

CHAPTER –9

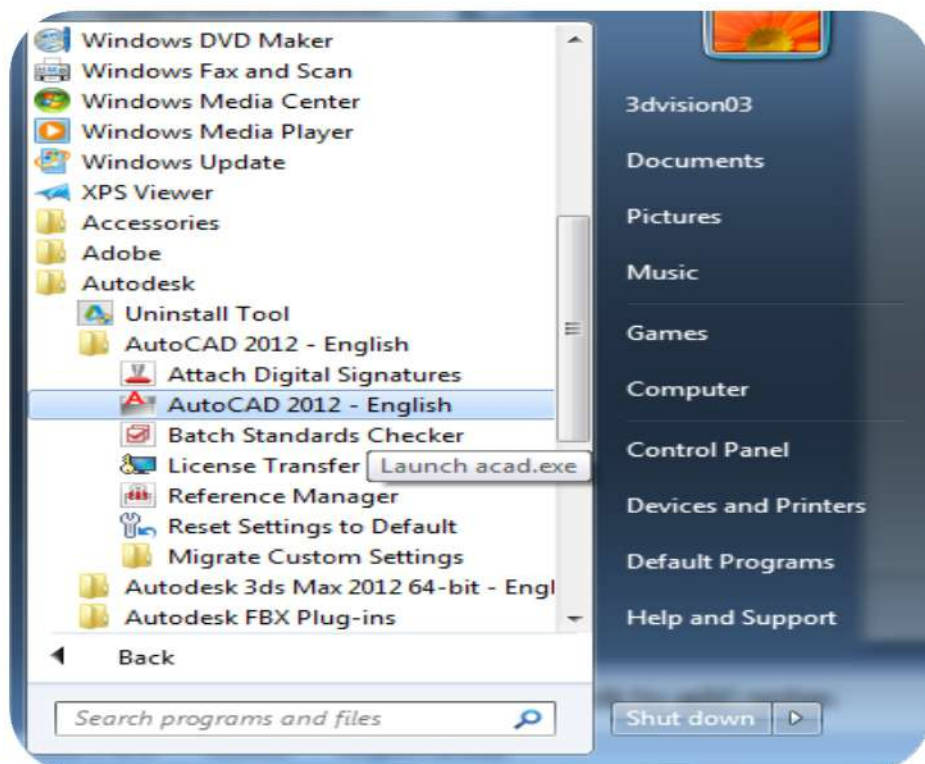
PRACTICE ON AUTOCAD

INTRODUCTION

People have been using visual images to convey ideas for much longer than they have been using the written word, and this method continues even in today's technologically advanced world. Drawings are still used to communicate ideas effectively, but tools used to create drawings have been changed. The advent of computer has created a new collection of tools / softwares for creation of drawings. One of these tools is Computer Aided Design (CAD). As computers have become more advanced in power, speed, and affordability, CAD has been developed as well becoming easier to obtain and use. AutoCAD is one of such CAD software, which is widely used around the world for 2D and 3D computer-aided design and drafting of engineering drawings. It is developed and marketed by Autodesk. AutoCAD was first released in December 1982 by John Walker. It has been subsequently revised and up-graded. At present window based AutoCAD- 2012 is available in the market.

How to open AutoCAD ?

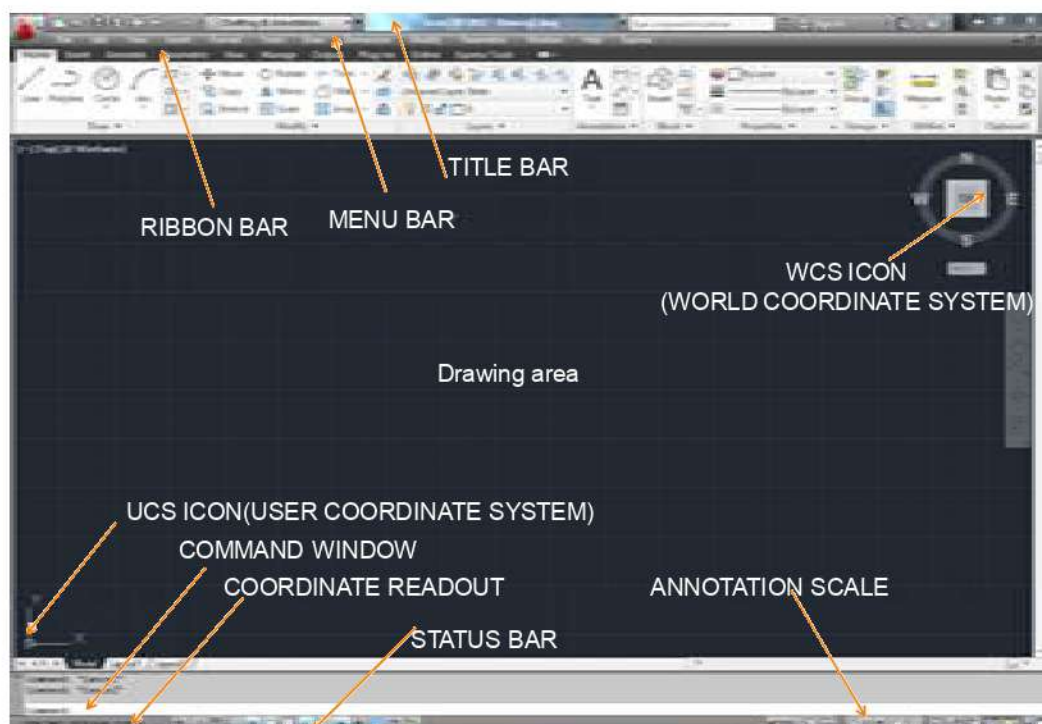
- Choose Start from the Windows program manager.
- Choose Programs - Autodesk- AutoCAD 2012.
- Click the AutoCAD 2012 for Windows icon.



- OR
- Choose the AutoCAD 2012 icon from the desktop as shown below.

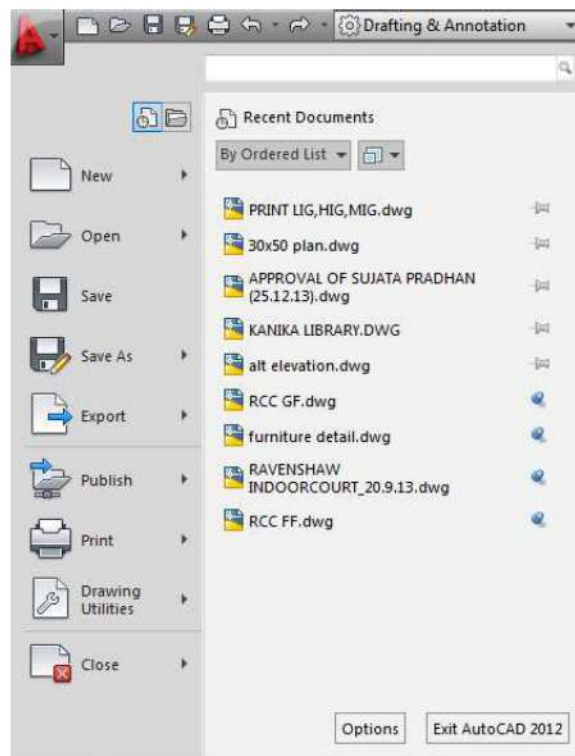


AutoCAD 2012 application window appears as shown below.



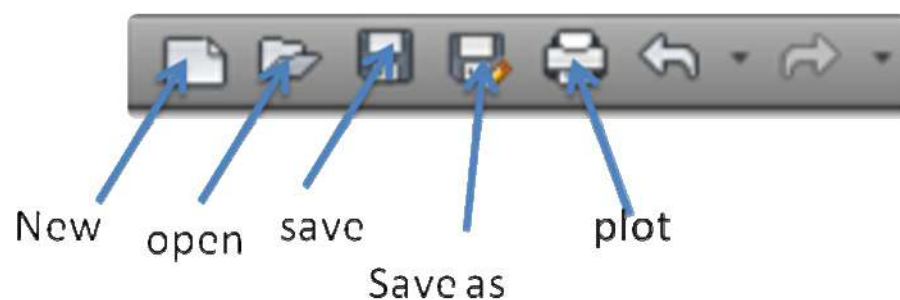
Menu browser

- Menu browser is represented by the letter A in red.
- It provides recently accessed documents.
- It consists of the file, open, save, save as, export, print, drawing utilities etc.



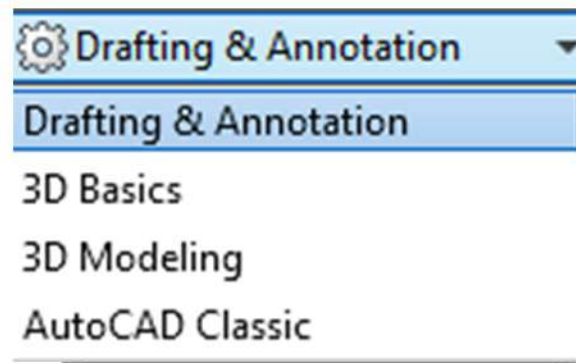
Quick access toolbar:

It includes commonly used options such as new, open, save, plot, undo, and redo etc.



Workspaces

- It is a collection of menus, toolbars, palettes & ribbon controls panel that are grouped and organized so that you can work in a custom, task-oriented drawing environment.
- Four types workspaces are defined in Autocad 2012.
- Those are
 - Drafting & Annotation
 - 3D Modeling
 - 3d basics
 - AutoCAD Classic

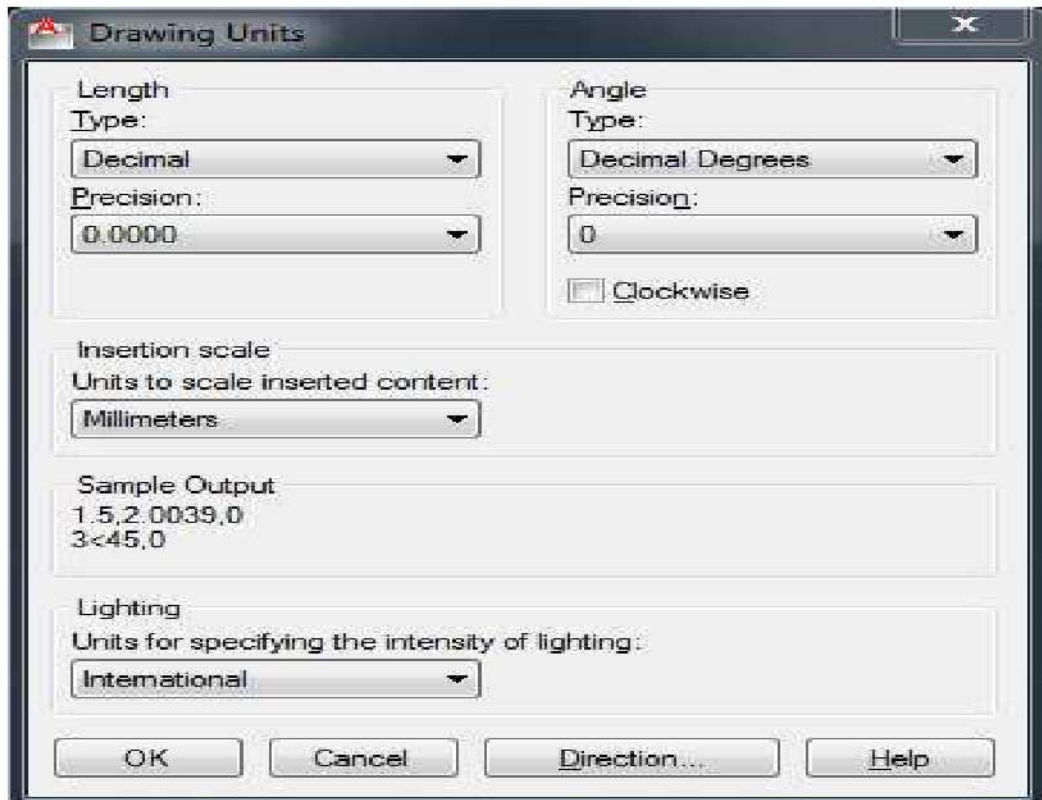


Function Keys

- F1- Help
- By pressing the F1 key actually enables the help. Further Pressing F1 while a tooltip or command is active displays Help for that command. Choosing the Help button in a dialog box displays Help for that dialog box.
- F2- Text Window
- By pressing the F2 key actually enables / activates the text window showing the previous command line activity (command history)
- F3- Osnap
- By pressing the F3 key actually enables / activates running object snap modes. Ways to access.
- F4- 3D Osnap
- F5- IsoPlane
- By pressing the F5 key activates the *Isoplane*. The isometric plane affects the cursor movement keys only when Snap mode is on and the snap style is Isometric. If the snap style is Isometric, Ortho mode uses the appropriate axis pair even if Snap mode is off. The current isometric plane also determines the orientation of isometric circles drawn by ELLIPSE.
- F6- Dynamic user Co-Ordinative system (DUCS)
- By pressing the F6 key activates the Dynamic UCS on or off for 3D Modeling
- F7- Grid
- By pressing the F7 key activates the Grid command and either turns the GRID ON or GRID OFF
- F8- Ortho
- By pressing the F8 key activates the ORTHO command to ON or OFF. Activating the ORTHO helps us create lines in straight (linear) Vertical or Horizontal
- F9- Snap
- By pressing the F9 key activates the SNAP command to ON or OFF. Further it restricts cursor movement to specified intervals. (Snap to grid)
- F10- Polar
- By pressing the F10 key activates the Polar Tracking ON or OFF. Further Polar tracking restricts cursor movement to specified angles. Polar Snap restricts cursor movement to specified increments along a polar angle.

How to set the units

- Open a new drawing page in the autocad.
- Then units shortcut 'UN' Enter in the command window.
- Select the type of unit & precision under the length.
- For example we select the Decimal type unit & precision is 0.000 unit



How to set the Limits

- In command bar type limits then press Enter
- Reset Model space limits
ON/ OFF/ <Lower left corner> <0,0 or current setting here>: Type in an absolute coordinate or press Enter to select default setting. **NOTE: It is best to leave the Lower Left setting at the 0,0 .**

--Prompt: Upper right corner <12,9 or current setting here>: Type in a new absolute coordinate value for this corner and press Enter. **NOTE: The default limits values in AutoCAD are 12 and 9. If you change to Metric units, the numbers change to the equivalent value in millimetres.**

How to draw a line in AutoCAD?

- LINE:- creates straight line segments
- Command: L press Enter
- Autocad will ask to specify the starting point of a line. Just one click on the screen or using coordinate axis with numerical values will specify the onset point.
- Then program will ask to end the line with another point.
- Then click somewhere on the screen or use coordinate axis with numerical values as well.
- For example :- A line draw in 25mm distance
- Command: L enter

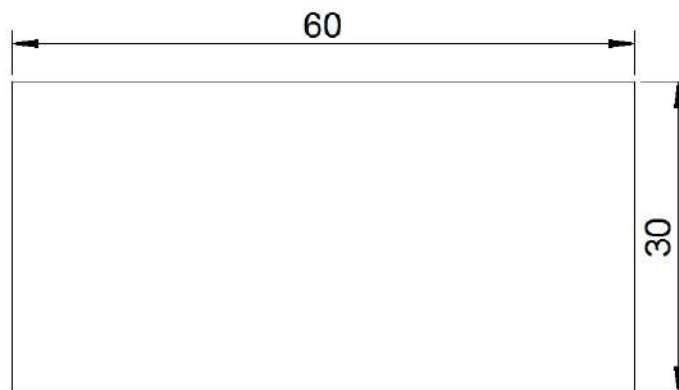
- LINE Specify first point: 0
- Specify next point or [Undo]: 25



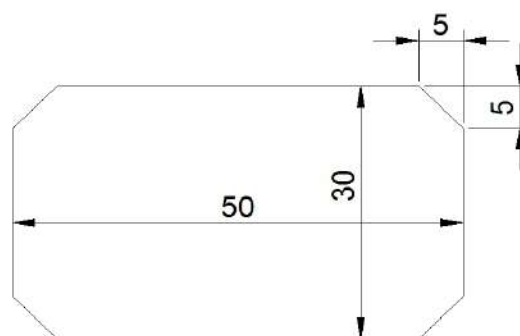
How to draw a rectangle in AutoCAD?

RECTANGLE:-The Rectangle command is used to draw a rectangle whose sides are vertical and horizontal. The Rectangle command also has a number of options like Chamfer, Elevation, Fillet, Thickness or Width.

- Command: REC or rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: pick point
Specify other corner point or [Area/Dimensions/Rotation]: @60,30



- Rectangle with Chamfer
- Command:rec _rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: C
Specify first chamfer distance for rectangles <0.0000>: 5
Specify second chamfer distance for rectangles <5.0000>: 5
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: pick point
Specify other corner point or [Area/Dimensions/Rotation]: @50,30



- Rectangle with Elevation

- Command: rec _rectang

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

Specify the elevation for rectangles <25.0000>: 25

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: 0,0

Specify other corner point or [Area/Dimensions/Rotation]: @50,30

Command: RECTANGLE

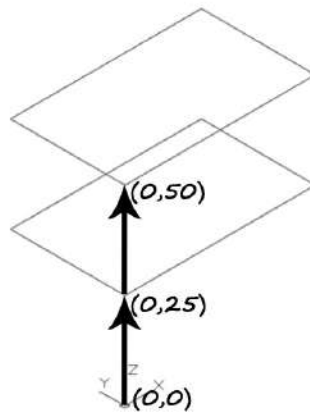
Current rectangle modes: Elevation=25.0000

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

Specify the elevation for rectangles <25.0000>: 50

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: 0,0

Specify other corner point or [Area/Dimensions/Rotation]: @50,30



○

- Rectangle with Fillet

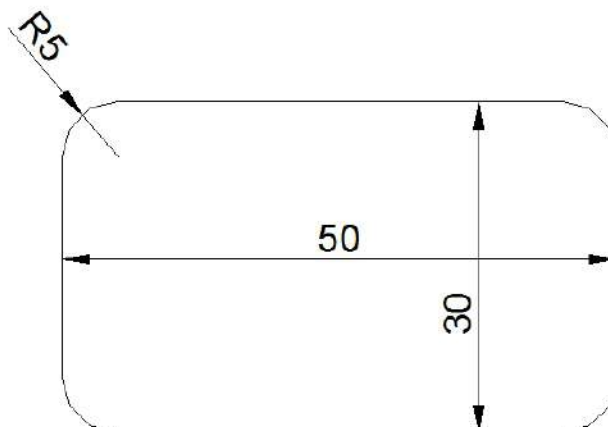
- Command: rec _rectang

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

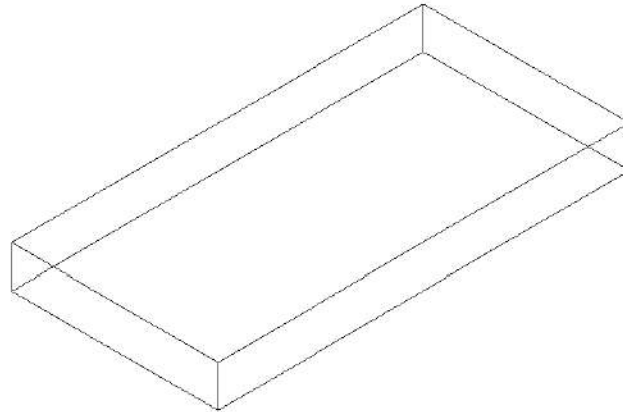
Specify fillet radius for rectangles <5.0000>: 5

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: 0,0 or pick point

Specify other corner point or [Area/Dimensions/Rotation]: @50,30

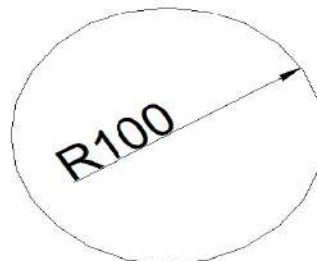


- Rectangle with Thickness
- Command:rec _rectang
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: T
Specify thickness for rectangles <0.0000>: 10
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: 0,0
Specify other corner point or [Area/Dimensions/Rotation]: @100,50



How to draw a circle in AutoCAD?

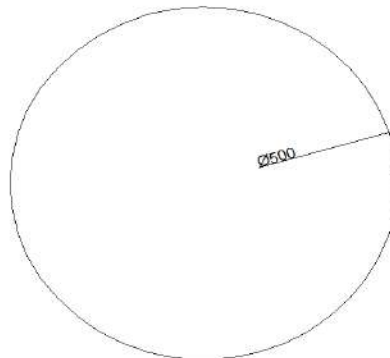
- Circle: create circles, various combinations of centre, radius, diameter, points on the circumference, and points on other objects.
- Keyboard Command: C
- CIRCLE command has 5 options for drawing circles.
- Circle by Centre - Radius
- Write C in command window and press Enter
CIRCLE Specify centre point for circle or [3P/2P/Ttr (tan tan radius)]:pick a point
- Specify radius of circle or [Diameter]: 100



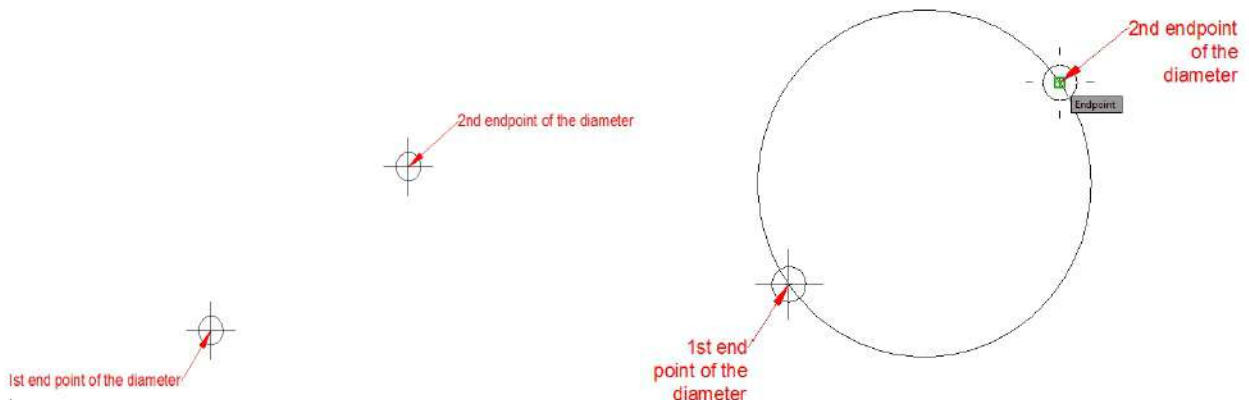
- Circle using Center - Diameter
- Command:C
CIRCLE Specify centre point for circle or [3P/2P/Ttr (tan tan radius)]:

Specify radius of circle or [Diameter] <0.0>: D

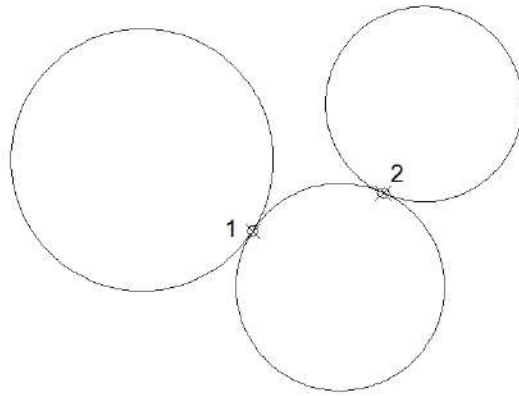
Specify diameter of circle <0.0>: 500



- Circle with the 2 Point
- This method is simpler than the first one because it only requires to pick two points on the screen to make the circle. It needs the two endpoints of a diameter of the circle.
- Command: C and press Enter
CIRCLE Specify centre point for circle or [3P/2P/Ttr (tan tan radius)]: 2p
Specify first end point of circle's diameter:
- Specify second end point of circle's diameter.

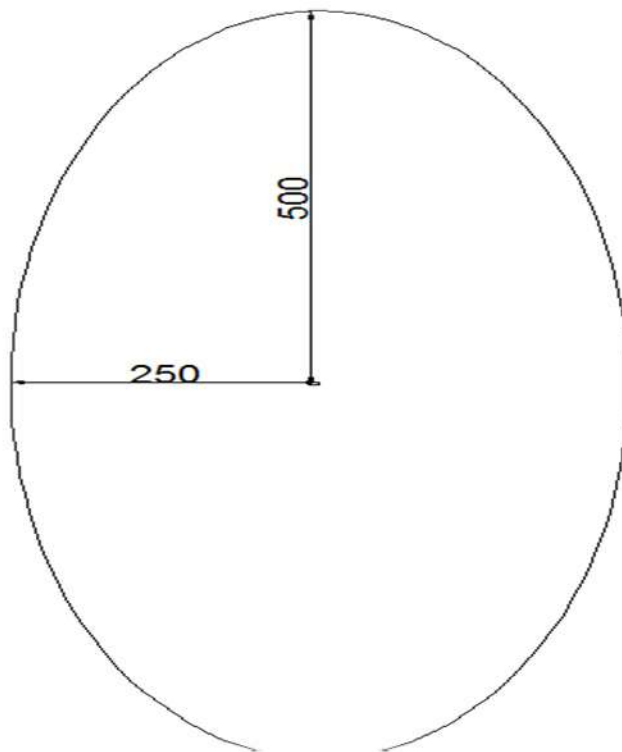


- Drawing the Tangent Tangent Radius circle
- Draws a circle with a specified radius tangent to two objects.
- Sometimes more than one circle matches the specified criteria. The program draws the circle of the specified radius whose tangent points are closest to the selected points.
- Command: C and press Enter
CIRCLE Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: t
 - Specify point on object for first tangent of circle:
 - Specify point on object for second tangent of circle:
 - Specify radius of circle <1413.0504>: 1500



How to draw an ellipse in AutoCAD?

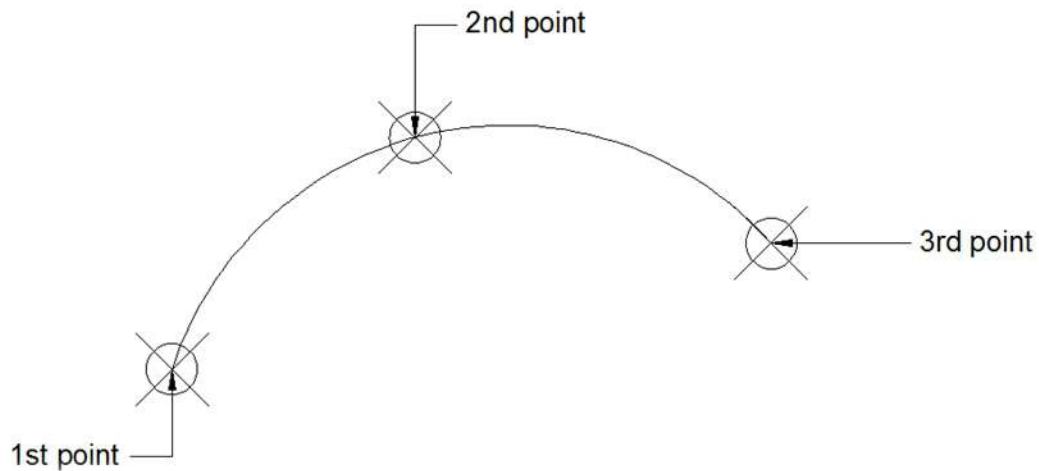
- Center ellipse
- Command: ellipse
- Specify axis endpoint of ellipse or [Arc/Center]: c
- Specify center of ellipse: pick point
- Specify endpoint of axis: <Ortho on> 250
- Specify distance to other axis or [Rotation]: 500



How to draw an arc in AutoCAD?

- 3 points Method
- Command: a and press Enter
- ARC Specify start point of arc or [Center]: pick 1st point
- Specify second point of arc or [Center/End]:

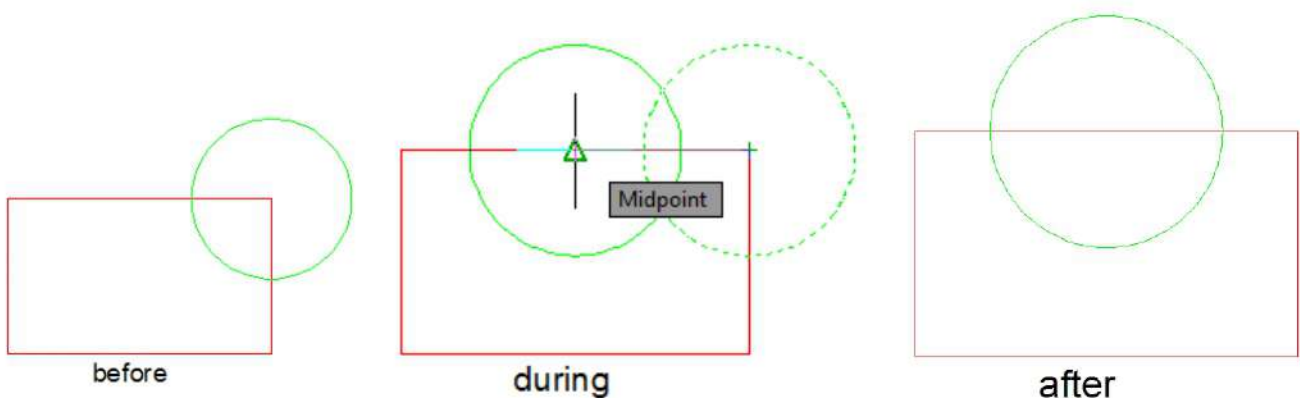
- Specify end point of arc:



EDIT COMMANDS and MODIFY COMMANDS IN AutoCAD

How to use move command AutoCAD?

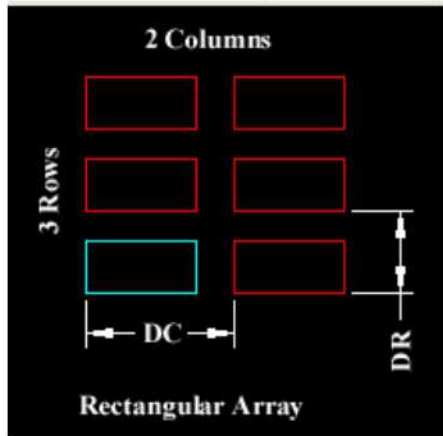
- Move: This command helps in moving an object using the distance & direction specified by a base point followed by a second point.
- Keyboard Command: M
- Command Sequence
- Command: m _MOVE
- Select objects: Specify opposite corner: 1 found
- Select objects:
- Specify base point or [Displacement] <Displacement>:
- Specify second point or <use first point as displacement>:



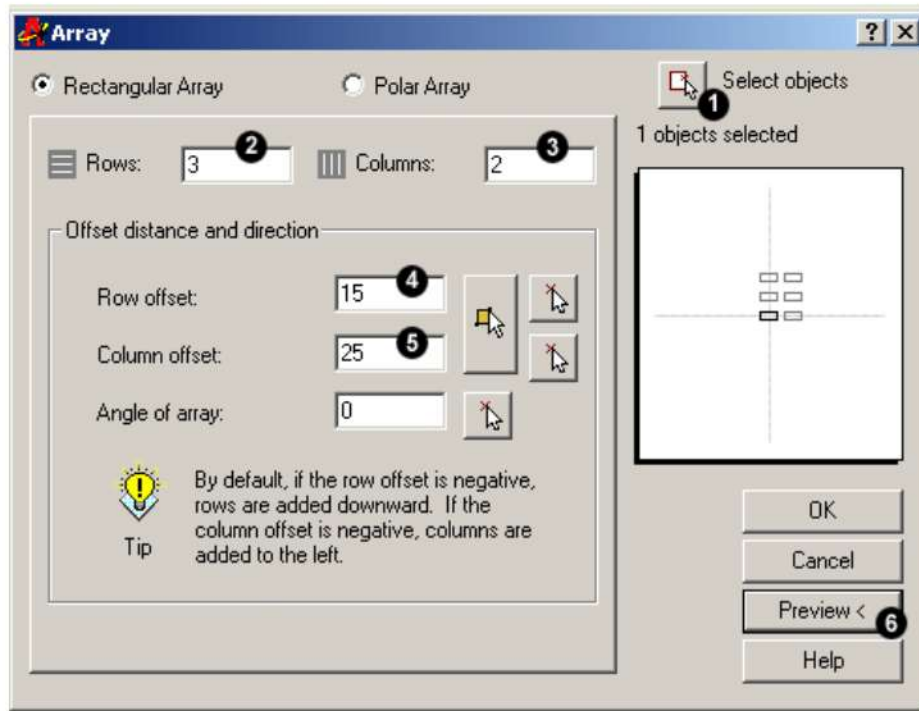
How to use array command in AutoCAD?

- Array- The Array command makes multiple copies of selected objects in a rectangular matrix (columns and rows) or a polar (circular) pattern.
- keyboard Command: AR
- The Rectangular Array

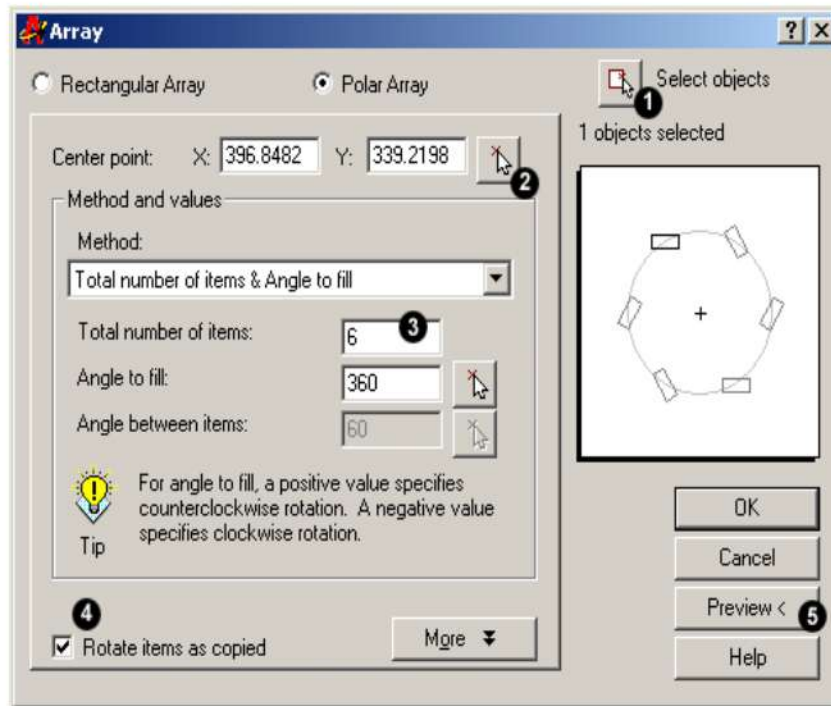
- The illustration on the right shows the results of a rectangular array with two columns and three rows. The distance between rows is indicated with the dimension DR and between columns with DC.
- When creating rectangular arrays it is important to remember that new rows are created above the original object and new columns are created to the right of the original object.



- Command: AR ARRAY
 Select objects: <SELECT THE RECTANGLE> 1 found
 Select objects: <ENTER>
 Enter array type [Rectangular/PAth/POlar] <Rectangular>: <ENTER>
 Select grip to edit array or [ASsociative/Base point/COUnt/Spacing/COlumns/ROws/Levels/eXit]<eXit>: R
 Enter the number of rows or [Expression] <2>: 3
 Specify the distance between rows or [Total/Expression] <0.7500>: 15
 Type = Rectangular Associative = Yes
 Select grip to edit array or [ASsociative/Base point/COUnt/Spacing/COlumns/ROws/Levels/eXit]<eXit>: COL
 Enter the number of columns or [Expression] <4>: 2
 Specify the distance between columns or [Total/Expression] <0.7500>: 25
 Specify the incrementing elevation between rows or [Expresson] <0.0000>:<ENTER>
 Select grip to edit array or [ASsociative/Base point/COUnt/Spacing/COlumns/ROws/Levels/eXit]<eXit>: <ENTER>

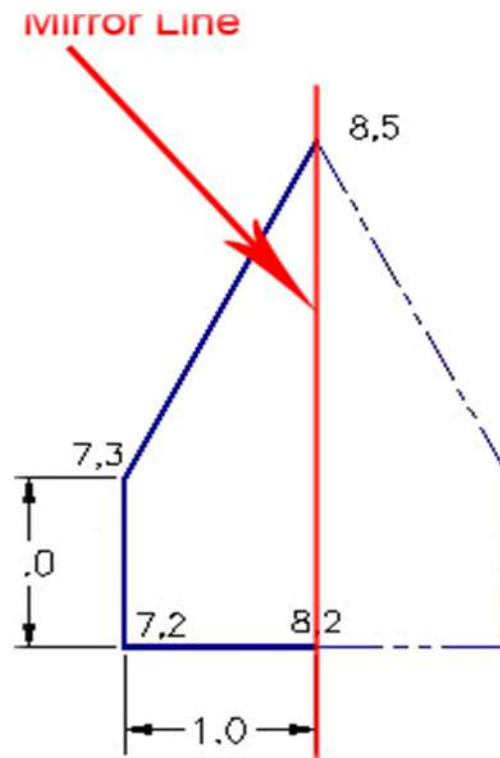


- Polar array-
- Command: ar ARRAY
- Select objects: 1 found
- Select objects:<ENTER>
- Enter array type [Rectangular/PAth/POLar] <Rectangular>: PO
- Type = Polar Associative = Yes
- Specify center point of array or [Base point/Axis of rotation]:C
- Enter number of items or [Angle between/Expression] <4>: 6
- Specify the angle to fill (+=ccw, -=cw) or [EXpression] <360>: 360
- Press Enter to accept or [ASsociative/Base point/Items/Angle between/Fill angle/ROWS/Levels/ROTate items/eXit]<eXit>:



How to use mirror command in AutoCAD?

- Mirror: Creates symmetrical mirror images by flipping objects to a specific axis. The axis on which object is flipped is called a mirror line.
 - Command: MI
 - Command: MI_MIRROR
- Select objects: <SELECT THE 3 LINES> Specify opposite corner: 3 found
 Select objects: <ENTER>
 Specify first point of mirror line: <SELECT 8,5>
 Specify second point of mirror line: <SELECT 8,2>
 Erase source objects? [Yes/No] <N>: <ENTER>



How to use stretch command in AutoCAD?

- Stretch:-Objects that are partially enclosed by a crossing window are stretched.
- Keyboard Command: S
- Command:S_STRETCH

Select objects to stretch by crossing-window or crossing-polygon...

Select objects: (pick first point of crossing window)

Specify opposite corner: (pick second point of window)

Select objects: (to end selection)

Specify base point or displacement: (pick base point)

Specify second point of displacement: (pick second point)

