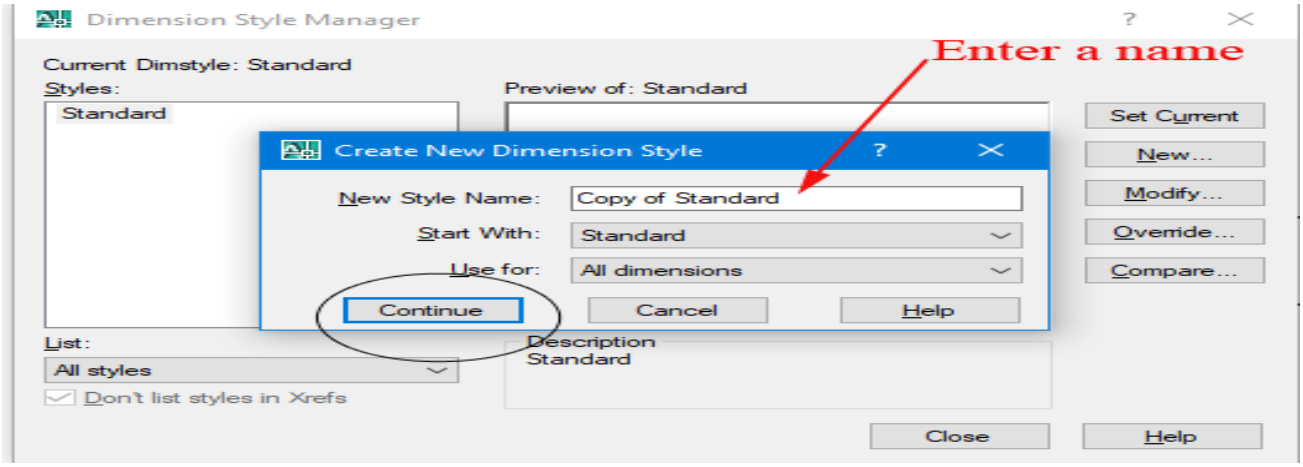


Fig: Creating New Dimension Style

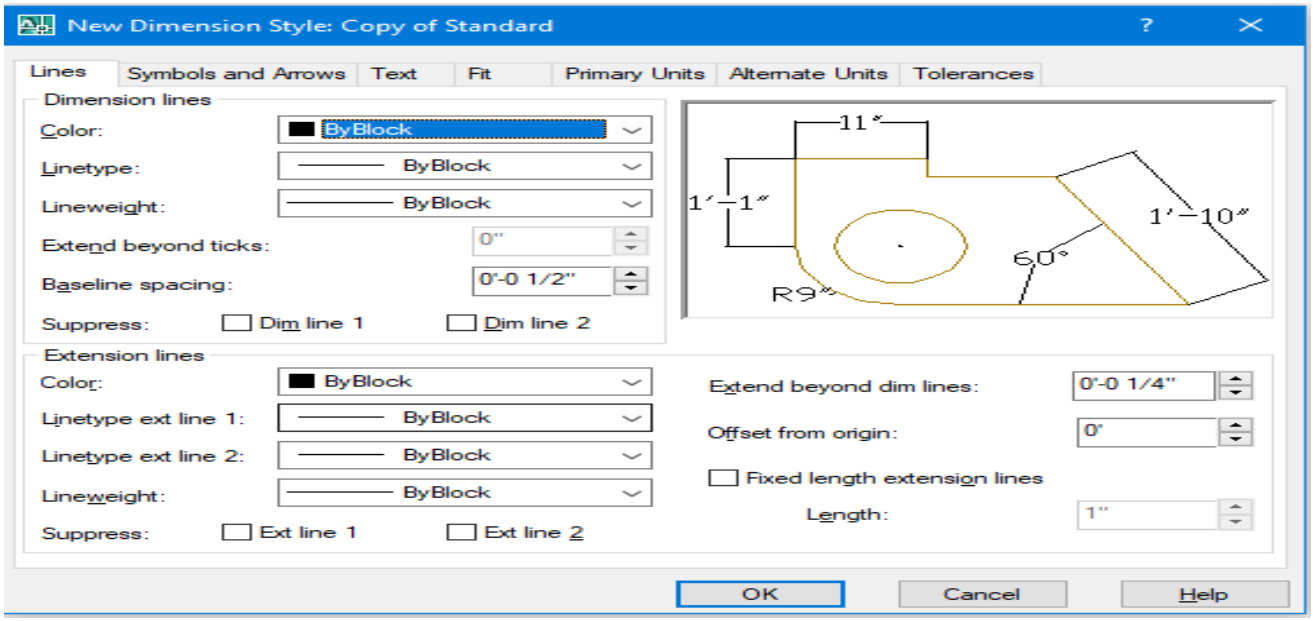


Fig: New Dimension Style window

Layer Toolbar

Layer toolbar is activated by default in AutoCAD. It is situated on top of the drawing window. Layering is an important tool as it can greatly reduce amount of time required for finishing a drawing. If for any reason layer toolbar is not activated it can be activated by right clicking the top right portion of the screen and selecting layer.

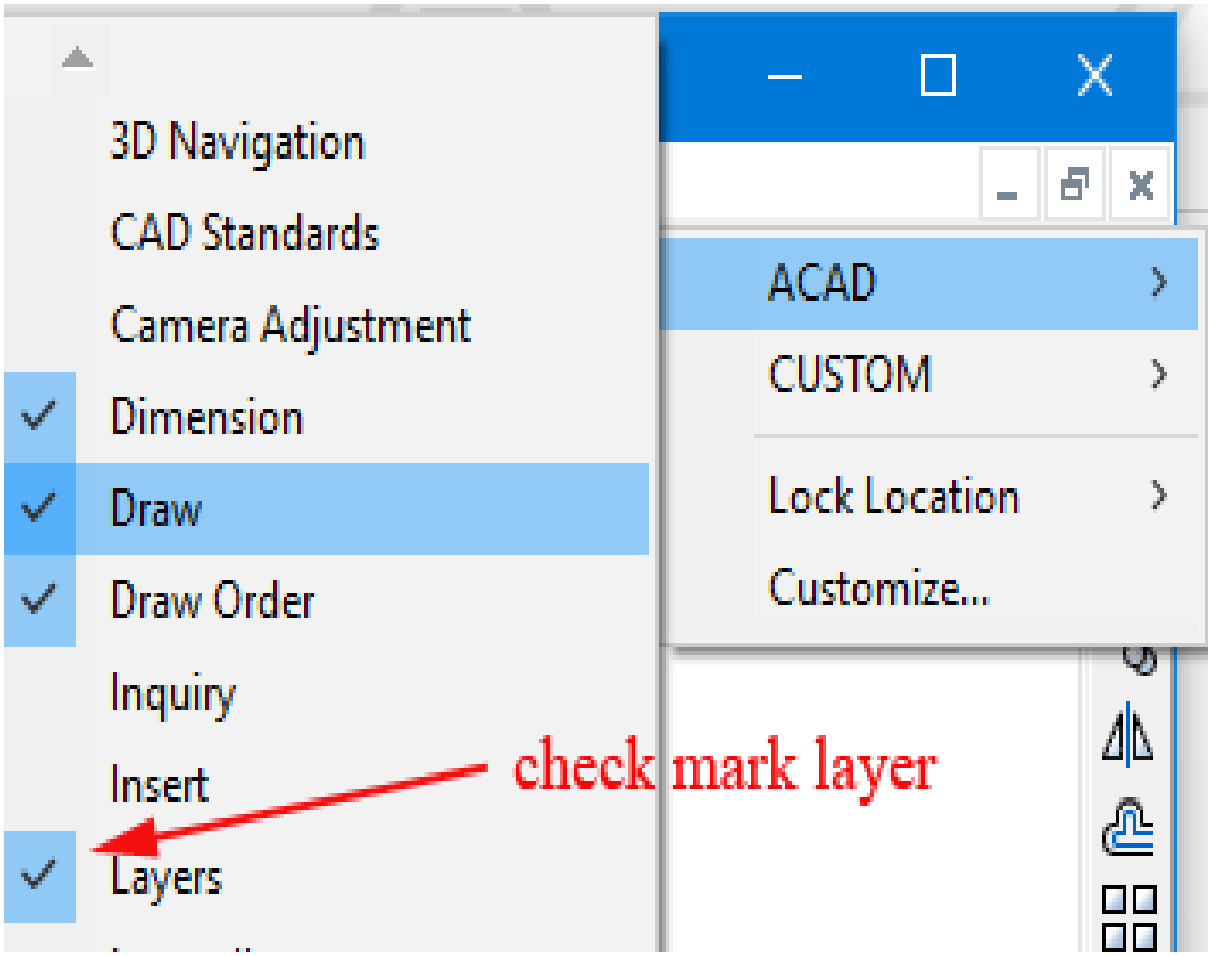


Fig: How to show the layer toolbar

Fig: Layer Toolbar

Command Name: Layer

Command Shortcut: LA

Toolbar Icon: 

Toolbar Name: Layer Properties Manager

Description: A drawing contains many components such as wall, door, window, railing, texture etc. All these components have different properties. Some have different colors while others have different line types. As differentiating different line types is rather time consuming layers can be used in AutoCAD to easily create different types of lines.

Use:

How to create new layer:

1. Go to the layer properties manager by either pressing the icon or by using the shortcut.
2. Click the "New Layer" icon or press ALT+N and a new layer will be created. The user can activate the layer by choosing "Set Current". Exit by pressing OK button.

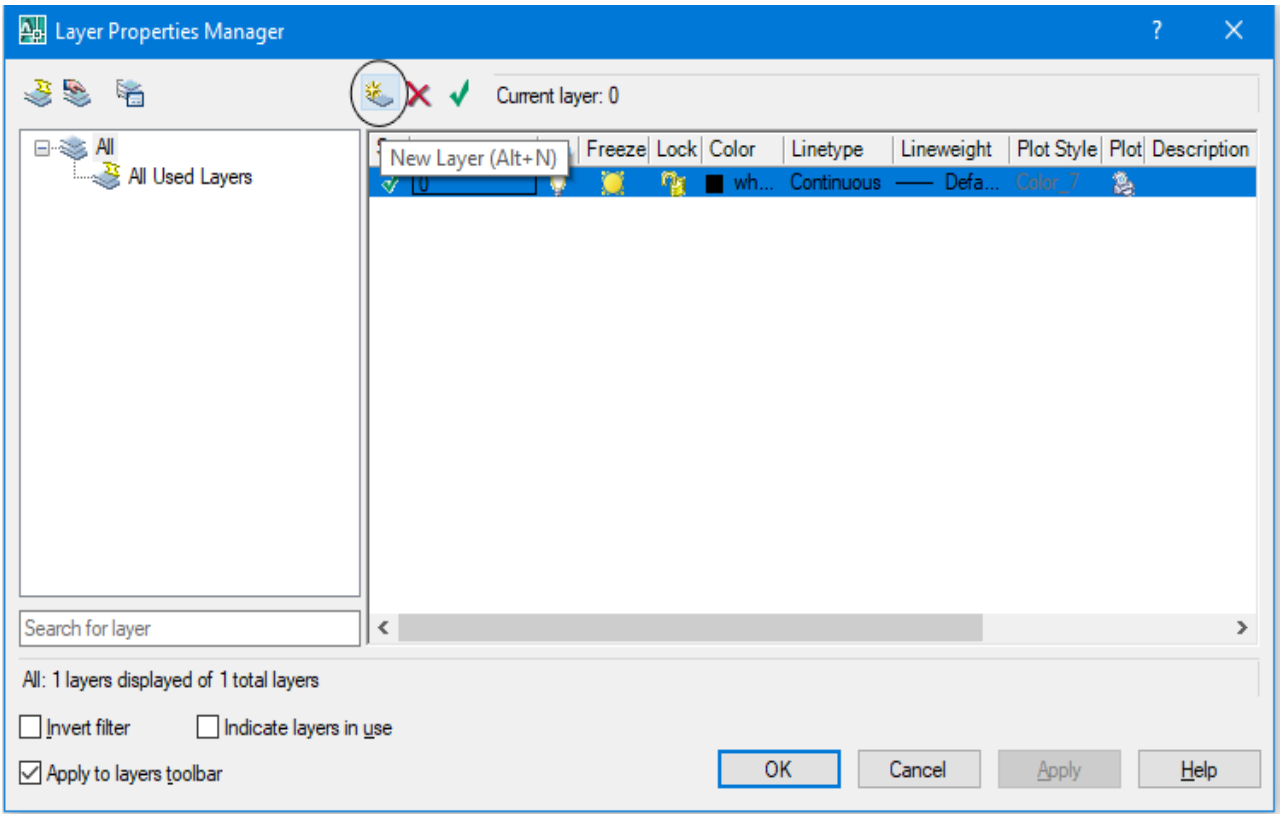


Fig: Layer Properties Manager

How to assign colors to a layer:

1. Open the layer properties manager.
2. Click on the color icon of the desired layer
3. A dialog box called “Select Color” will be shown. Select the desired color and then press Ok.

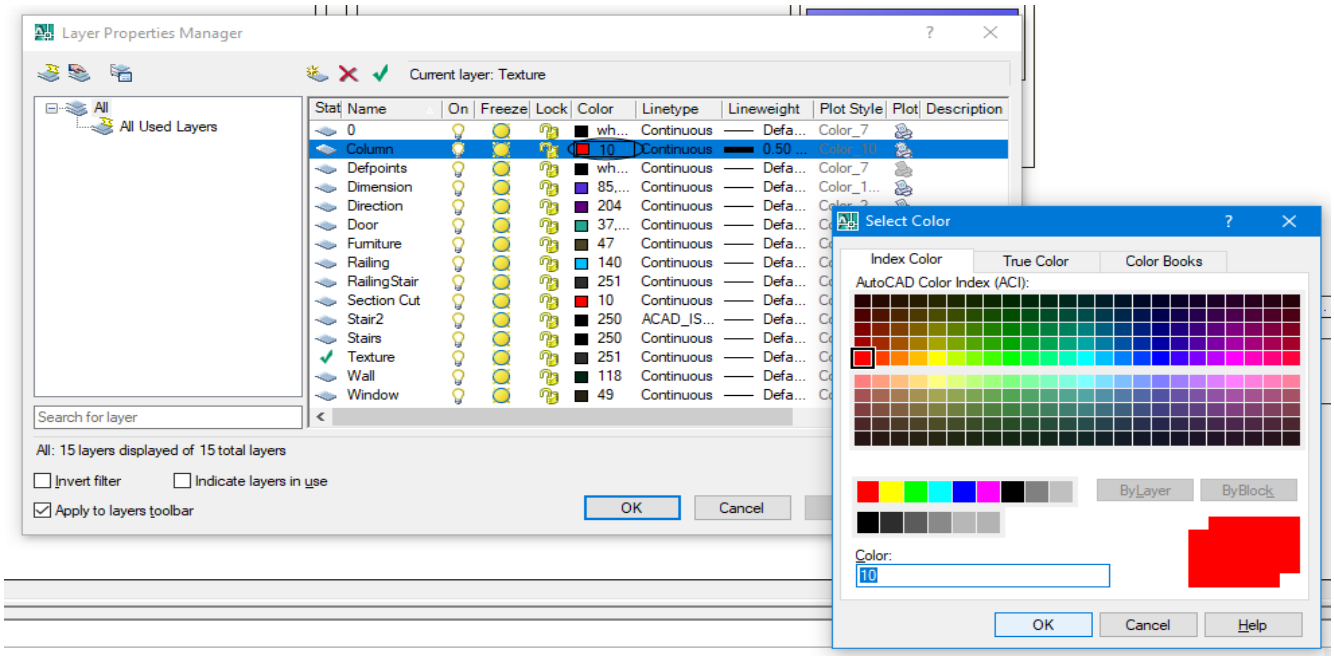


Fig: Assigning color to a layer

How to turn a layer on or off:

1. A layer can be turned on or off by clicking the bulb icon in layer properties manager or the icon the layer icon manager dropdown.

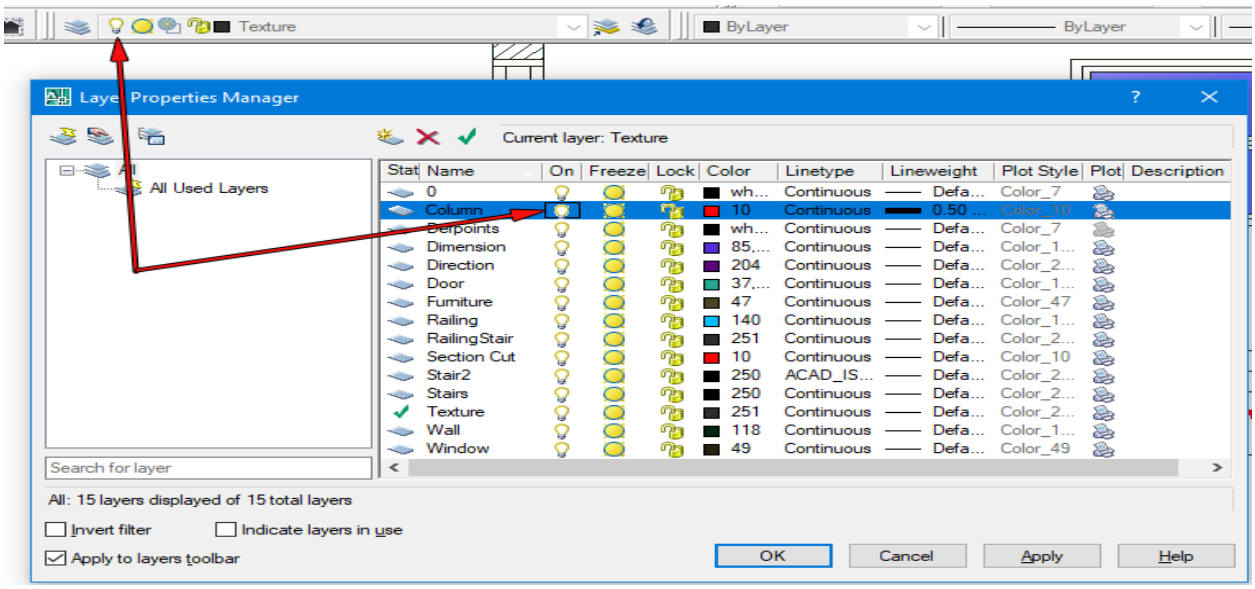


Fig: Turning a layer on or off

How to freeze/unfreeze layer:

1. Freezing a layer makes it temporarily invisible. To freeze a layer click the circle icon.

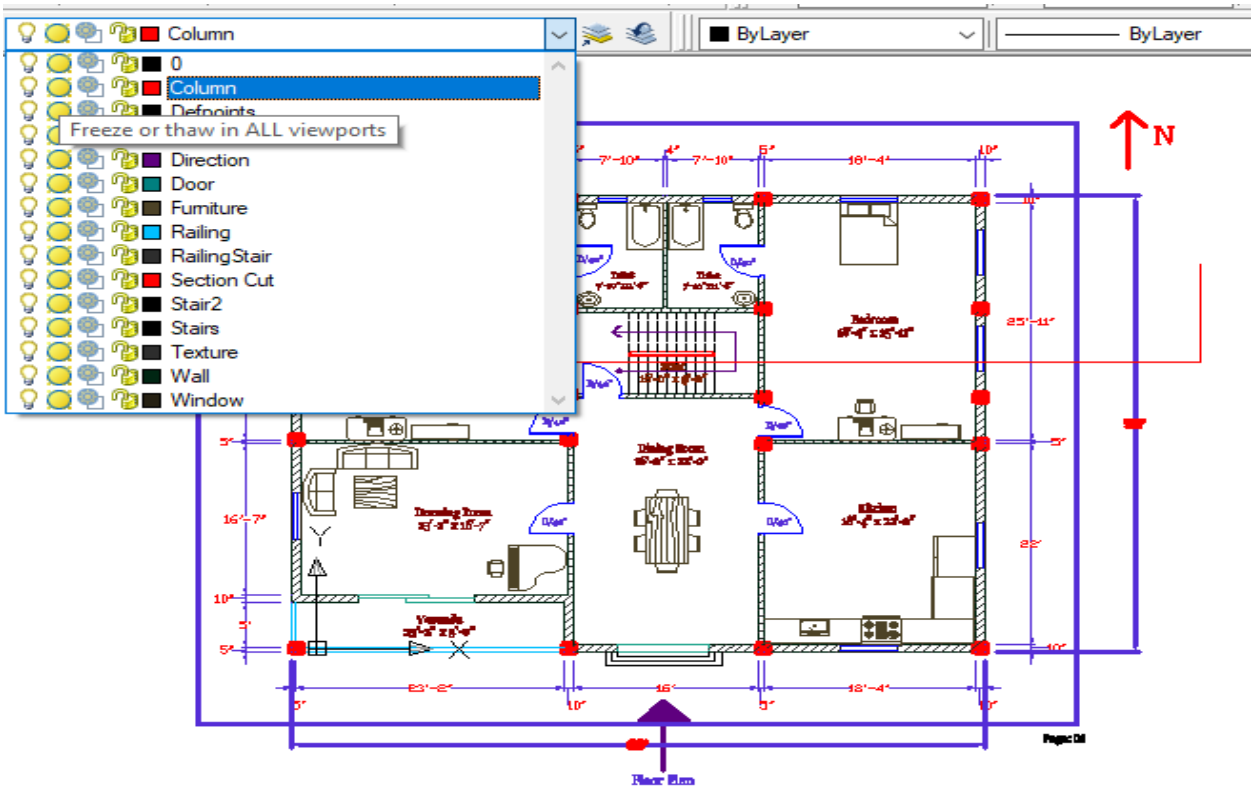


Fig: before freezing & selecting freeze

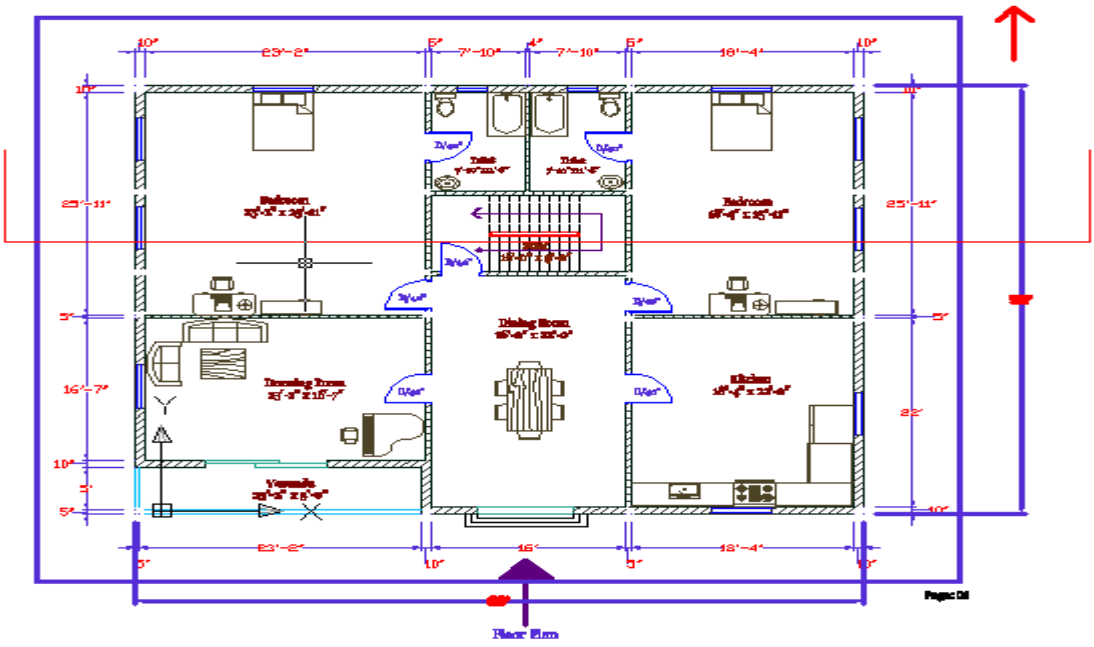


Fig: After freezing

How to lock/unlock layer:

1. A layer is locked when the user wants to work on one layer and keep the rest of the layers in a fixed position
2. Press the Lock/Unlock icon to lock and to unlock layers

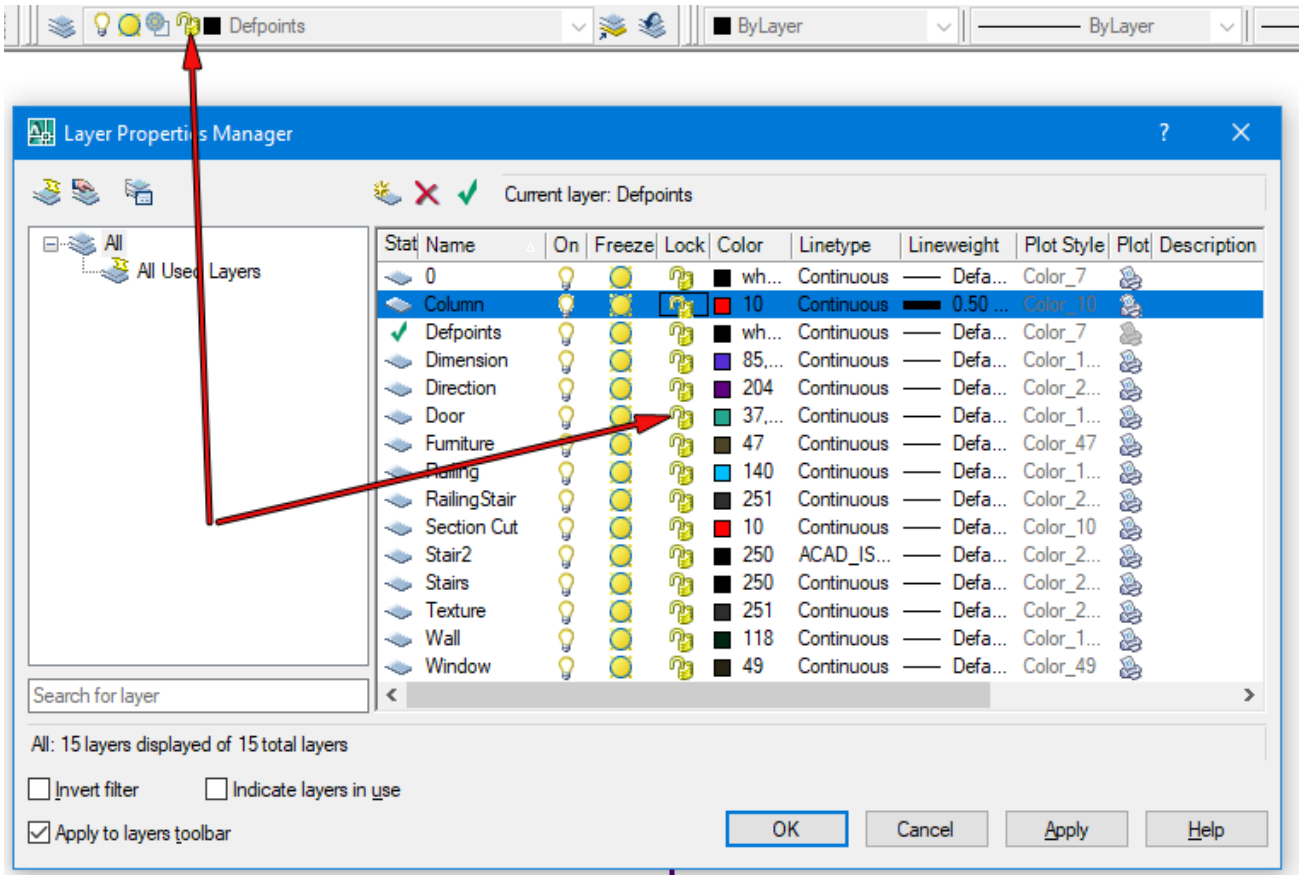


Fig: Lock/Unlock icon

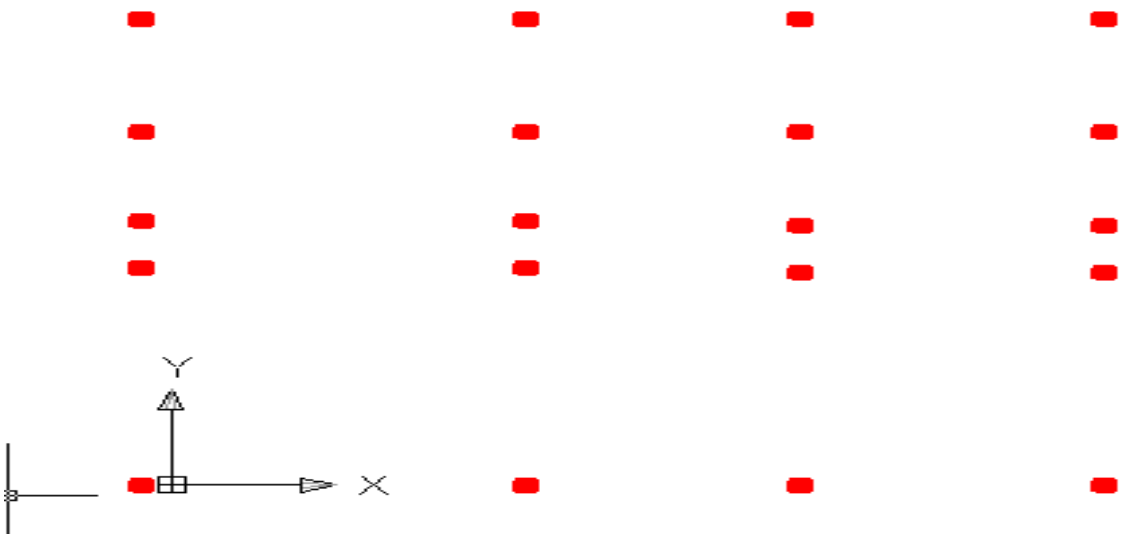


Fig: Using cut on the plan while keeping the column layer locked

Assigning line types:

1. Open Layer Properties manager & click on Line type of the layer that you want to change. It is set to continuous by default.
2. Select Line type window will open which will contain the previously loaded line types. Select Line type window can also be opened by using the shortcut LT.
3. To add more line types click load & then select the line type that you want.

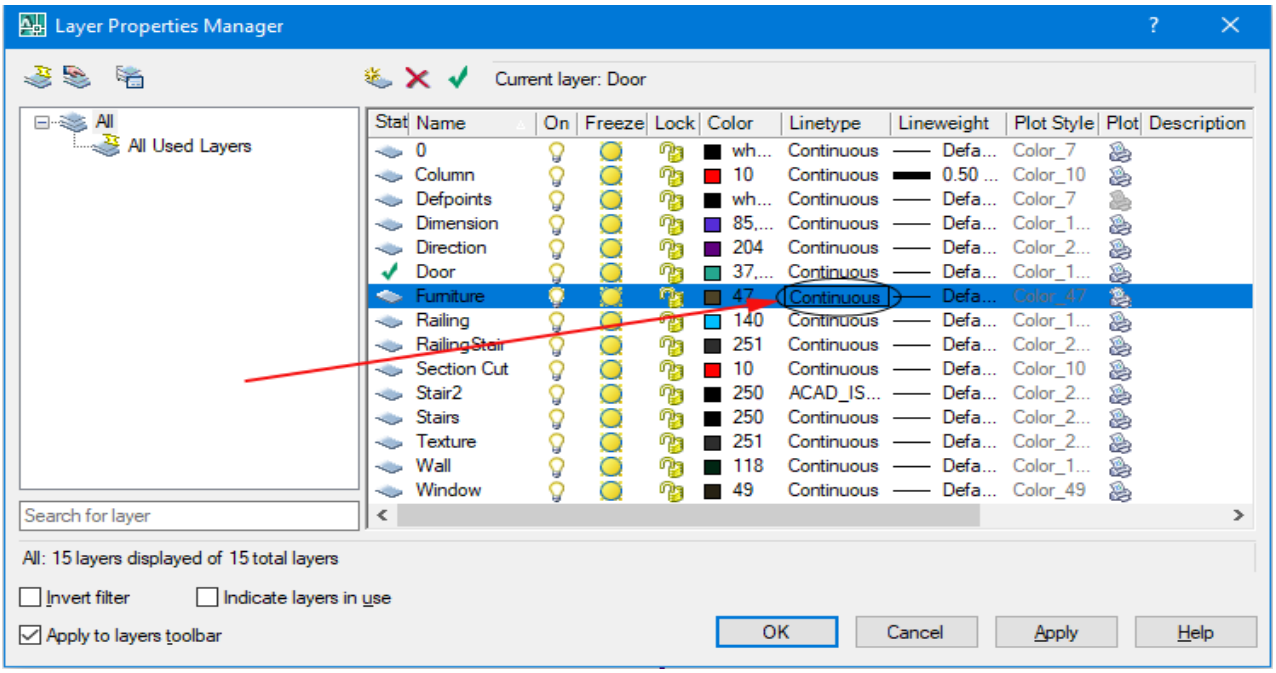


Fig: How to open the select linetype window

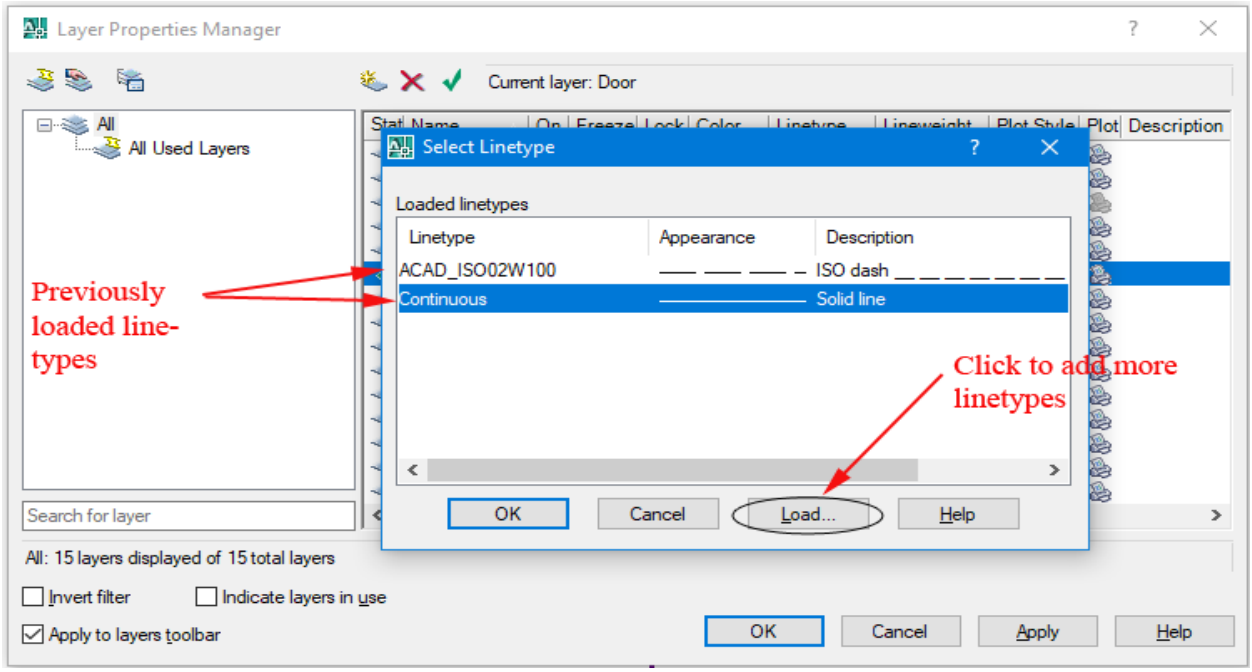


Fig: The select linetype window

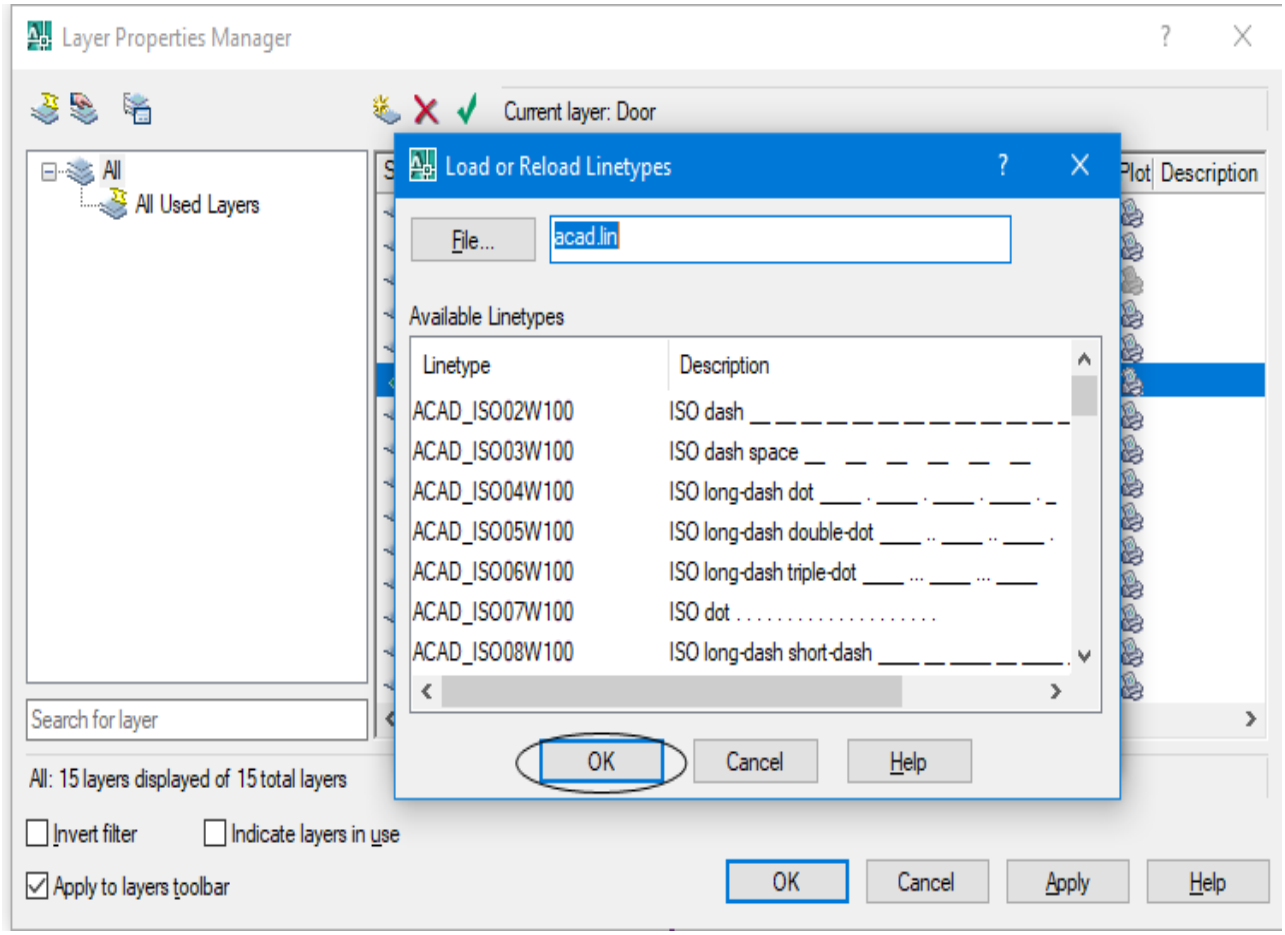


Fig: Load or Reload Linetypes window

How to assign line weight:

1. From the Layer Properties Manager window click on the line weight of the desired layer. By default it is set to 0.01 inch or 0.25 mm.
2. Select the desired line weight from the list and click ok
3. If the line weight is not visible in the drawing, you will have to press the LWT button the status bar on the bottom of the screen.
4. If you don't want to use the assigned line weight on a object in the same layer, you can go to Format → Lineweight or type LW in the command box to open the Lineweight settings window.

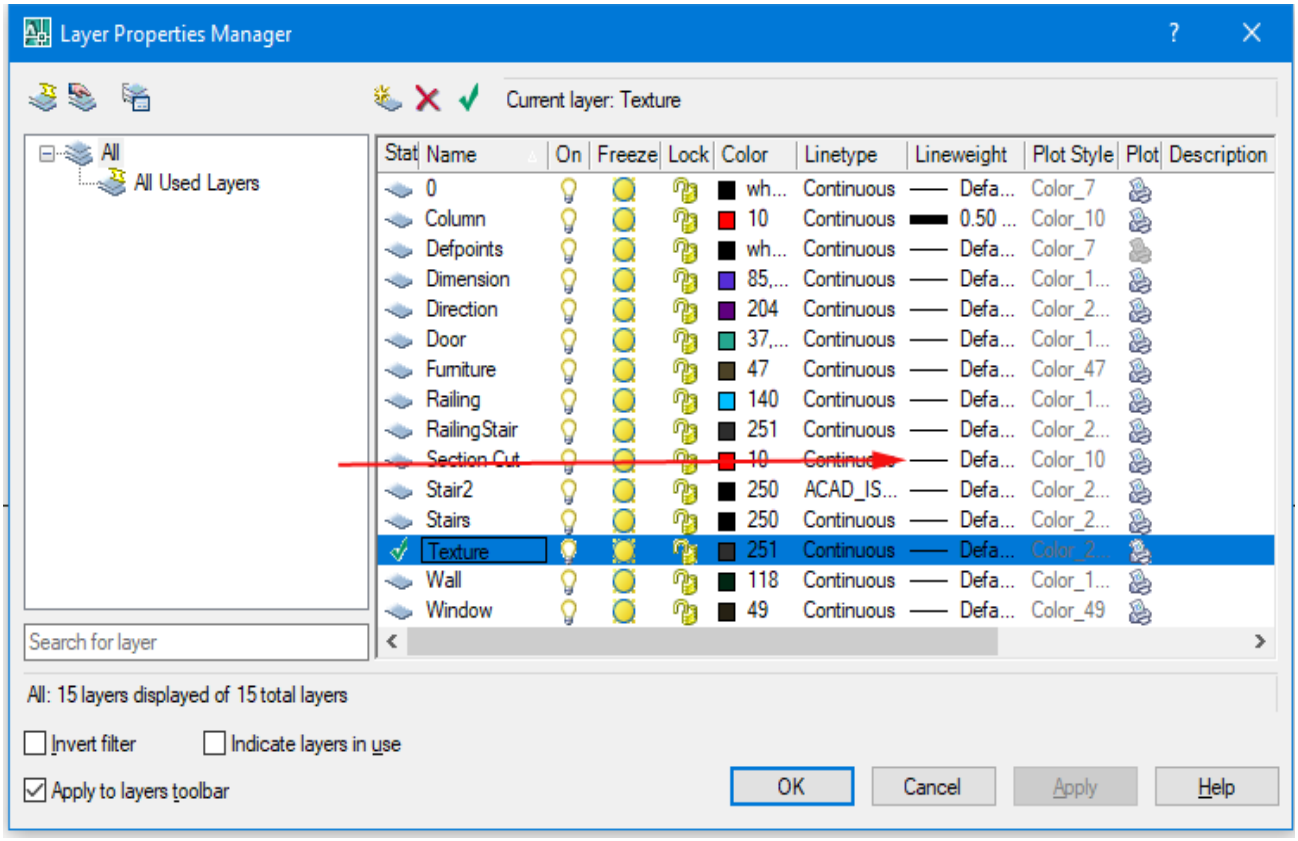


Fig: How to open the Lineweight window

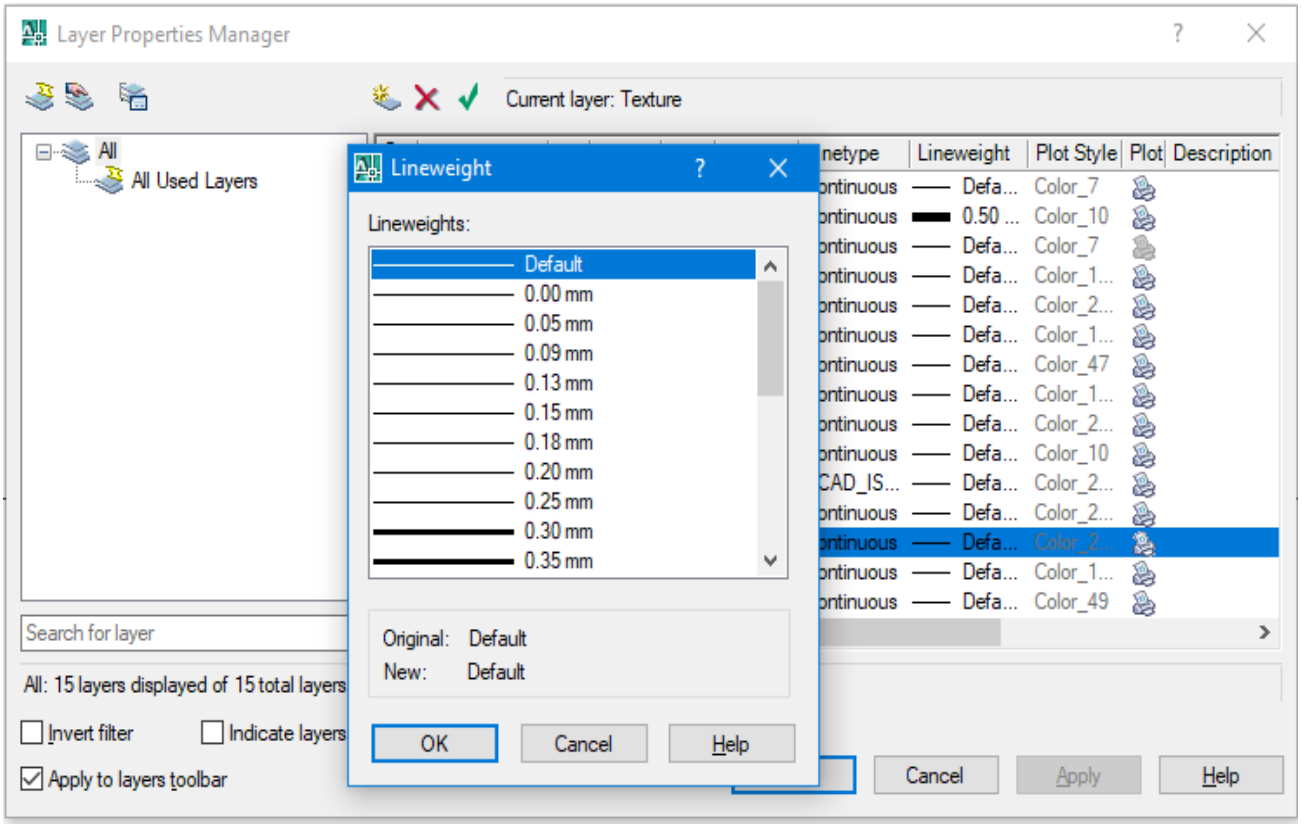


Fig: The Lineweight window

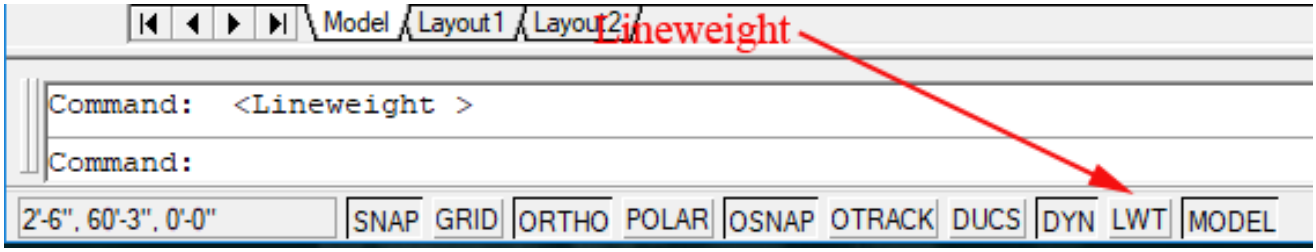


Fig: LWT button for making the line weights visible

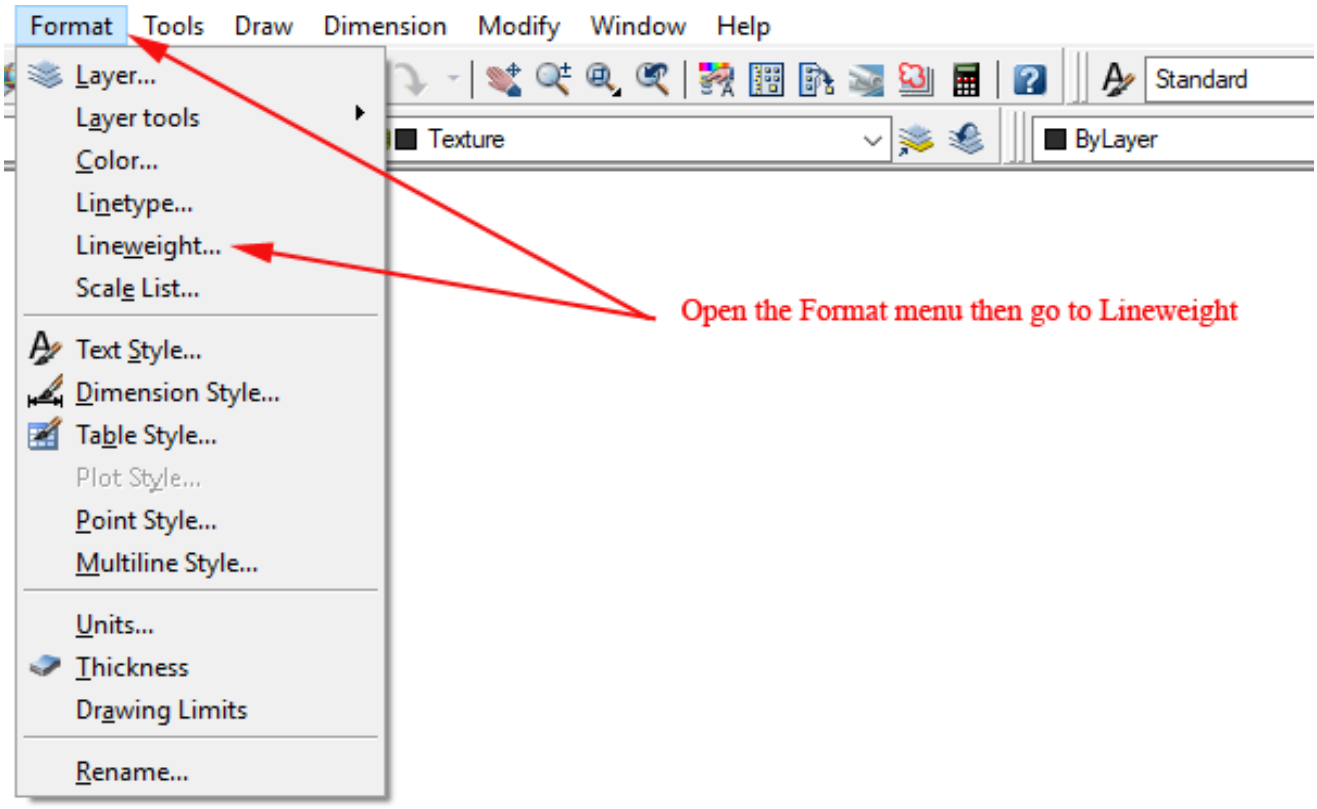


Fig: How to Open the Lineweight Settings window

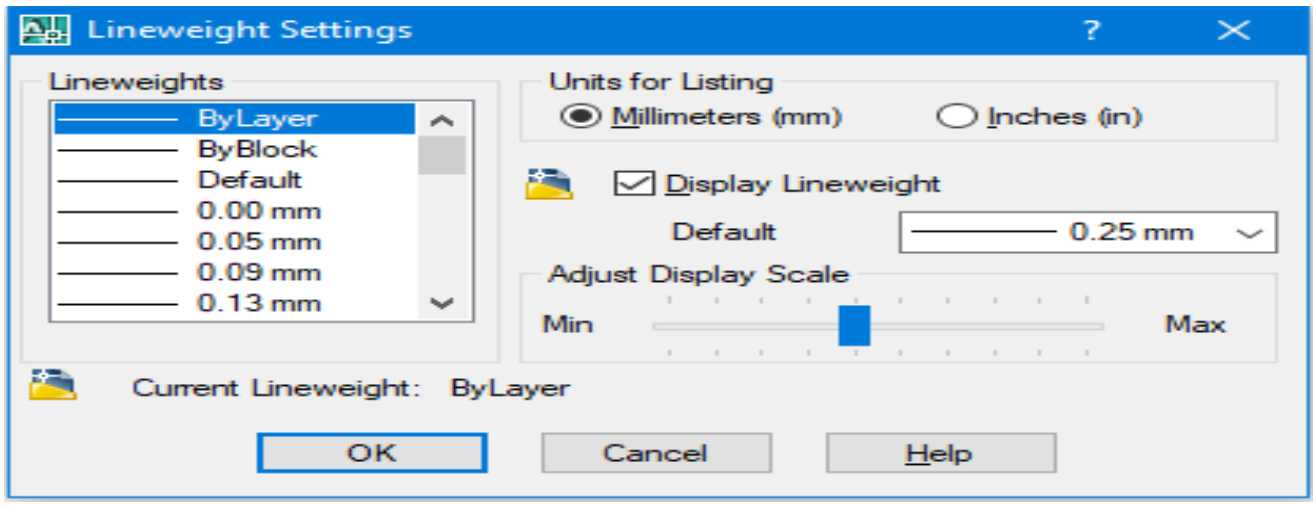


Fig: Adjust line weights for individual objects or lines here

How to disable plotting on a layout:

1. With this feature the user can disable plotting for a layer. As a result that layer will not printed when plotting using a printer or a plotter. Go to Layer Properties Manager.
2. Click on the plot icon to enable or to disable this feature.

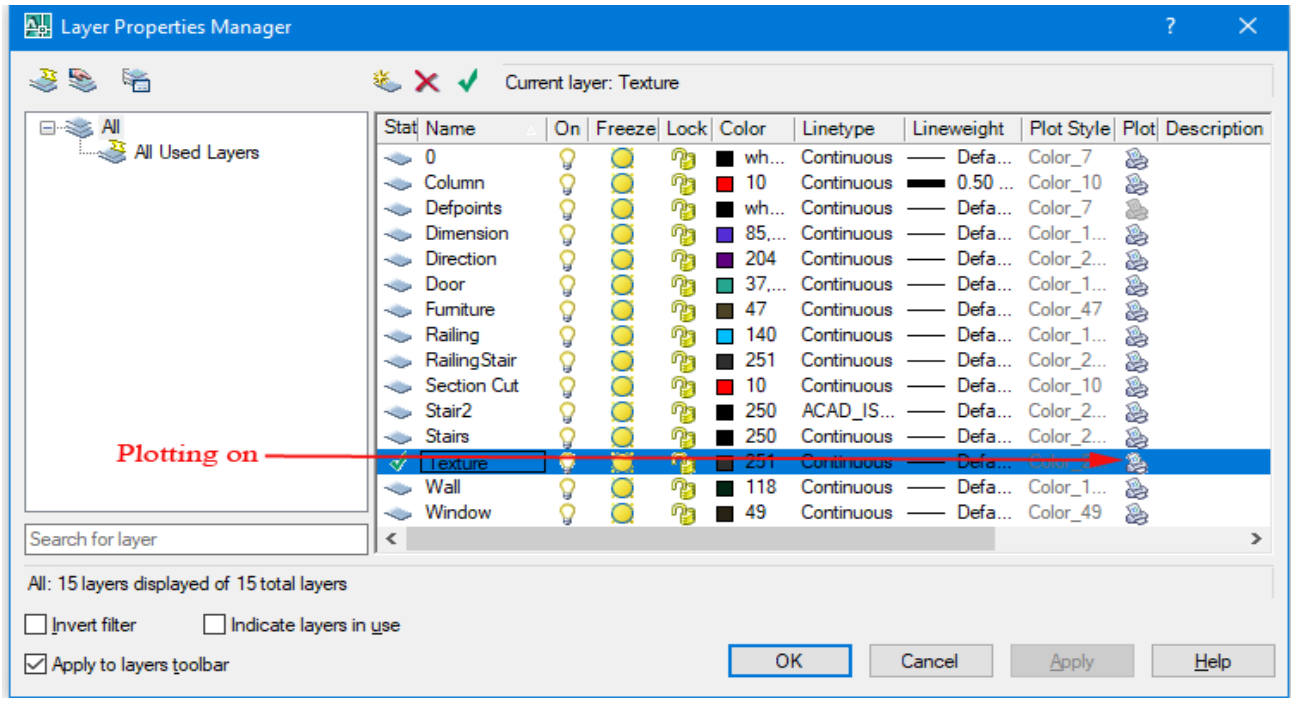


Fig: Plotting On

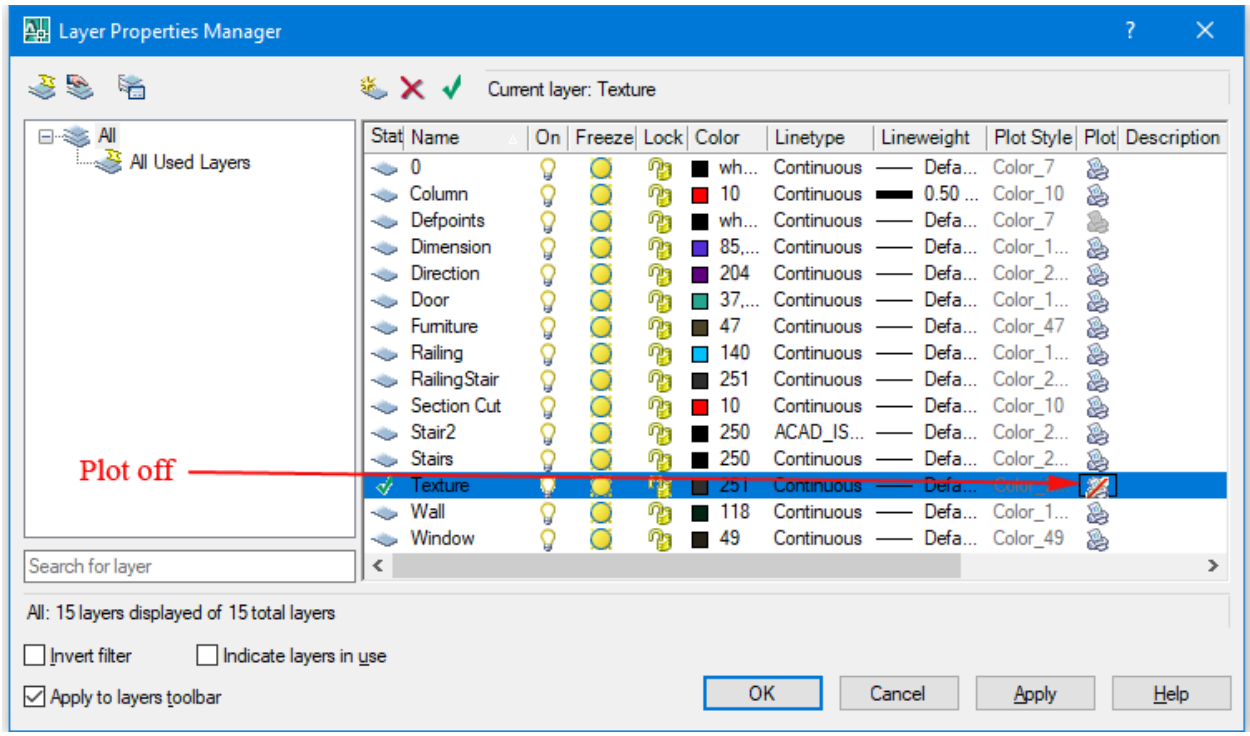


Fig: Plotting Off

Plotting/Printing

Command Name: Plot

Command Shortcut: CTRL + P

Standard Toolbar Icon: 

File Menu Name: Plot…

Description: Plot is used for printing the Drawing. With this command you can print in monochrome or in grayscale, create pdf or an image format etc.

Use:

1. Press CTRL + P or go to File → Plot. This will open the Plot window.
2. From here-
    - **Name:** Give the print setup a name, load previous print setup etc.
    - **Printer/plotter:**
        i. **Name:** Select the printer for printing with the printer or select DWG to PDF.pc3 to print to PDF, Publish toWeb JPG.pc3 for JPG file, Publish toWeb PNG.pc3 for PNG file.
        ii. **Plot to File:** As the name suggests ticking this option creates a file.
    - **Paper Size:** From the dropdown list select the paper that you are using. Most common paper size is A4 210 mm x 297 mm.
    - **Plot Area:** From the dropdown list select window then specify the area that you want to plot using the mouse
    - **Plot Offset:** Tick center the plot.
    - **Plot Scale:** Tick Fit to Paper.
    - Clicking the Arrow button near the Help button will extend the window.
    - **Plot Style Table:** From the dropdown list you can select monochrome or grayscale mode, 100% or 200% screening etc.
    - **Shaded Viewport Options:** Here you can change the Shading of the plot, change the shade quality etc.
    - **Plot Options:** Here you can add Plot stamps, enable background plotting etc.
    - **Drawing Orientation:** Here you can select either Portrait or Landscape printing and tick the option for upside-down plotting.

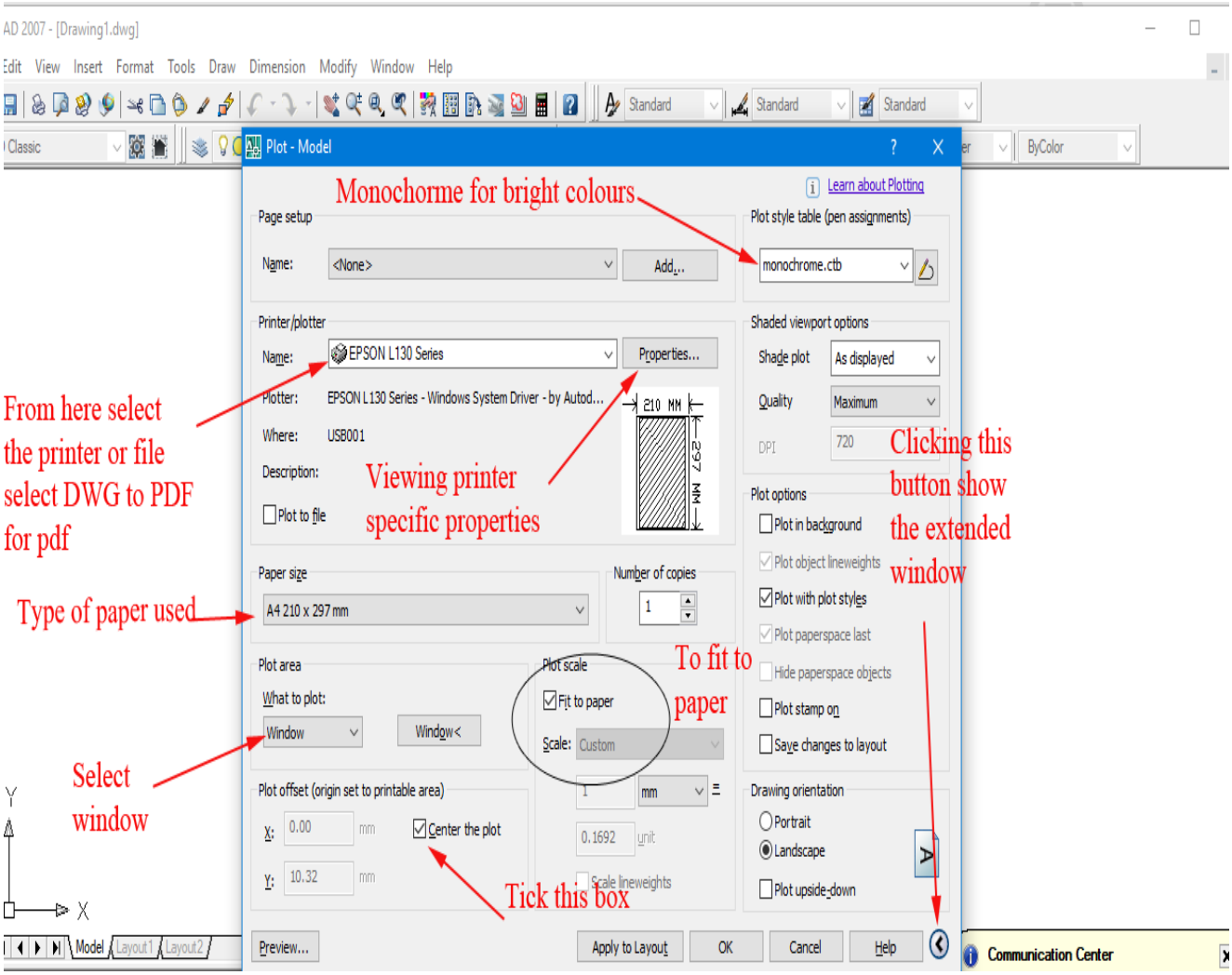


Fig: Plot window & recommended settings

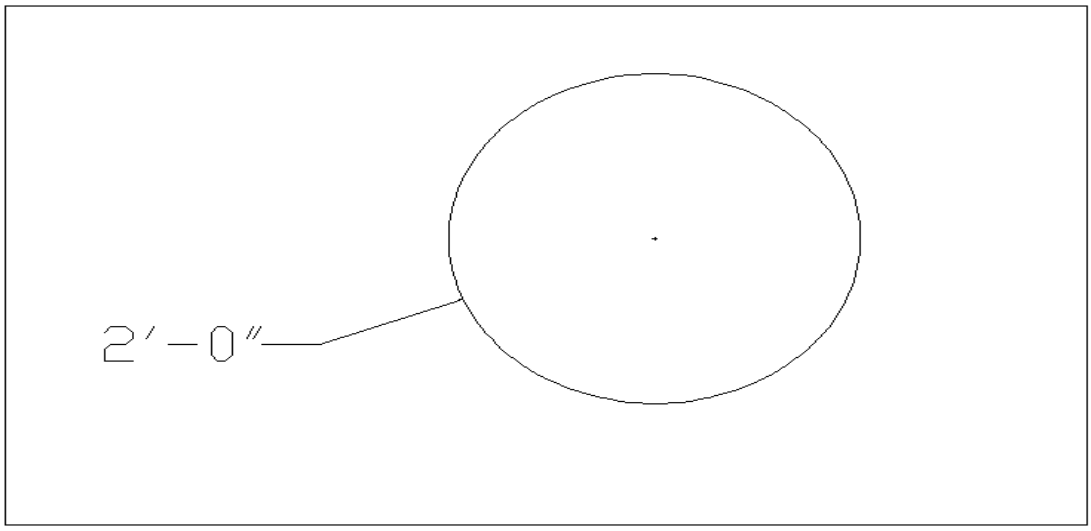


Fig: Selecting the window to be plotted

References:

1. Mastering AutoCAD 2D 3D by Engr. Samual Mallik
2. AutoCAD 2007 (Opened using press **F1** in AutoCAD)
3. http://help.autodesk.com/view/ACD/2017/ENU/?page=commands