

CAD FOR GEOMATICS

GEOM 111

Introduction

What is CAD

CAD, also known as computer-aided design and drafting (CADD), is the use of computer technology for the process of design and design-documentation. CAD software is a type of computer program that replaces tedious manual drafting with an automated process. If you work in architecture, mechanical or engineering fields you have probably worked with CAD software. This software can help you explore design ideas, visualize concepts through animations and photorealistic renderings, and simulate how a design will perform in the real world. AutoCAD® software was the first and most widely used CAD software.

What is AUTOCAD

AutoCAD is a computer-aided design (CAD) program used for 2-D and 3-D design and drafting. AutoCAD is developed and marketed by Autodesk Inc. and was one of the initial CAD programs that could be executed on personal computers.

Autocad was first released in December 1982 having been purchased a year prior in its original form by Autodesk founder John Walker.

Chapter1

AUTOCAD 2015 INTERFACE

Lesson1

Autocad 2015: environment

Application
Menu

Quick access toolbar

Info center

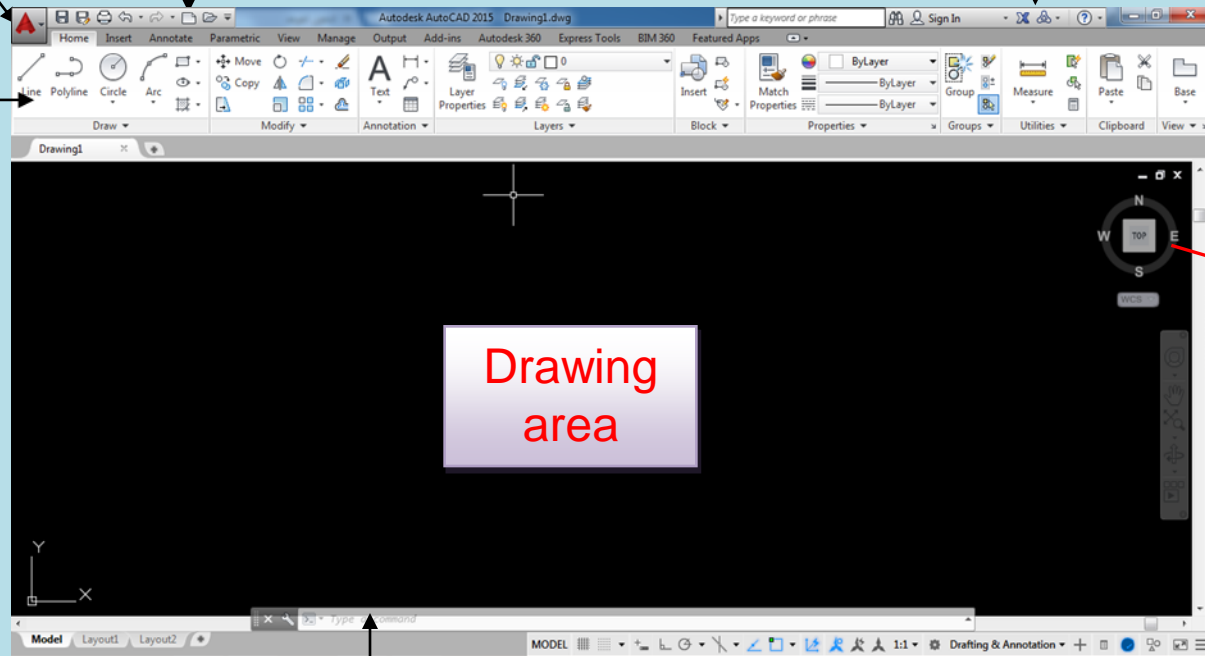
Ribbon

ViewCube

Drawing
area

Command line area

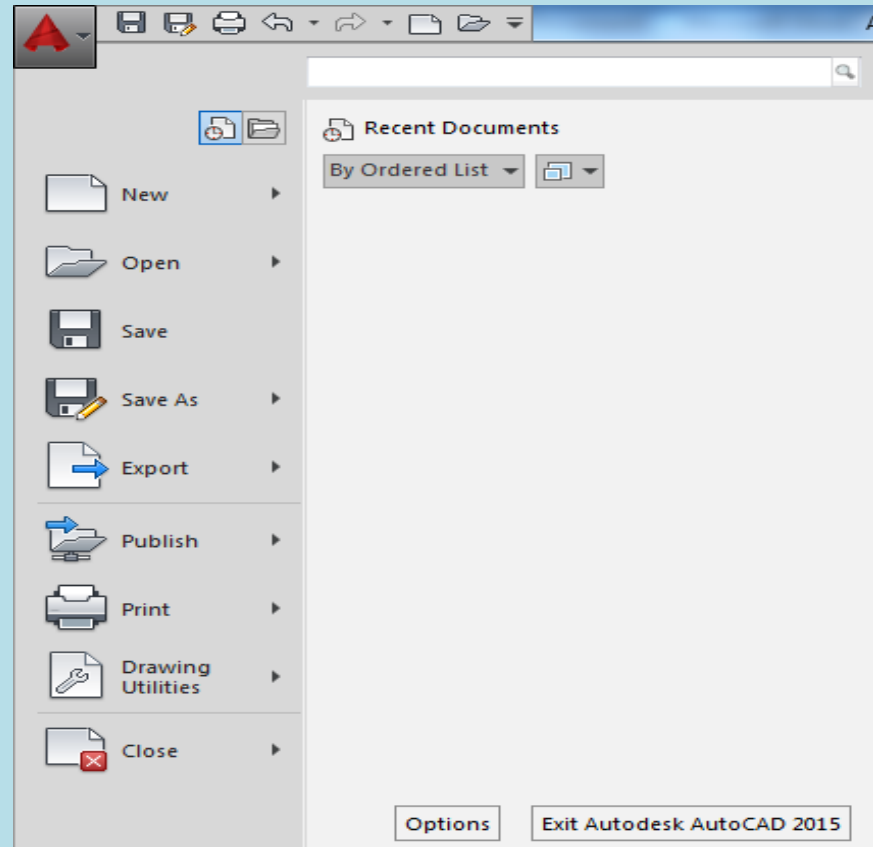
Status bar



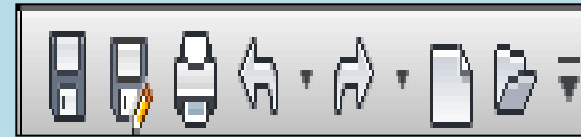
Application Menu



The save as option in the application menu showing the list of save as options



Quick access toolbar



New



Undo



Open



Redo



Save



Print



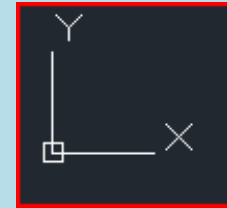
Save as

Info Center



Quickly search for a variety of information sources, access product updates and announcements, and save topics with InfoCenter.

The USC Icon



The low-left corner of drawing area, you see an **L** shaped arrow. This is the User Coordinate System (USC) icon, which tell you your orientation in the drawing.

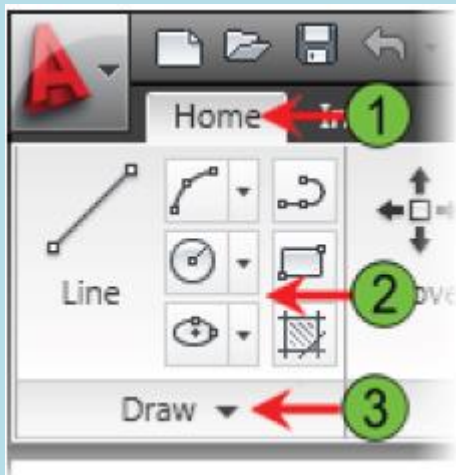
Command line area



The command window is normally located at the bottom of the application window and docked between the drawing area and status bar. Whether you enter a command manually at the command line or click a command tool on a toolbar, all commands are passed through and evaluated by the command line.

At each stage of the command process, the software either provides you with a series of options to choose from or requires that you input values relevant to that stage of the process.

The Ribbon



Tabs

- 1 Identifies the purpose and name of the control panel.

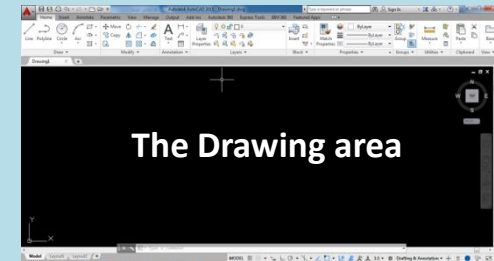
Panels

- 2 Contains groups of related tools associated with the selected tools.

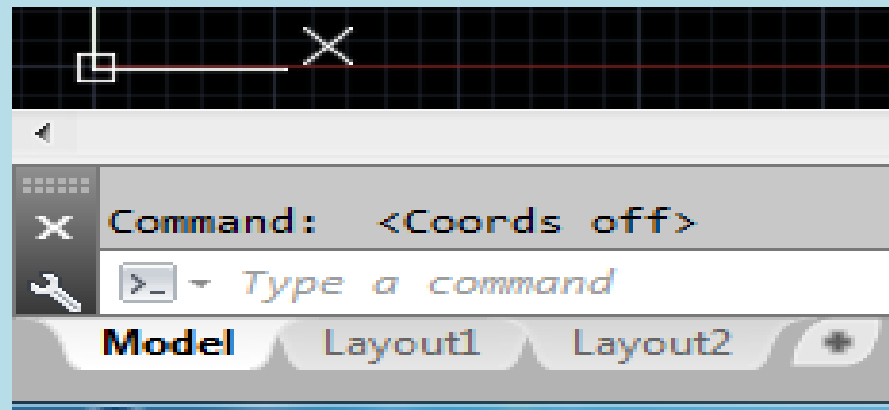
More tools

- 3 Click and hold the down arrow to display more tools and options in the selected panel.

The Drawing area



The drawing area is a rectangular area used to create a new drawing or modify a previously created drawing. Although the drawing area is unlimited, the size of the computer . The drawing area is classified as a model space or layout. The user can switch to the model space or layout tabs, respectively, from the lower left corner of the drawing area.



The ViewCube



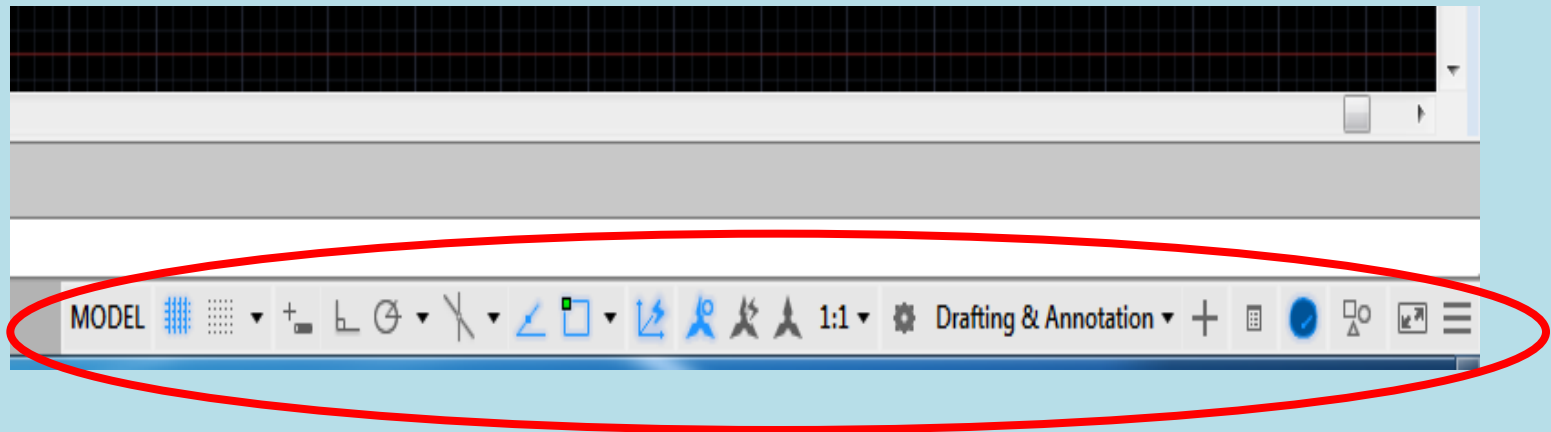
The ViewCube is a navigational tool. By default, it is located in the upper right corner of the drawing area .

The ViewCube can be turn *on/off* by using of the following procedures:

- ✚ Panel method: from the **view tab** and **viewport** tools panel click **ViewCube** tool.
- ✚ Command line method: type **navvcube** or **NAVVCUBE** in the command line and press the enter key. The option list will appear.

The status bar

The set of command below the command line or on the lower of the corner of the interface are collectively known as a status bar.



Chapter2

DRAWING AIDS

Lesson1

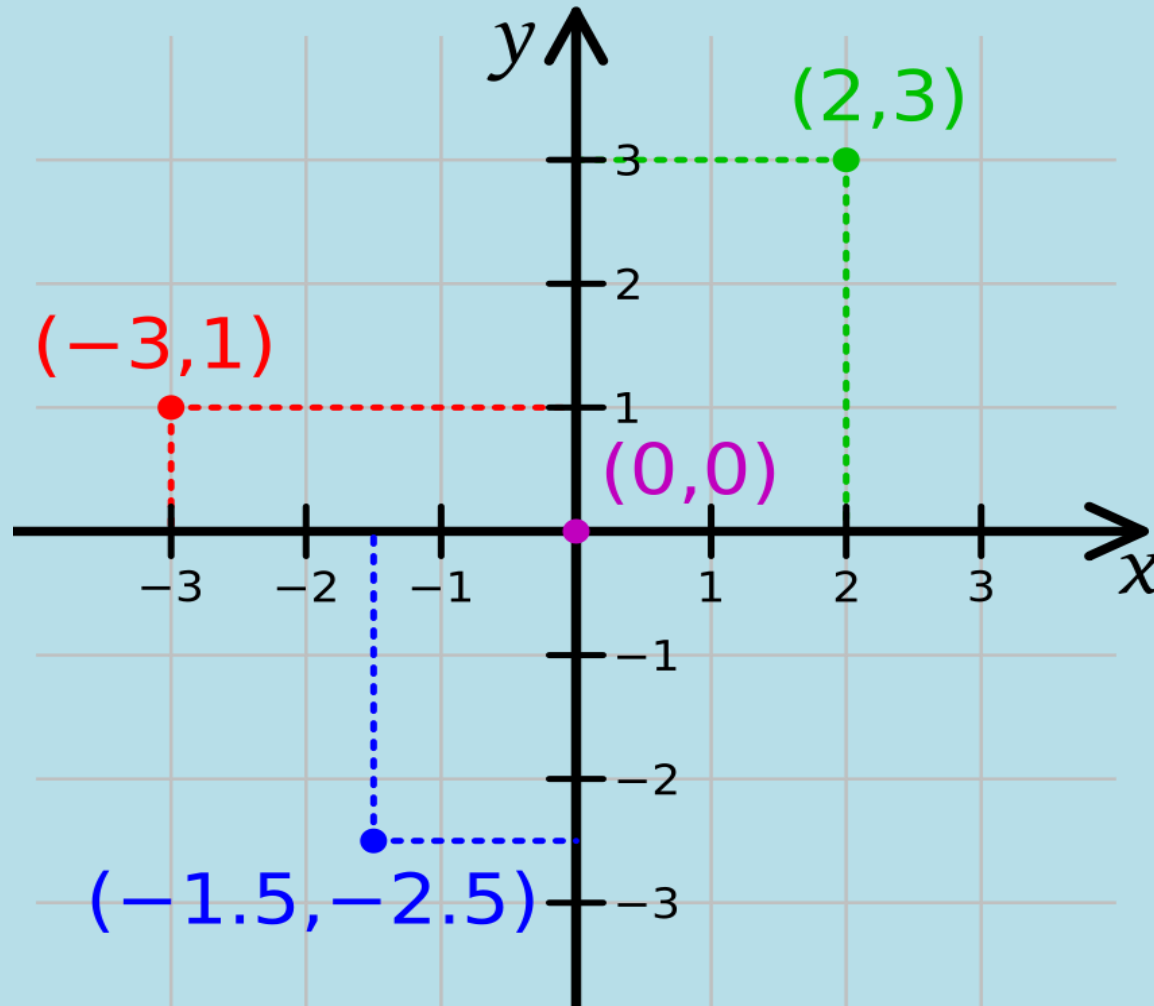
The Cartesian coordinate system

Is a system which uses to determine the position of point or other geometric element.

The scheme gets its name from one of the first people known to have used it, the French mathematician and philosopher R. DECARTES.

The Cartesian plane consists of two perpendicular axes that cross at a central point called the origin. Positions or coordinates are determined according to the east/west and north/south displacements from the origin. The east/west axis is often called the x axis, and the north/south axis is called the y axis. For this reason, the Cartesian plane is also known as the xy -plane

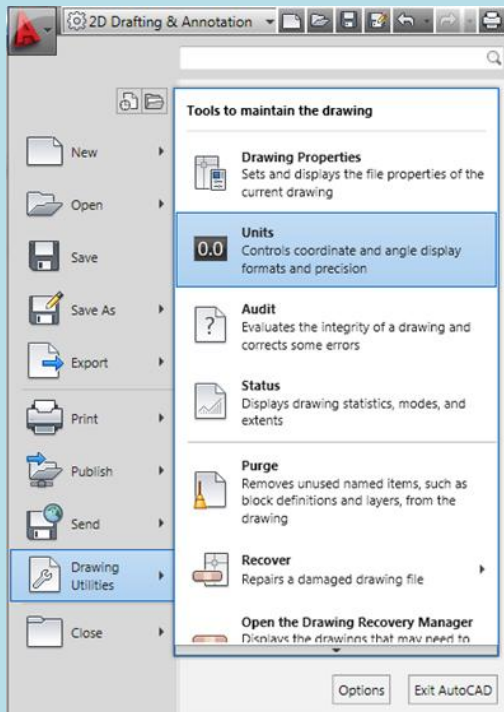
Illustration of a Cartesian coordinate system



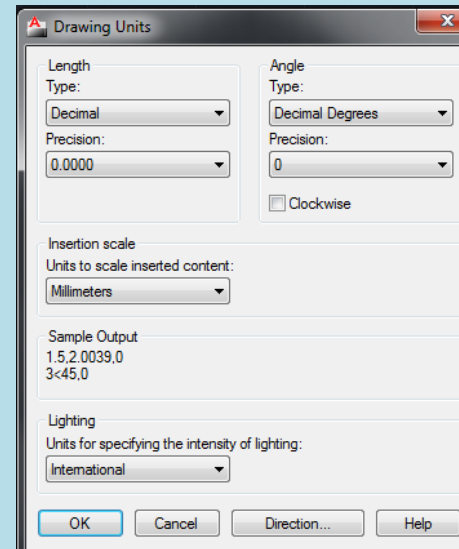
Lesson2

Drawing units

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



Application menu → **drawing unities** → **Units**



Command line area



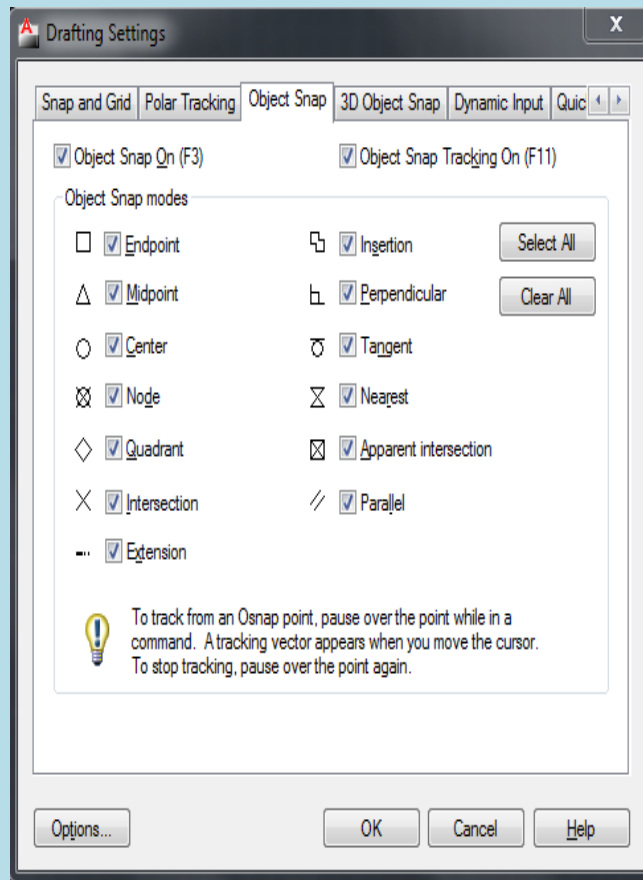
Type UN



Press enter

Lesson3

Object Snap



At the command line area: type **OS** and press **enter**

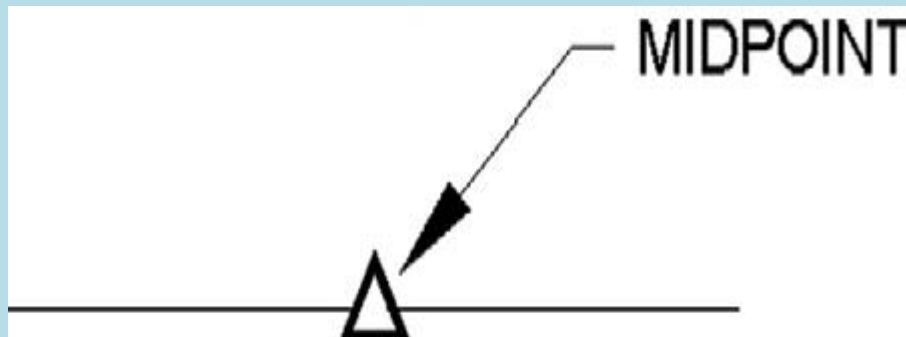
An object snap mode specifies a snap point at an exact location on an object. OSNAP specifies running object snap modes, which remain active until you turn them off

Object Snap

ENDPOINT

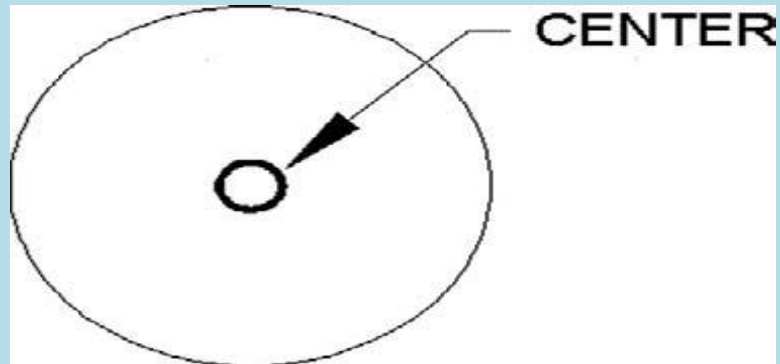


MIDPOINT

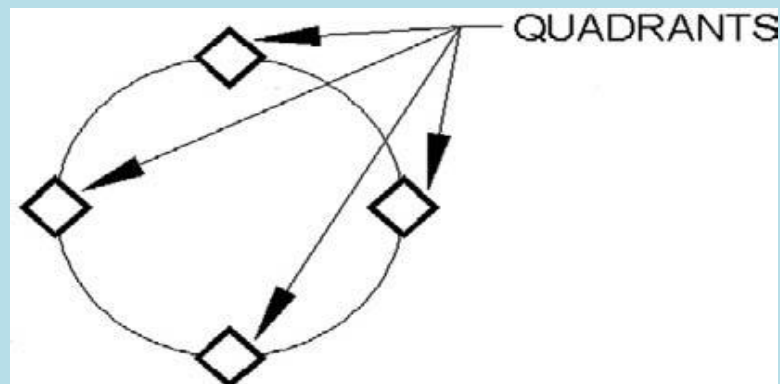


Object Snap

CENTER

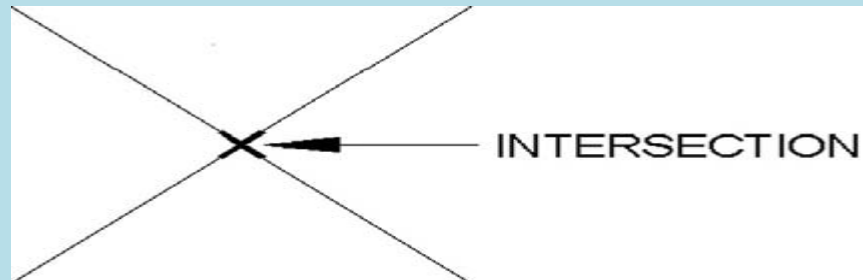


QUADRANT

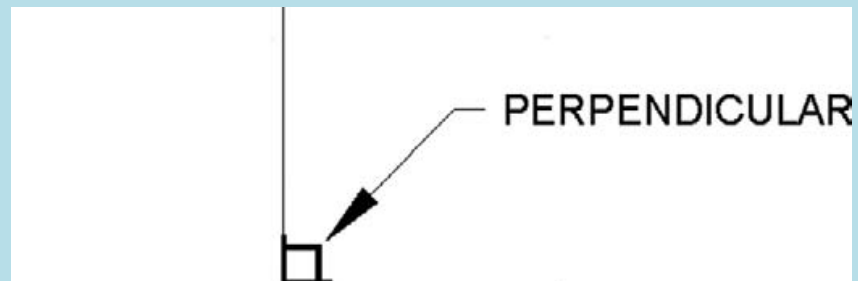


Object Snap

INTERSECTION



PERPENDICULAR



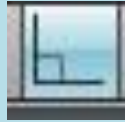
Lesson4

Ortho Mode

At the command line area: type **Ortho** and press **enter**

Key board : **press F8**

Status Bar



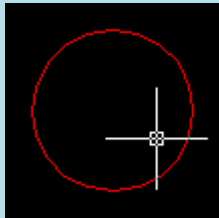
Ortho is short for orthogonal, which means either vertical or horizontal. Like the other options on the status bar, Ortho is not really a command, it is a drawing mode which can either be turned on or off. Ortho mode can be toggled on or off in one of three ways. The quickest way is just to click on the ORTHO button on the status bar. The appearance of the button tells you whether Ortho is currently turned on or turned off. When Ortho is turned on, the ORTHO button appears pressed in. You can see how this appears by looking at the status bar illustration below. In the illustration, Ortho is turned on but Grid and Snap are turned off.

Lesson5

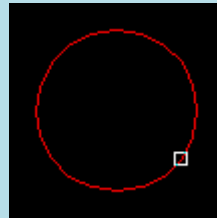
Selecting objects

-1- By picking

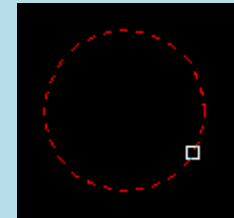
The most obvious way to select an object in AutoCAD is simply to pick it. Those of you who have used other graphics based utilities will be familiar with this concept. Generally all you have to do is place your cursor over an object, click the mouse button and the object will be selected.



The Crosshairs



The Pickbox



Highlighted Object

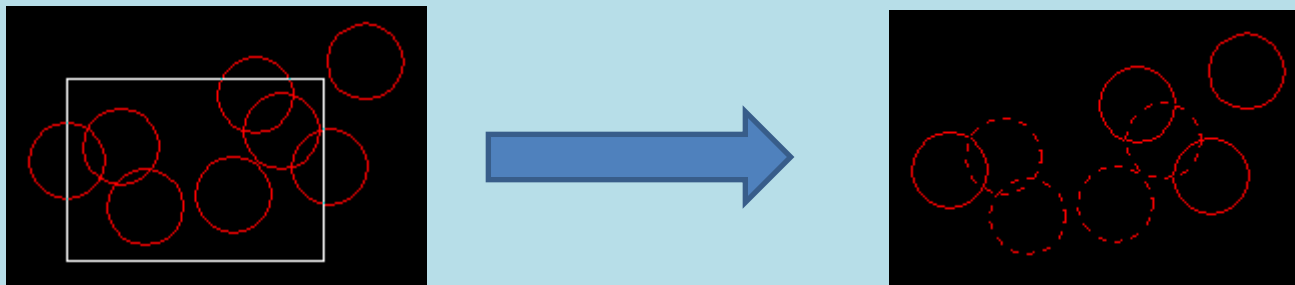
Select object

-2- Window Selection

At the command line area: 1 type **select** and press **enter**

2 Type **W** and press **enter**

Window allows you to define a rectangle using two points in exactly the same way as the RECTANGLE command. Once the window is defined, all objects which lie entirely within the window will be selected.



Selecting objects

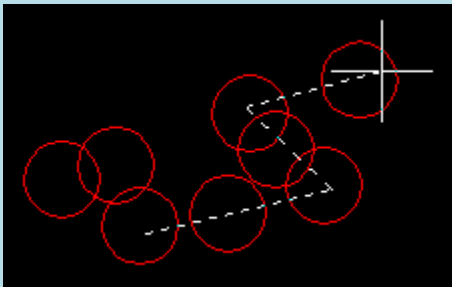
-3-All objects

keyboard: press Ctrl+A

-4-Fence selection

At the command line area type:

- 1 **select** and press **enter**
- 2 **F** and press **enter**



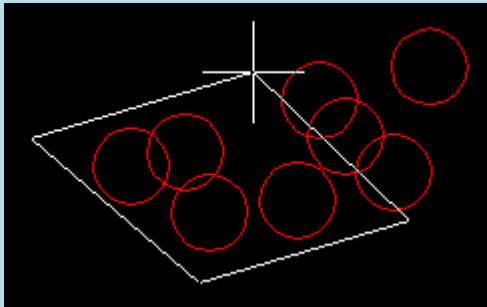
The Fence option allows you to draw a multi-segment line, like a Polyline. All objects which cross the fence will be selected.

Select object

-5- Window polygon selection

At the command line area: 1 type **select** and press **enter**

2 Type **WP** and press **enter**



The Window Polygon option, invoked by typing WP is similar to the Window option except that you can define an irregular polygon shape within which objects will be selected. As with the Window option, only objects which fall entirely within the polygon will be selected.

Select object

-6- Using previous selection

At the command line area: **1** type **select** and press **enter**

2 type **P** and press **enter**

AutoCAD always remembers the last selection set you defined. This is very useful because you may need to make a number of changes using different commands to the same group of objects. In order to re-select the last selection set you can use the Previous option. The previous option is invoked by typing P at the "Select objects" prompt.

Select object

-7- Selecting the last object

At the command line area: 1 type **select** and press **enter**

2 type **L** and press **enter**

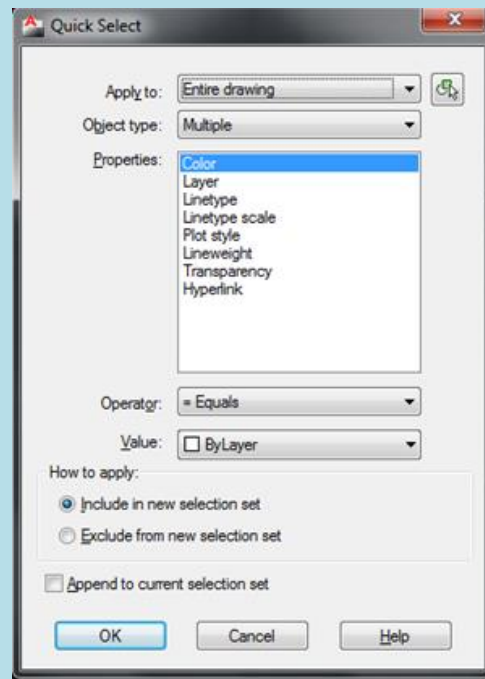
You can select the last object created by entering L at the "Select objects" prompt.

Select object

-8- Quick Select

At the command line area:

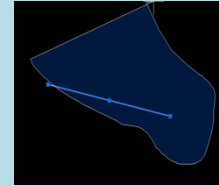
type **qselect** and press **enter**



Quick Select made its debut in AutoCAD 2000 and is designed to help users make complex selections quickly. It allows you to make selections based upon object properties.

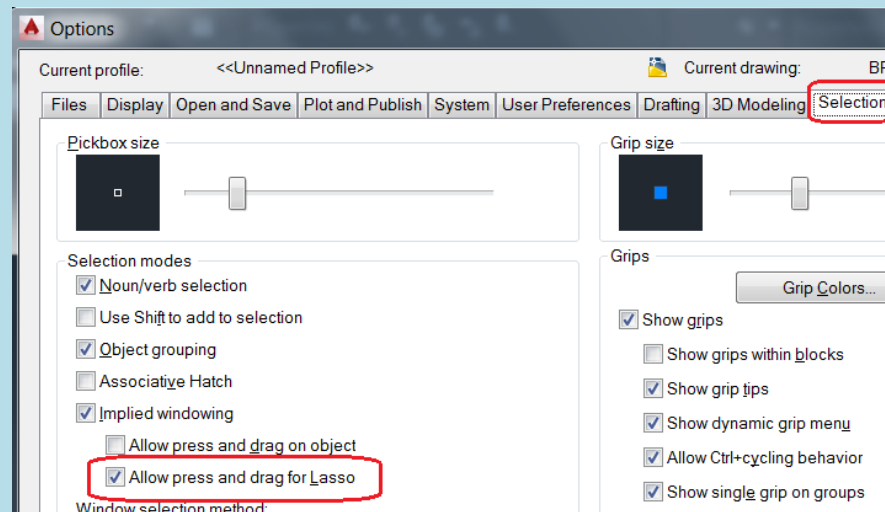
Select object

-9- To turn off the lasso selection feature in AutoCAD



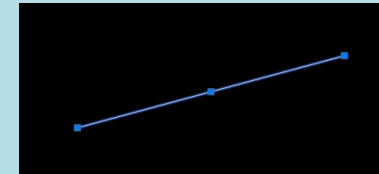
At the command line area: type **op** or **option** and press **enter**

In the Options, go to the Selection tab and uncheck the lasso feature, shown in the screens hot below.

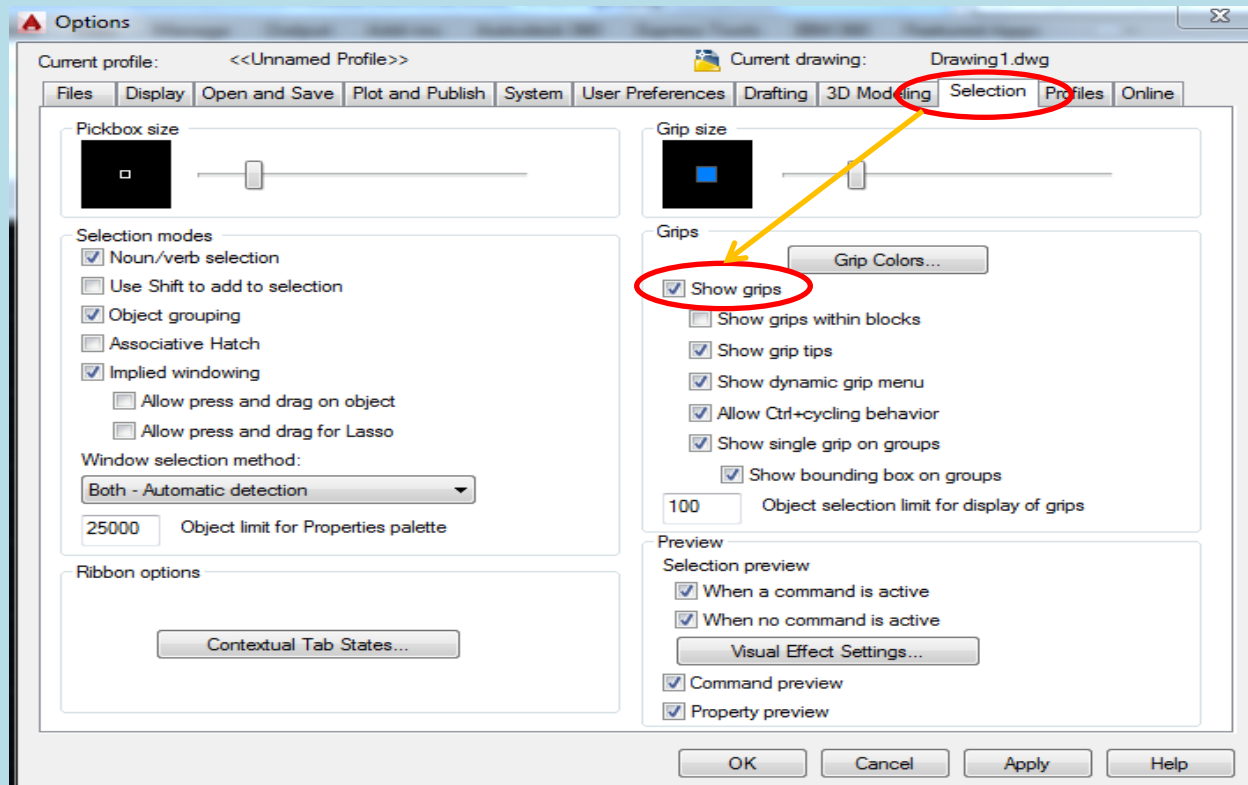


Select object

-10- To display grips when you select object in AutoCAD

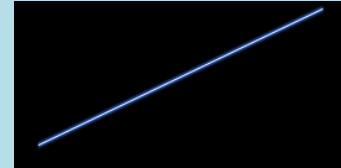


At the command line area: type **op** or **option** and press **enter**

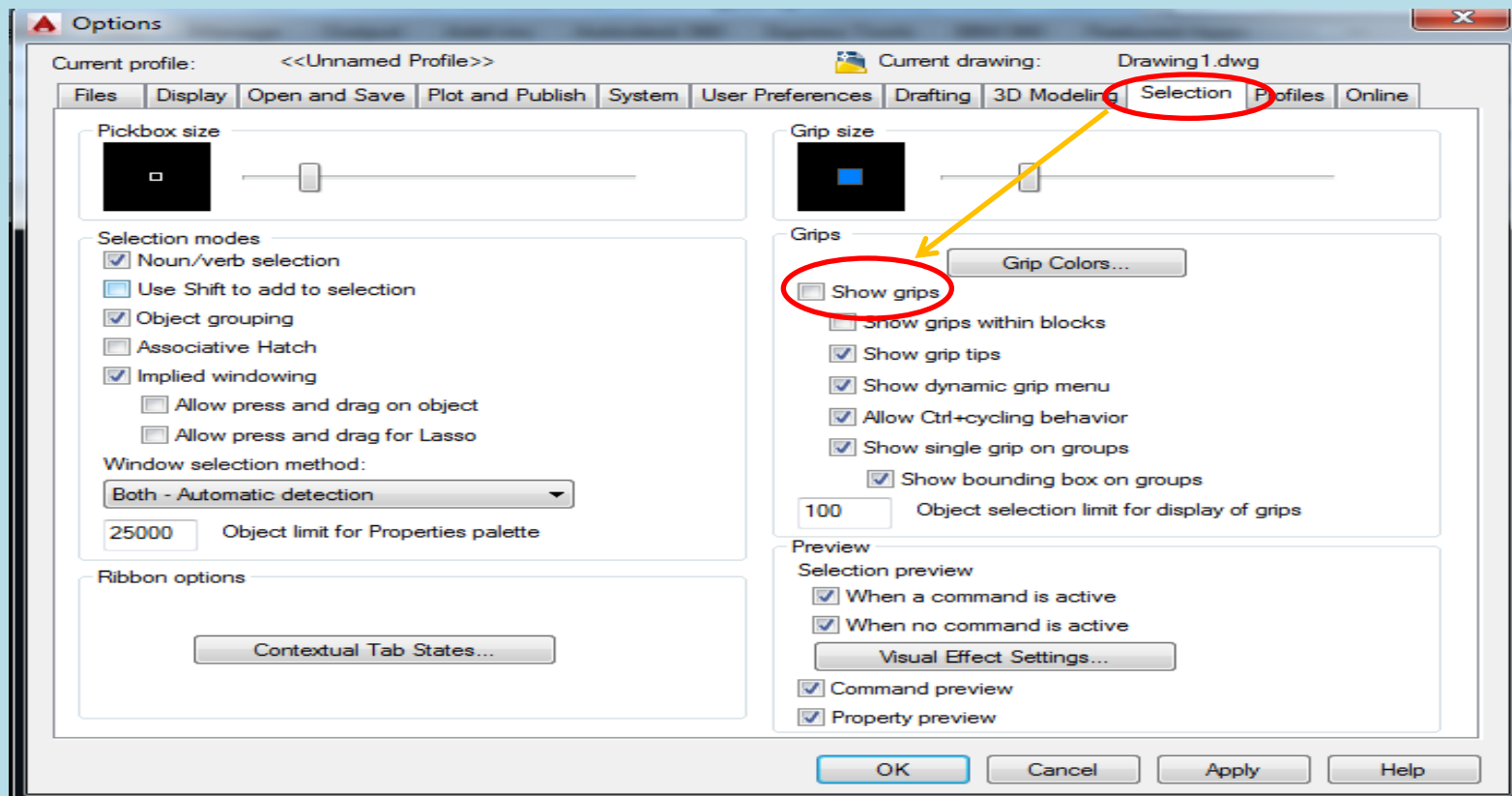


Select object

-11- To hide grips when you select object in AutoCAD



At the command line area: type **op** or **option** and press **enter**



Lesson6

Zoom command

-1- Zoom All



At the command line area:

type **Z** and press **enter**



type **A** and press **enter**

This option causes AutoCAD to display the whole drawing as far as its drawing limits or drawing extents (whichever is the greater of the two).

Zoom command

-2- Zoom extents



At the command line area:

type **Z** and press **enter**



type **E** and press **enter**

This option will display all the graphics that are contained in the drawing (referred to as the drawing extents) with the largest image possible.

Zoom command



-3- Zoom Previous

At the command line area:

type **Z** and press **enter**



type **P** and press **enter**

This option restores the displayed view prior to the current one. For the purpose of this option, up to 10 views are saved so that the last ten views can be recalled. This option includes every time you use the scroll bar, which is one reason to avoid the scroll bars for panning a lot in your drawing.

Zoom command



-4- Zoom window

At the command line area:

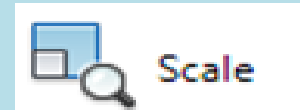
type **Z** and press **enter**



type **W** and press **enter**

This option (also a 'hidden' default) prompts the user to pick two corners of a box on the existing view in order to enlarge that area to fill the display

Zoom command



-5- Zoom Scale

At the command line area:

type **Z** and press **enter**



type **S** and press **enter**

This is a 'hidden' default option. You do not have to type "S" to choose this option. It simply requires the entry of a number that represents a magnification factor.

Lesson7

Drawing Limits

The drawing limits are two-dimensional points in the World Coordinate System that represent a lower-left limit and an upper-right limit. The drawing limits also govern the portion of the drawing covered by the visible grid and determine the minimum area a ZOOM All displays.

Choose Format, Drawing Limits.

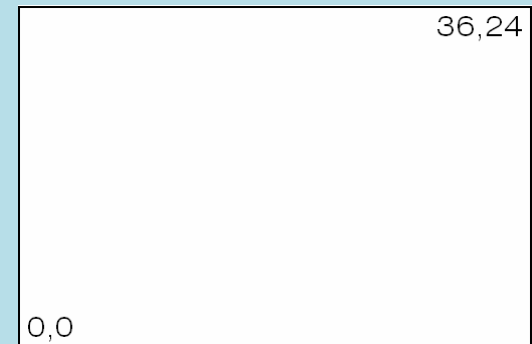
Or

2. **Type** LIMITS at the command prompt
Command: **LIMITS**

3. **Type** One of the following options
On/Off/Lower left corner **0,0**

4. **Type** One of the following options for the upper right limit:
Upper right corner :**36,24**

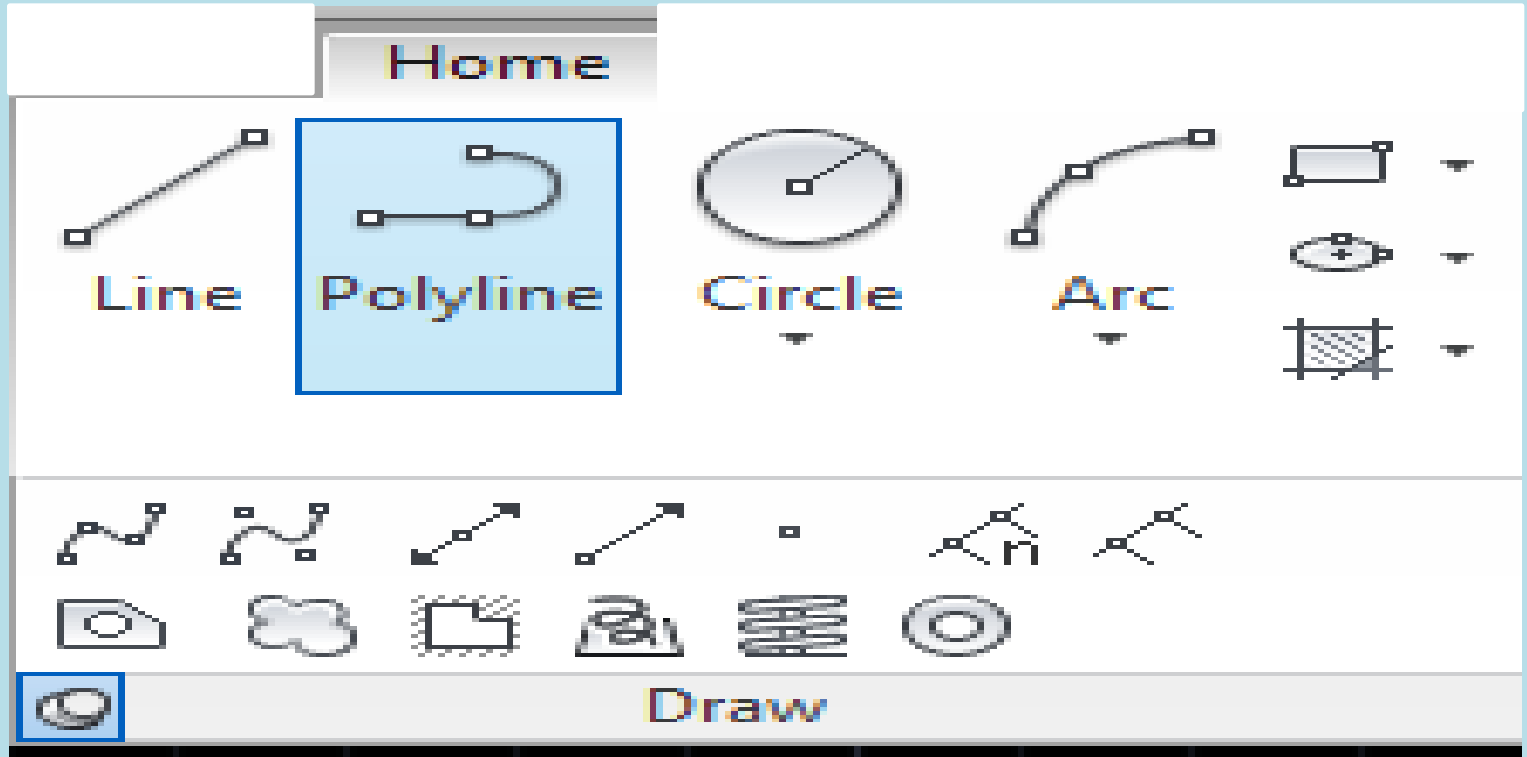
Drawing with lower left limit of 0,0 and upper right limit of 36,24



Chapter3

DRAWING OBJECTS

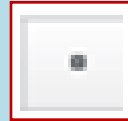
Draw commands



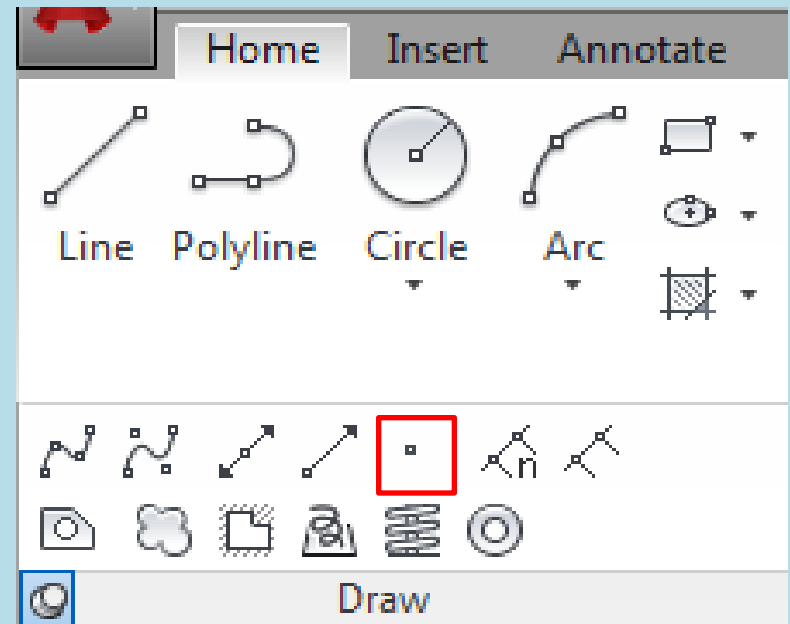
Lesson1

To draw a Point

→ Ribbon/draw



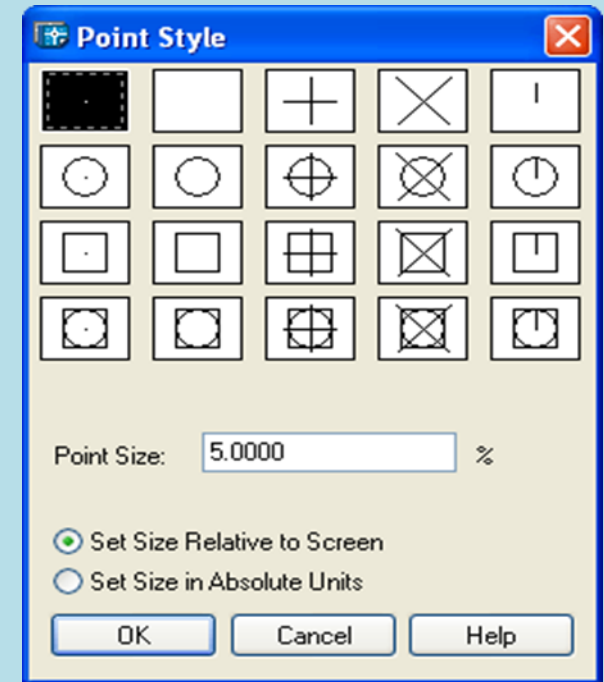
→ At the command line area type **PO**



Point properties

Point style

Command line area type: **ptype** and press enter

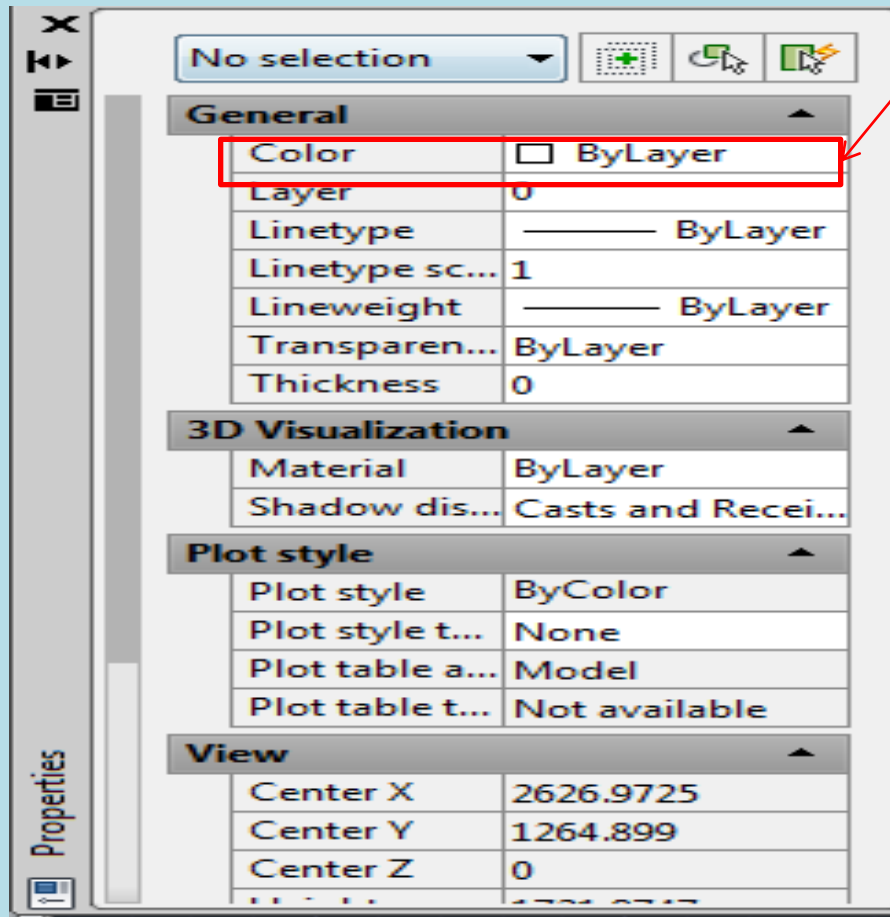


At the command type **pdsi** and press enter

Point color

At the command type **pr** and press enter

You can change the
point color

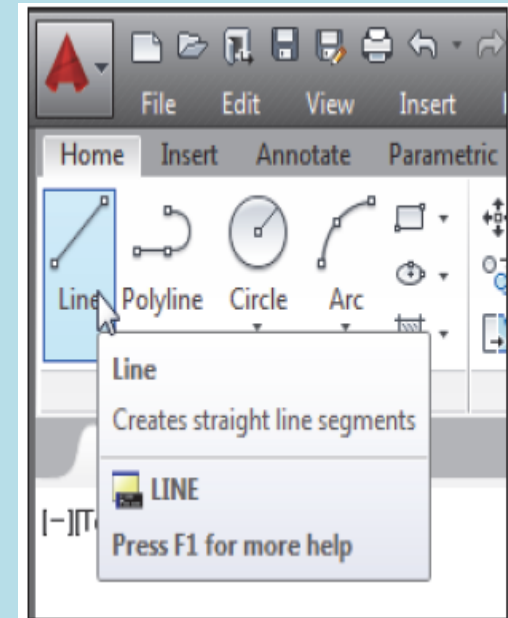


Lesson2

To draw a line

❖ Ribbon: home → draw → Line

❖ At the command line area type :L



□ To draw a line

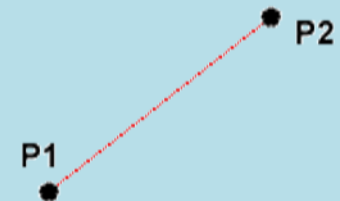
Specify first point



Specify second point

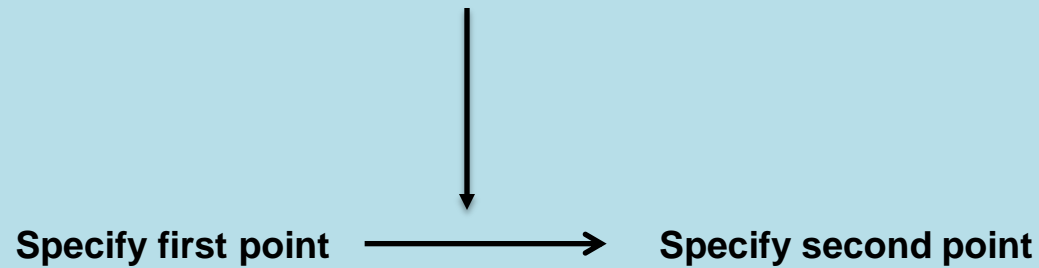
Where you're finished drawing segments. End with one of these steps:

- Press enter, or right-click anywhere in the drawing area and choose enter.
- Type C and press enter

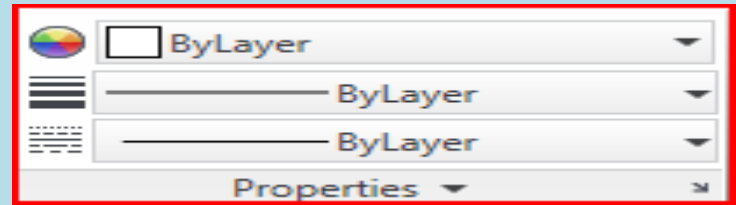


To measure a distance

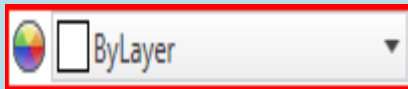
Keyboard: type **DI** end press enter



To modify a line



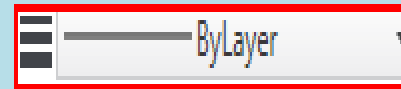
1



Color control



line type control



line weight control

2

Select a line



double click



modify

3

Command line area

Type **PR**

Press enter

To join lines

At the command line area type pedit



Select line, press ENTER to convert the selected line.



Enter **j** (join)



Select one more lines that are located end to end. Each selected is now joined into single line.



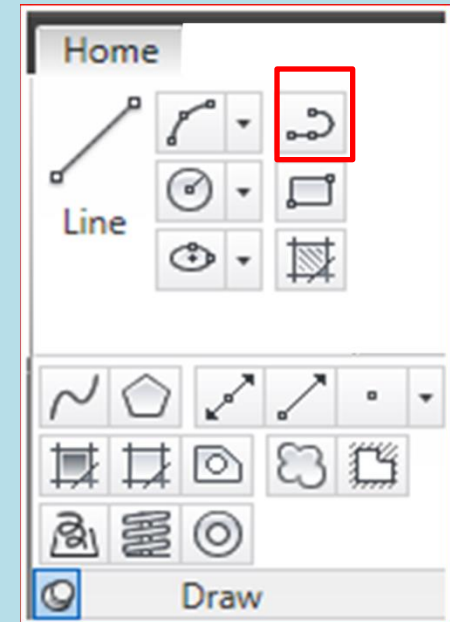
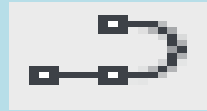
Press **ENTER** to end the command

Lesson3

To draw a polyline

2

Draw/polyline



3

At the command line area, type: **PL**

Procedure

- Specify the first point of the polyline
- Specify the second point of the polyline
- Specify the end point of the polyline
- press enter to end

Enter **C** to close the polyline

To modify a polyline (width)

Specify the first point of the polyline



At the command line area enter **w**



Press enter



Specify starting width



Specify ending width

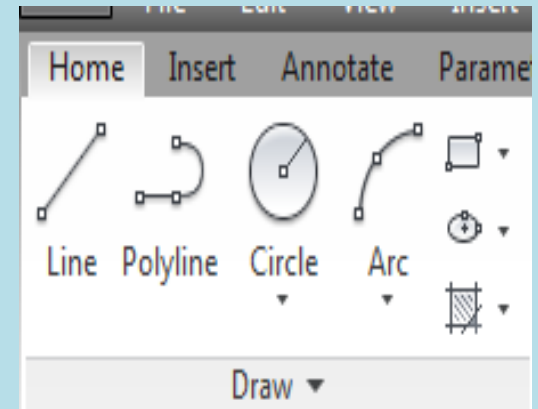


Press enter

Lesson4

To create a rectangle

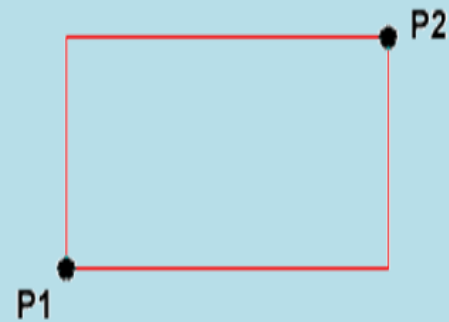
■ Ribbon/draw



■ Command line area: TYPE REC

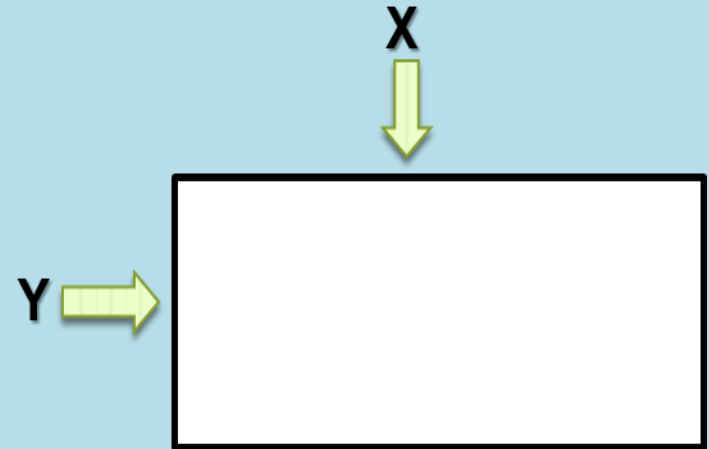
Procedure

- Specify first corner point.
- Specify other corner point.



To specify the dimensions (x, y)

@ X,Y



Procedure

- 1- Specify first corner point
- 2- At the command line area type @ X,Y

To calculate the area

draw



properties

list



Select object



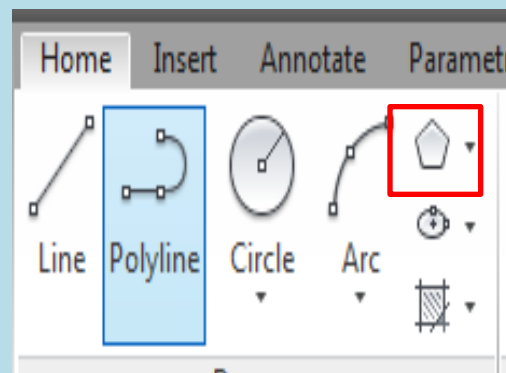
Enter

■ At the command type **li** → select object → enter

Lesson5

To create a POLYGON

Ribbon/draw



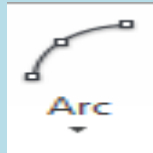
Command line area: TYPE POL

Procedure

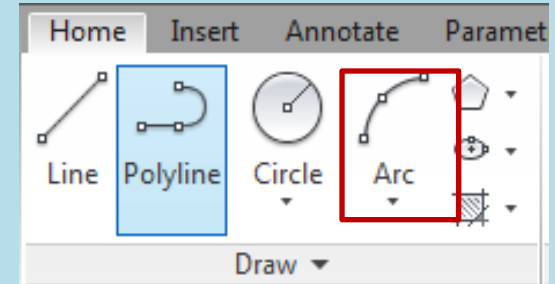
- 1- Enter number of sides
- 2- Enter
- 3- Specify center of polygon
- 4- Enter
- 5- Specify radius of circle
- 6- Enter

Lesson6

To create an arc



Ribbon/draw



Command line area:

TYPE A

Procedure

-1- specify start point of arc

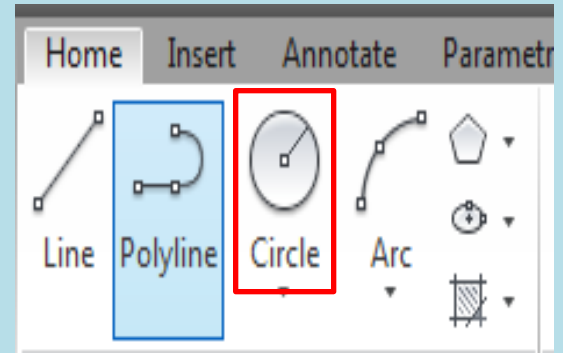
-2- specify second point of arc

-3- Enter

Lesson7

To create a Circle

Ribbon/draw



Command line area:

TYPE C

Procedure

-1- Specify center of circle

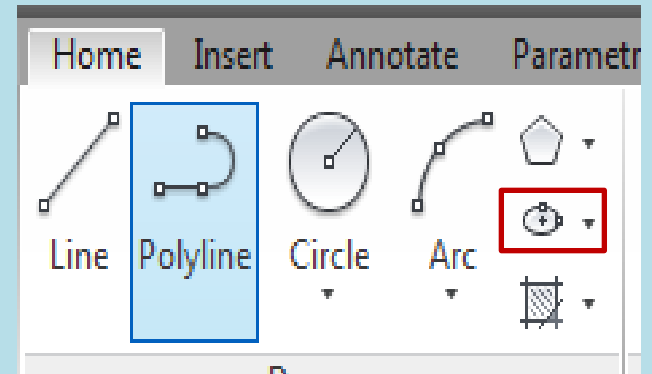
-2- Specify radius of circle

-3- Enter

Lesson8

To create An Ellipse

Ribbon/draw



At the Command line area:

TYPE EL

Procedure

-1- Specify center of ellipse

-2- Specify endpoint of axis

INTRODUCTION

AutoCAD provides different ways to specify coordinate points in your drawing:

- ✚ Direct distance entry
- ✚ Absolute coordinate entry
- ✚ Relative coordinate entry
- ✚ Polar coordinate entry
- ✚ Dynamic input

Direct Distance Entry

Direct distance entry is useful when used with Ortho or Polar Tracking option. It allows you to specify a point relative to the previous one you entered.

Procedure:

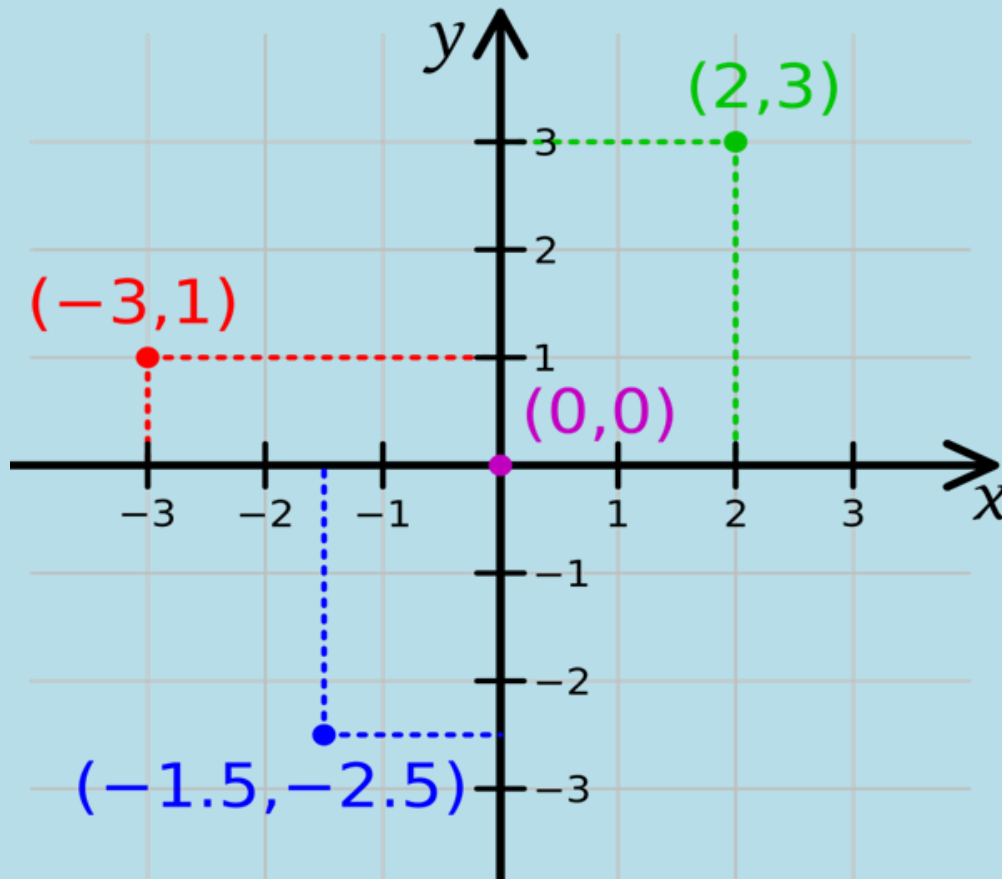
- Move the crosshairs in the direction you want.
- Enter a numeric distance.

Absolute Coordinates

In the absolute coordinate system all points are measured from the origin (0, 0). They are suited to situations where you know the exact X and Y location of the point you want to place. To enter an absolute coordinate use the format X, Y where:

- X is the distance and direction along the horizontal axis from the origin (0, 0).
- Y is the distance and direction along the vertical axis from the origin (0, 0).

Cartesian coordinate system



Relative Coordinates

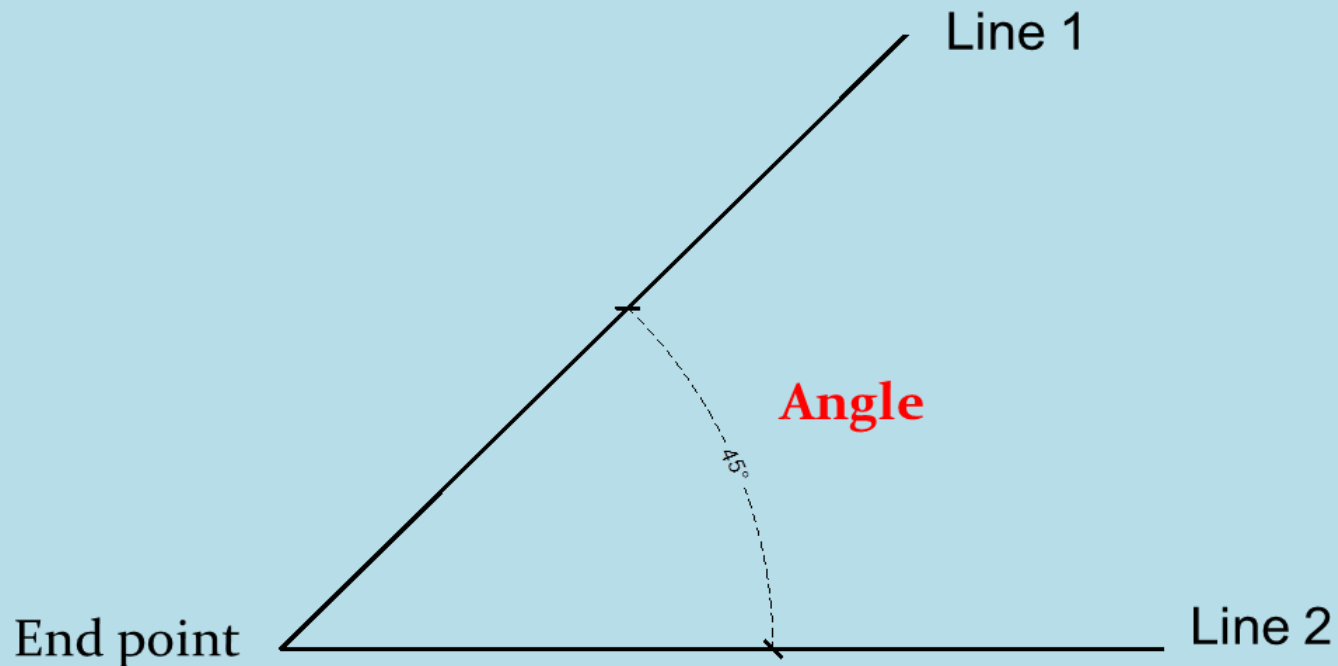
Relative coordinate entries are based on the last point entered. Use a relative coordinate when you know the location of a point in relation to the previous point. To specify a relative coordinate, precede the coordinate with the 'at' @ sign using the format @X, Y:

- @ indicates a relative distance:
- X is the positive or negative distance along the X axis from the previous position.
- Y is the positive or negative distance along the Y axis from the previous position

Polar Coordinate system

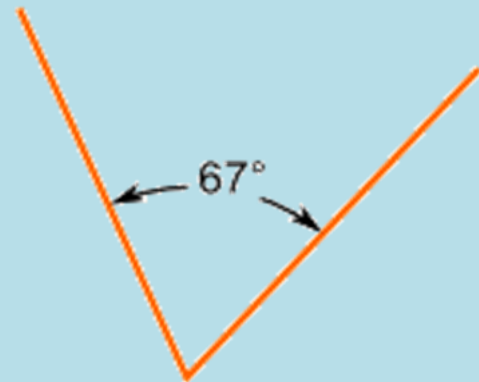
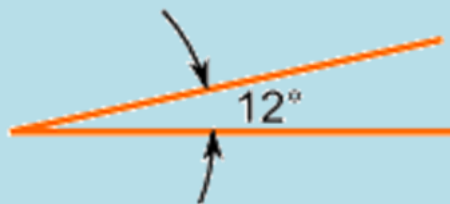
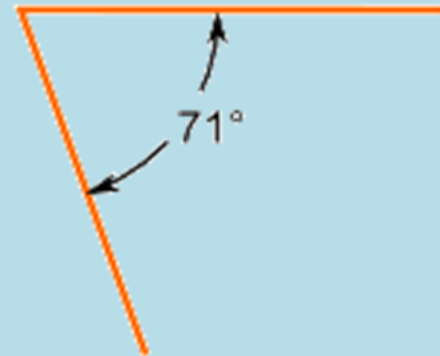
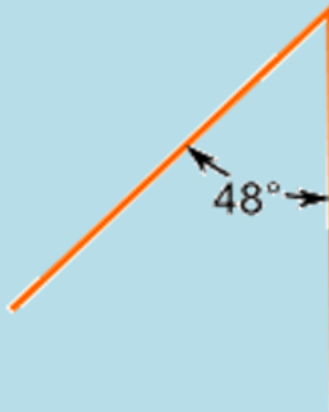
Angle

An angle is a combination of two lines with a common end point.

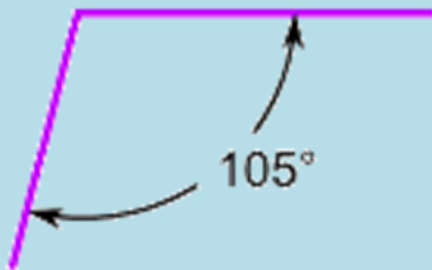
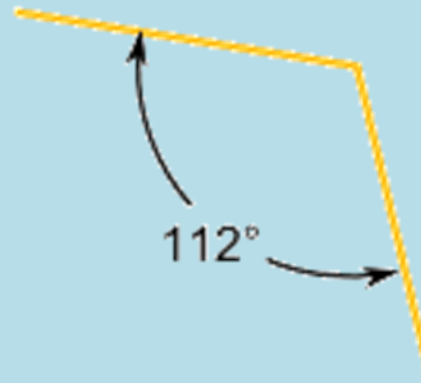


Types of angles

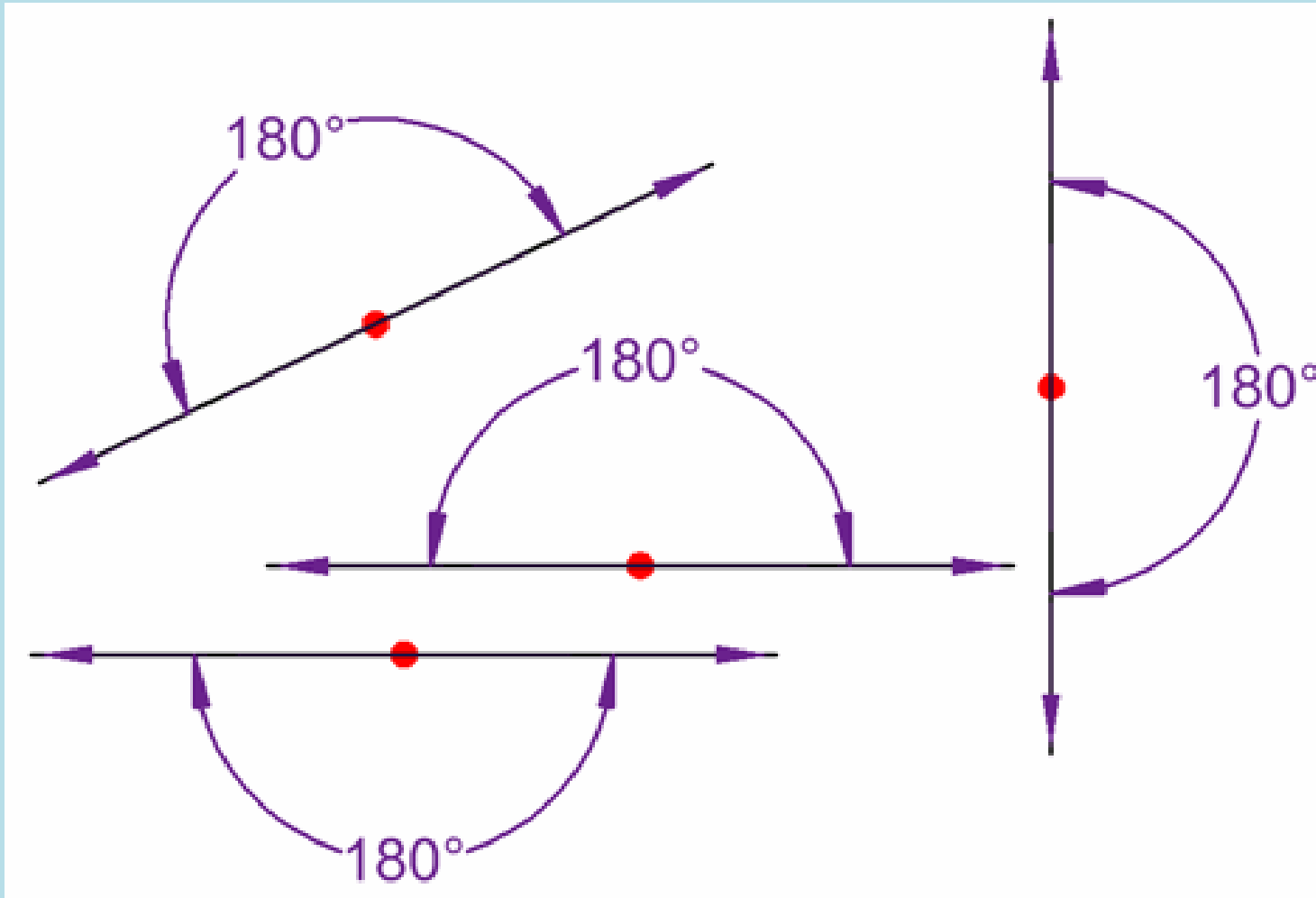
Acute angle



Obtuse angles:

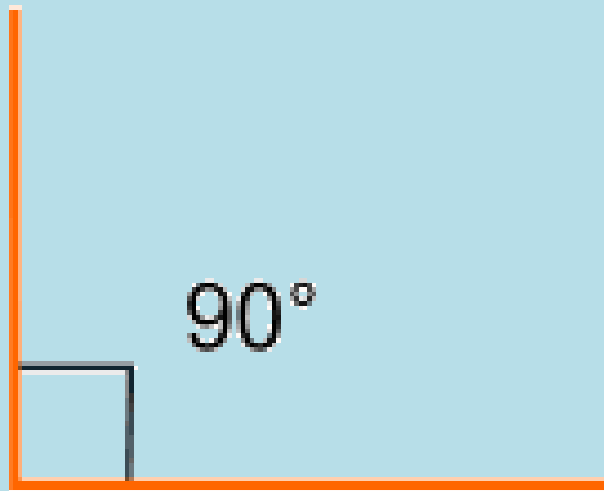


A straight angles



Right angle

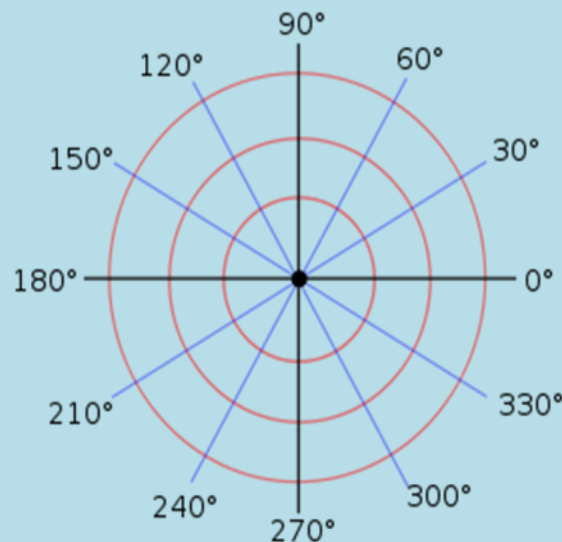
A right angle is an internal angle which is equal to 90°



This is a right angle

Polar coordinate system

- The polar coordinate system is used to draw lines using distance and angle
- Polar coordinates specify a point's exact location by a distance and angle from the last point that was entered. The distance is always positive and the angle is measured from the positive X axis. For example, typing @4<45 tells AutoCAD to locate a point that is four units away from the current location and at an angle of 45 degrees from the horizontal.



A polar grid with several angles labeled in degrees

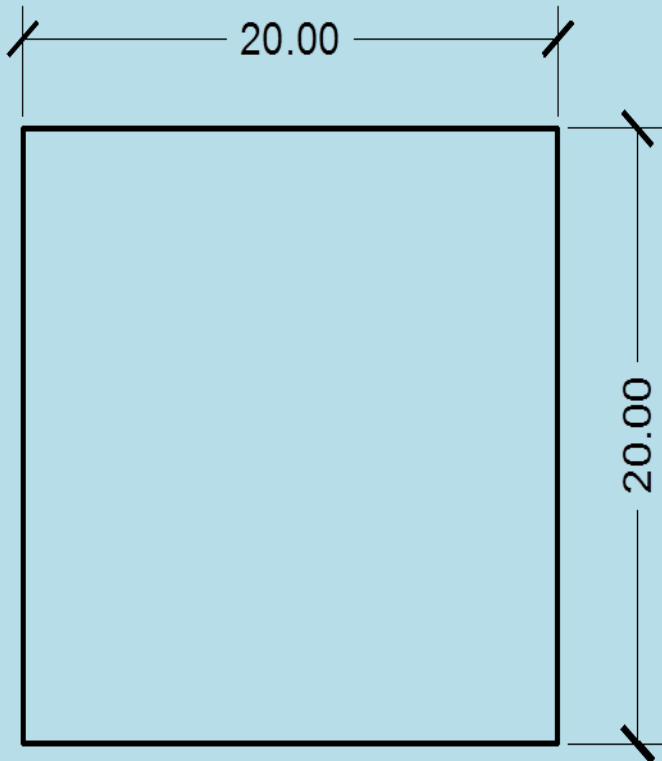
Dynamic Input

Dynamic Input provides a command interface near the cursor to help you keep your focus in the drafting area. When Dynamic Input is on, tooltips display information near the cursor that is dynamically updated as the cursor moves. When a command is active, the tooltips provide a place for user entry.

Turning Dynamic Input ON/OFF:

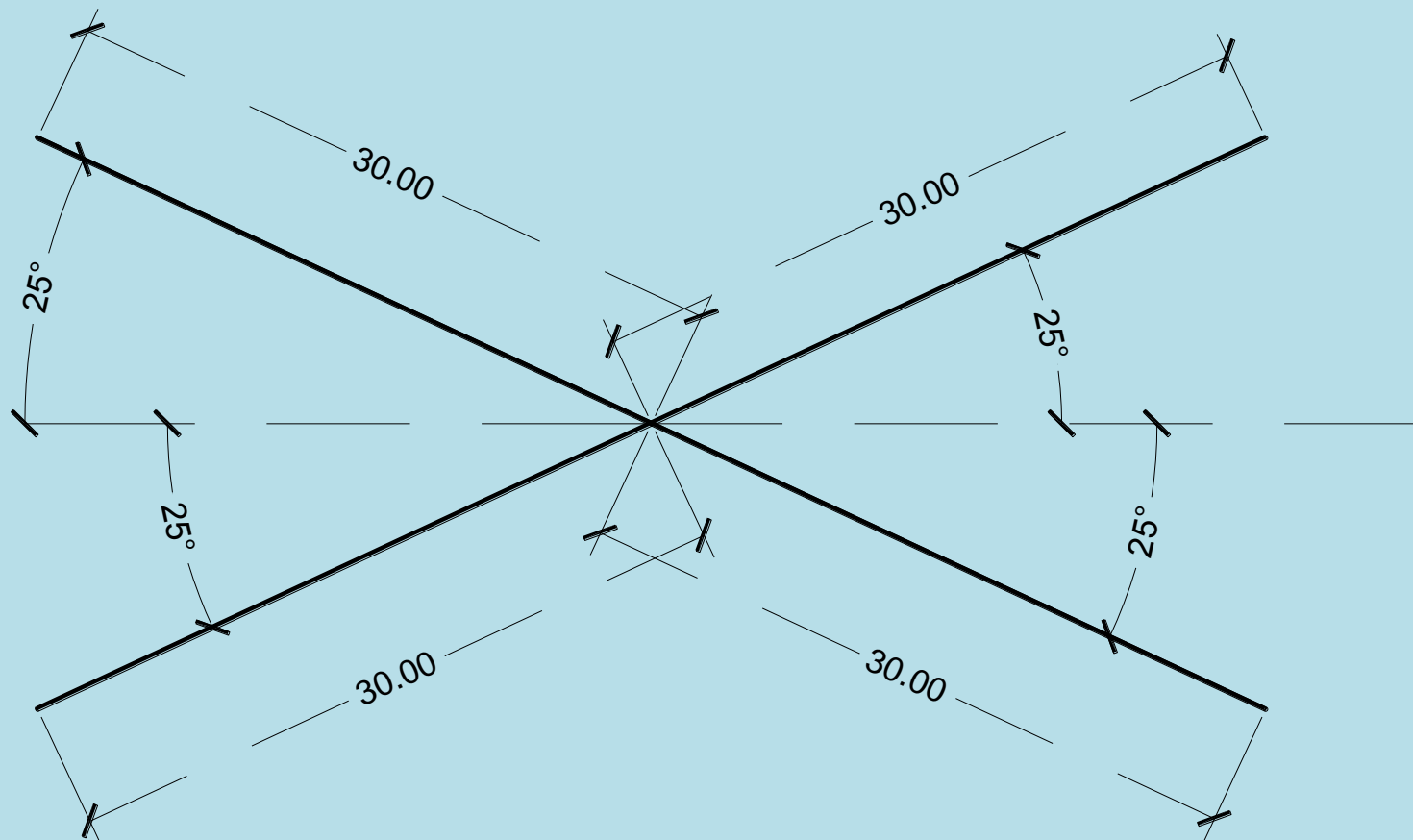
1. Click Dynamic input on the status bar
- or
2. Press F12

Example 1



- ✚ Command: `_line` Specify first point:
- ✚ Specify next point or [Undo]: `@20<0`
- ✚ Specify next point or [Undo]: `@20<90`
- ✚ Specify next point or [Close/Undo]:
`@20<180`
- ✚ Specify next point or [Close/Undo]:
`@20<270`

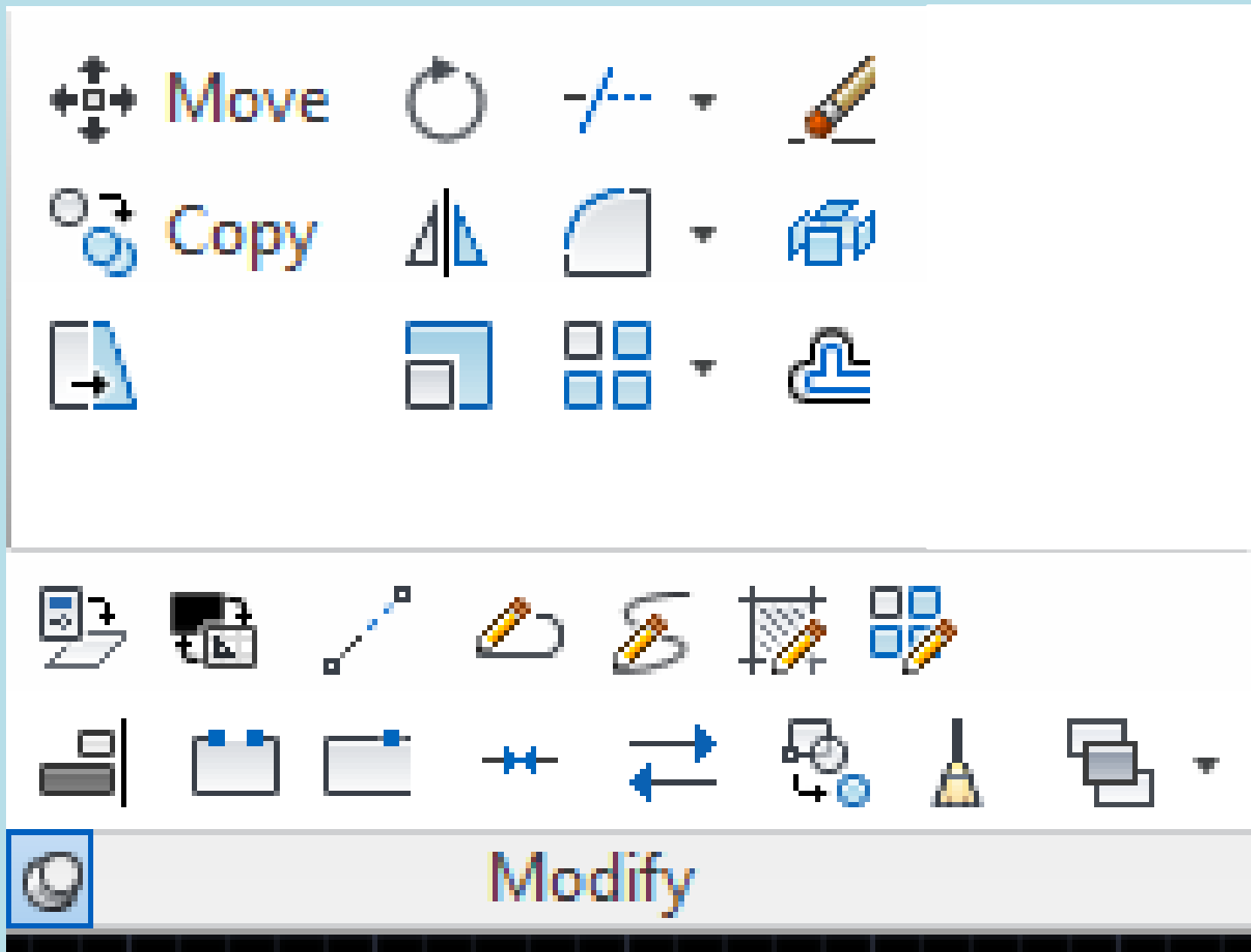
Example 2



Chapter4

EDIT OBJECTS

☐ Modify commands



Lesson1

To Erase object



Ribbon/modify: Erase

Key board:

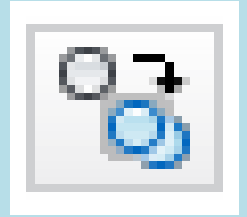
Select object → Delete

Command line area:

TYPE E → Enter

Lesson2

Copy object



Ribbon/Modify:



Command line area:

TYPE CO → enter

Procedure

Select object → Enter



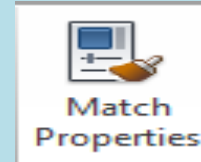
Specify base point → Enter



Specify second point left → click → Enter

Lesson3

Match properties



Ribbon/clipboard:

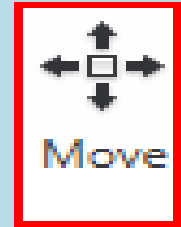
At the Command : TYPE Ma → enter

Procedure

- 1- Select source object
- 2- Select destination object
- 3- Enter

Lesson4

To move object



Ribbon/Modify:



At the Command : TYPE M  enter

Procedure

Select object → Enter



Specify base point → Enter



Specify second point → specify second point → enter


Lesson5

Trim



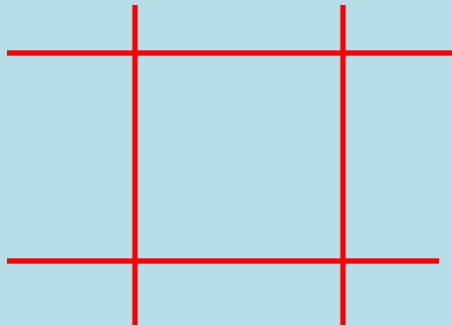
Ribbon/Modify:



At the Command line area: TYPE **TR**  enter

Procedure

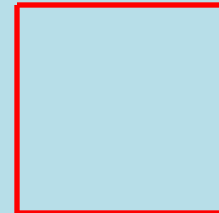
Click trim  Enter  select object to trim



Trim

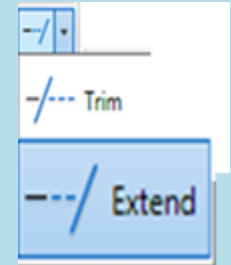


Result

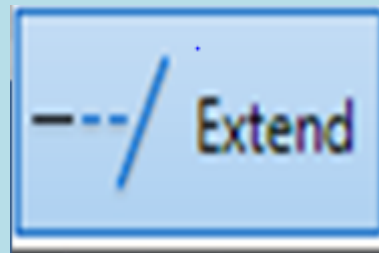


Lesson6

Extend

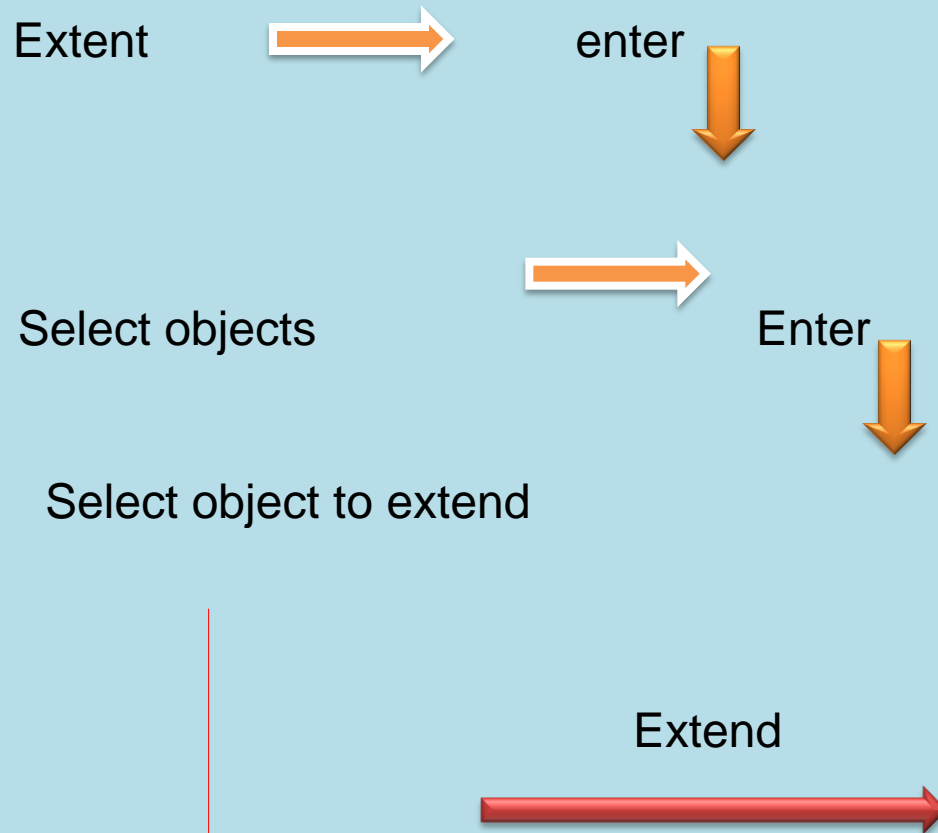


Ribbon/Modify:



At the Command line area: TYPE **EX** → enter

Procedure



Lesson7

Offset



Ribbon/modify:

offset



At the Command : TYPE **0**  enter

Procedure

Click offset → specify offset distance → enter



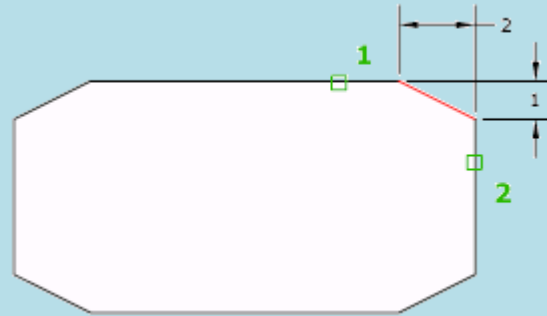
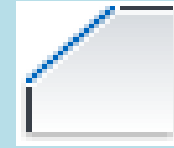
Select object to offset → specify point onside to offset → enter

Offset



Lesson8

Chamfer



Ribbon/modify: chamfer



Command line area: Type **CHA**  enter

Procedure

1 Click chamfer/type cha  D + enter

2 Specify first chamfer distance  enter

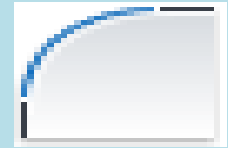
3 Specify second chamfer distance  enter

4 Select first line

5 Select second line

Lesson9

Fillet



Ribbon/modify:

Fillet



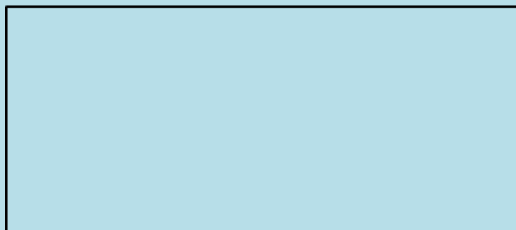
Command line area:

TYPE

F



enter




Fillet



Procedure

1 Click fillet /type **F**  R + enter

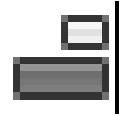
2 Specify fillet radius  enter

3 Select first object

4 Select second object

Lesson10

Align



Ribbon/modify:

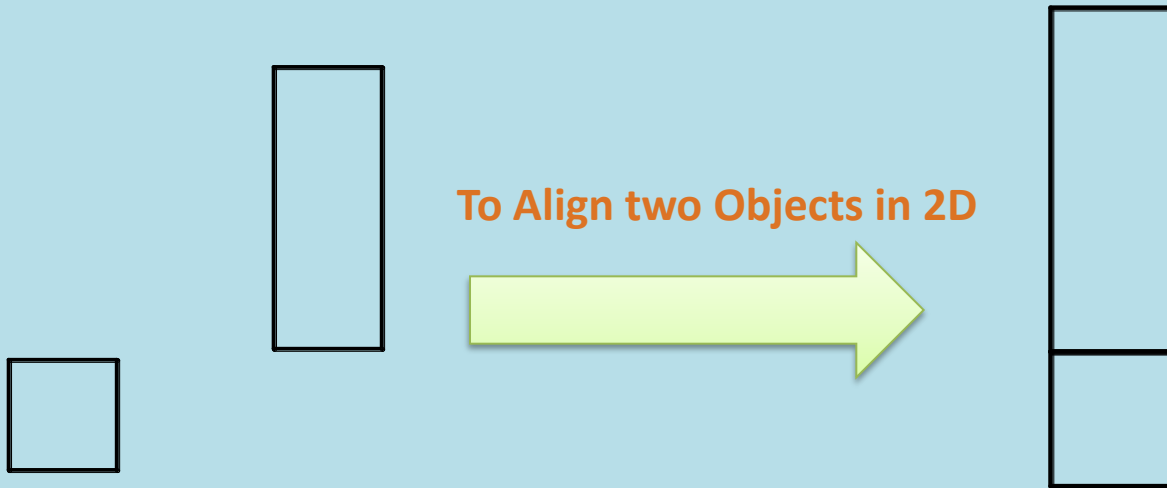
Align

Command line area:

TYPE AL



enter



Before Alignment

After Alignment

Procedure

1 Click Home tab ➡ Modify panel ➡ Align

Or

Command line area: TYPE AL

enter



2 Select the objects that you want to align

3 Specify a source point and then the corresponding destination point

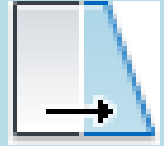
4 Specify a second source point followed by a second destination point

5 Press Enter to end the command

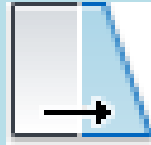
The selected objects are moved from the source point to the destination point

Lesson11

Stretch



Ribbon/modify:



Command line area:

TYPE **S**



enter

Procedure

1

Type S in command line area



enter

2

Specify base point or displacement



enter

Lesson12

Scale



Ribbon/modify:



Command line area:

TYPE SC



enter

Procedure

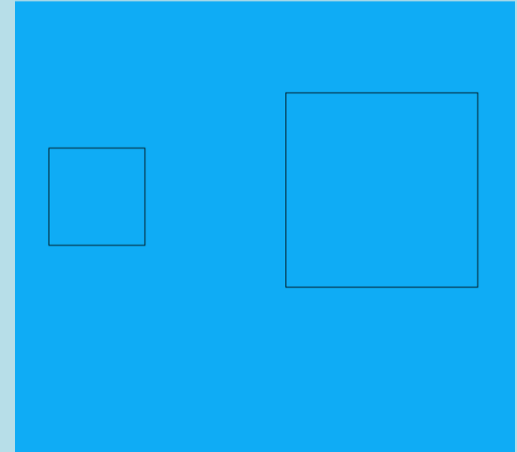
Scale by factor

1 Type SC → enter

2 Select objects → enter

3 Specify base point → enter

4 Specify scale factor or [Copy/Reference] → type number → enter



Scale by reference

- 1 SCALE <enter> (or SC <enter>) to start the SCALE command
- 2 Select the objects that you want to scale
- 3 <enter> when finished selecting
- 4 Specify a base point
- 5 R <enter> to use the “Reference” option
- 6 Pick 2 points on the object that is going to be scaled that you want to be scaled to the desired length.
- 7 P <enter> to use the “Points” option
- 8 Pick 2 points that establish the final length

Lesson13

Mirror



Ribbon/modify:



Command line area:

TYPE **MI**



enter

Before mirroring




after mirroring



Procedure

Scale by factor

- 1 Enter the Mirror command
- 2 Select objects to Mirror  Enter
- 3 Specify the first point of the Mirror line
- 4 Specify the second point
- 5 Left click
- 6 Right clic +enter

Lesson14

Array

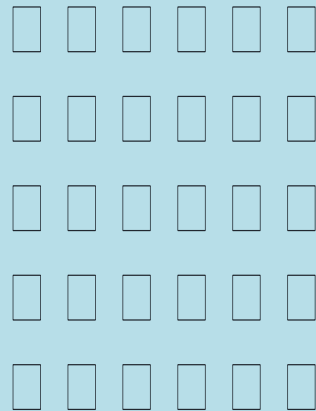


Arraying command is used to create multiple copies of objects in a polar or rectangular pattern.

Before



After

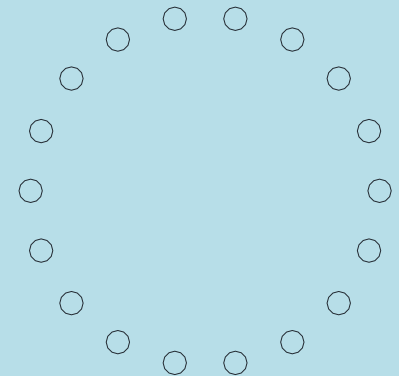


Arraying in rectangular pattern

Before



After



Arraying in polar pattern

Array
















Ribbon/modify:

Command line area:














TYPE **AR**




















enter

 Rectangular	 Columns: 4  Between: 30.5185  Total: 91.5554	 Rows: 3  Between: 31.1029  Total: 62.2059	 Levels: 1  Between: 1.0000  Total: 1.0000	 Associative  Base Point	 Close Array
Type	Columns	Rows ▼	Levels	Properties	Close

Arraying in rectangular pattern

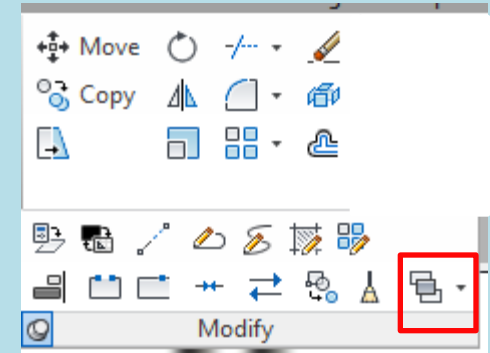
 Rectangular	 Columns: <input type="text" value="4"/>  Between: <input type="text" value="30.5185"/>  Total: <input type="text" value="91.5554"/>	 Rows: <input type="text" value="3"/>  Between: <input type="text" value="31.1029"/>  Total: <input type="text" value="62.2059"/>	 Levels: <input type="text" value="1"/>  Between: <input type="text" value="1.0000"/>  Total: <input type="text" value="1.0000"/>	  Associative Base Point	 Close Array
Type	Columns	Rows ▼	Levels	Properties	Close

Arraying in polar pattern

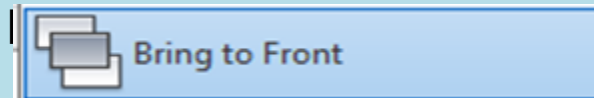
 Polar	 Items: <input type="text" value="8"/>  Between: <input type="text" value="13"/>  Fill: <input type="text" value="90"/>	 Rows: <input type="text" value="4"/>  Between: <input type="text" value="5.00"/>  Total: <input type="text" value="15.00"/>	 Levels: <input type="text" value="1"/>  Between: <input type="text" value="1.00"/>  Total: <input type="text" value="1.00"/>	 Base Point  Rotate Items  Direction	 Edit  Replace  Reset	 Close
Type	Items	Rows ▼	Levels	Properties	Options	Close

Lesson15

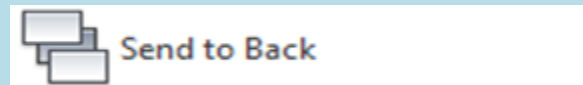
Draw order



The draw order capabilities are used to change the relative positions of objects.



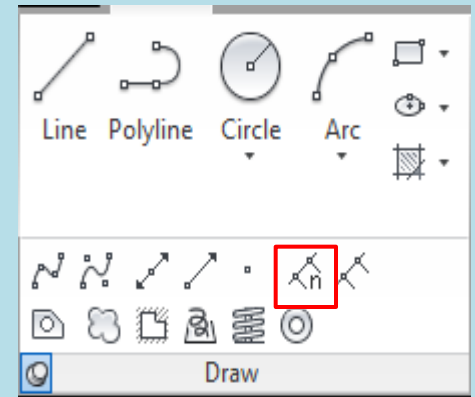
Bring to front is used bring the selected object in front of every other object .



Send to back command is used to send the selected object back of every other object.

Lesson16

Divide object

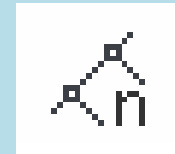


Home\Draw:

Command line area:

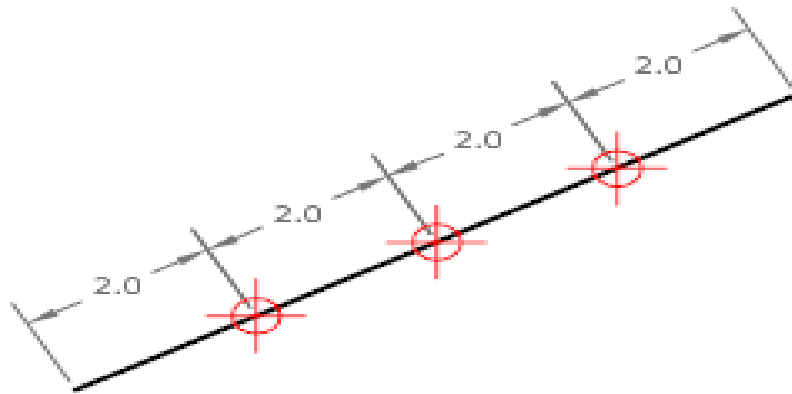
Type **DIV**

enter



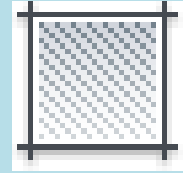
Procedure

- 1 At the command line area, enter **div**.
- 2 Select object to **divid**.
- 3 Enter the number of segments.
- 4 Use **PTYPE** or **DDPTYPE** to set the style and size of all point objects in a drawing.



Lesson17

HATCH



Home\Draw:



Command line area:

Type **H**



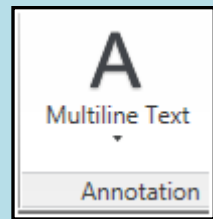
enter

Procedure

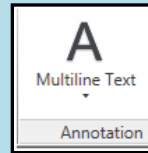
- 1 At the command line area, enter **H**.
- 2 In the Hatch and Gradient dialog box, click Add: Pick points.
- 3 In your drawing, specify a point inside each area that you want hatched, and then press ENTER. This point is known as the internal point.
- 4 In the Hatch and Gradient dialog box, Hatch , in the swatch box, verify that the sample pattern is the pattern you want to use. To change patterns, select another pattern from the Pattern list.
- 5 In the Hatch and Gradient dialog box, make adjustments, if necessary.
- 6 Under Draw Order, click one of the options.
- 7 You can change the draw order of the hatch so that the hatch is drawn either behind or in front of the hatch boundary, or behind or in front of all other objects.
- 8 Click OK.

Lesson18

Text



Draw toolbar/Annotation



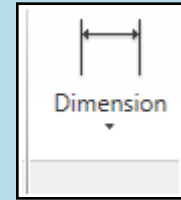
Annotate toolbar/Annotation



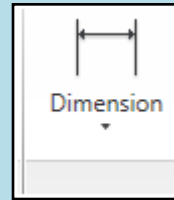
Command line area: type T  enter

Lesson19

Dimension style



Annotate toolbar

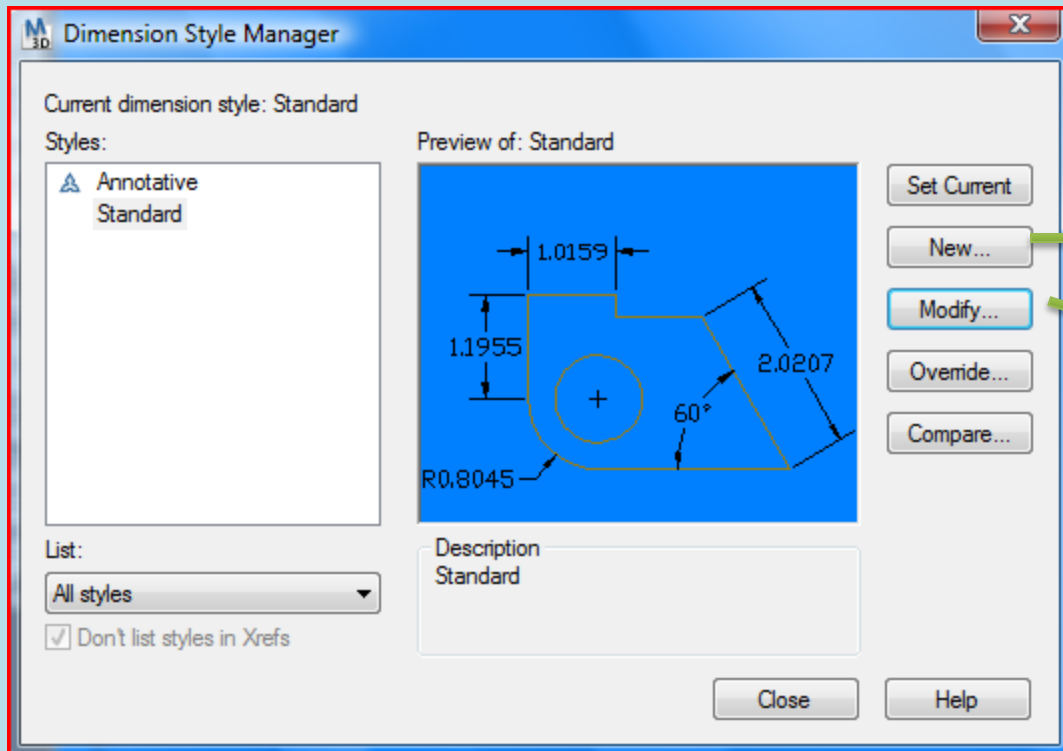


Command line area:

Type: ***D***  enter

Procedure

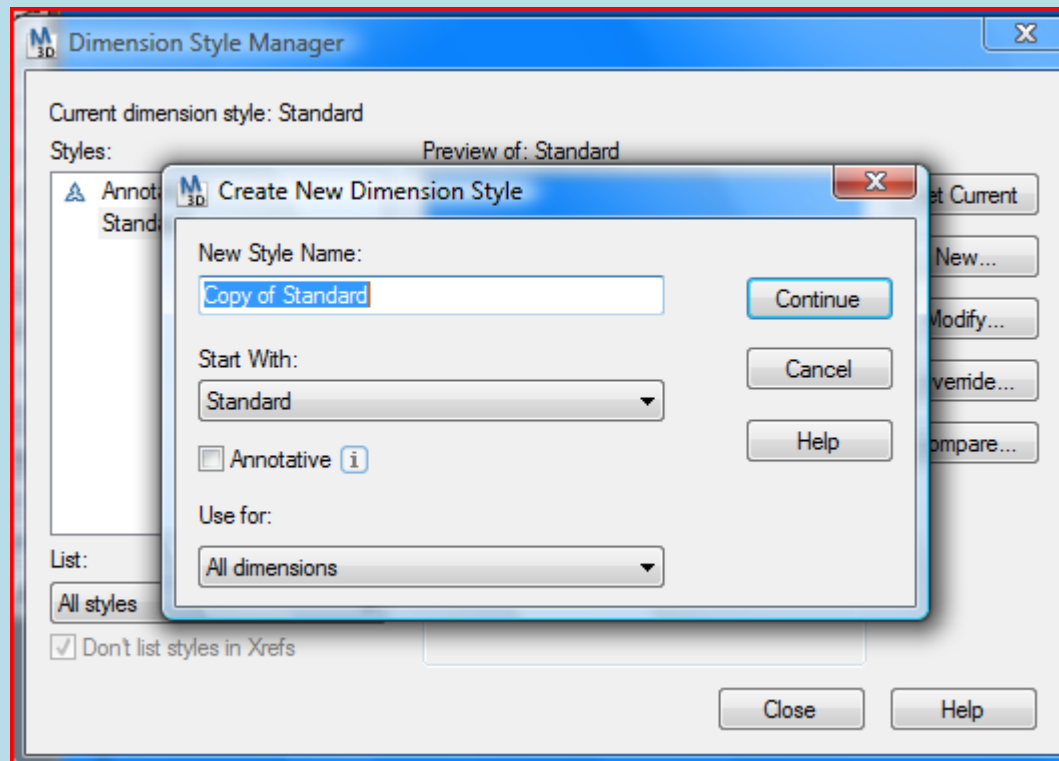
dimension <enter> (or D <enter>) to start the DIMENSION



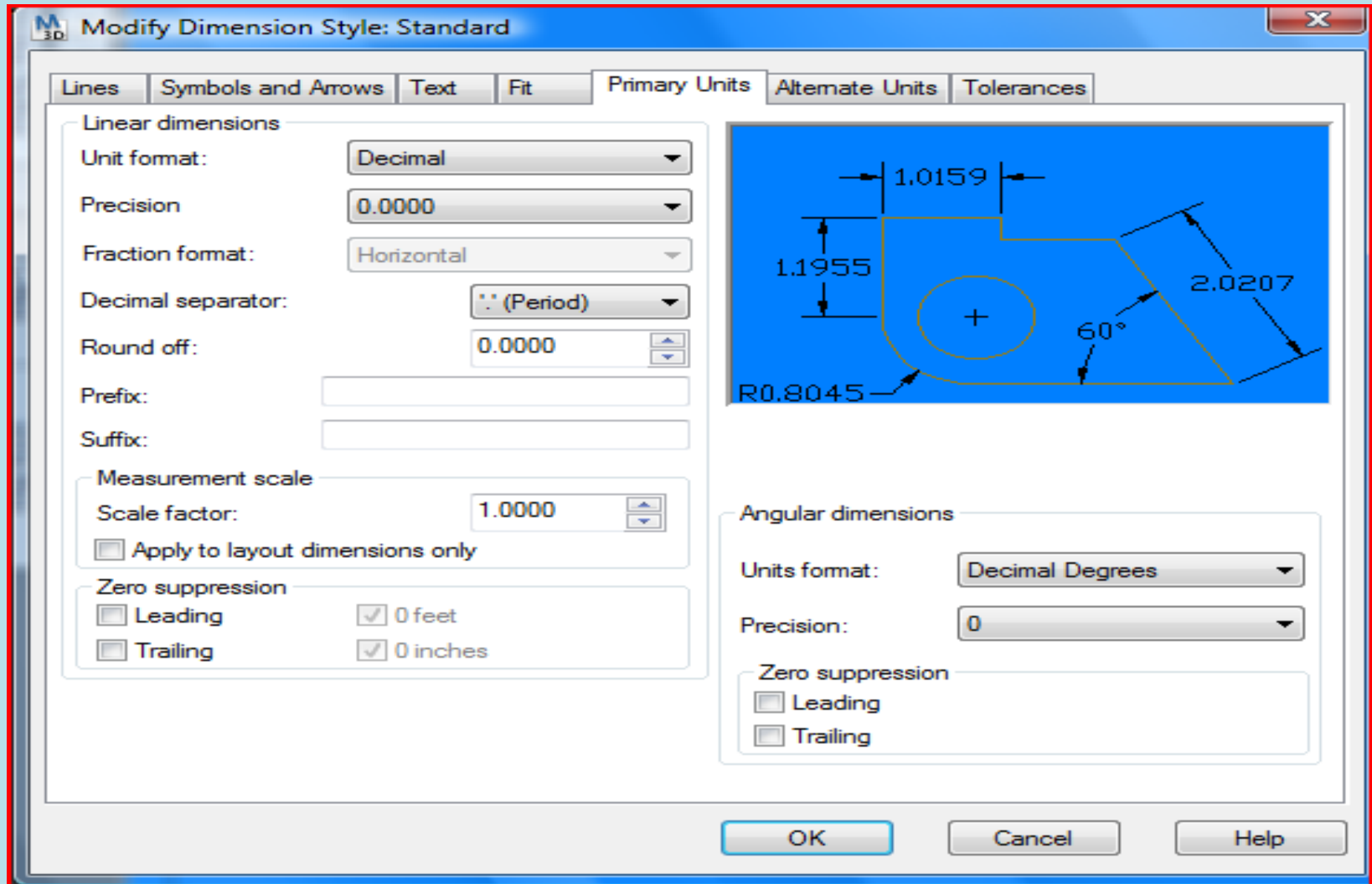
To create a new dimension style click new

To modify dimension style click modify:

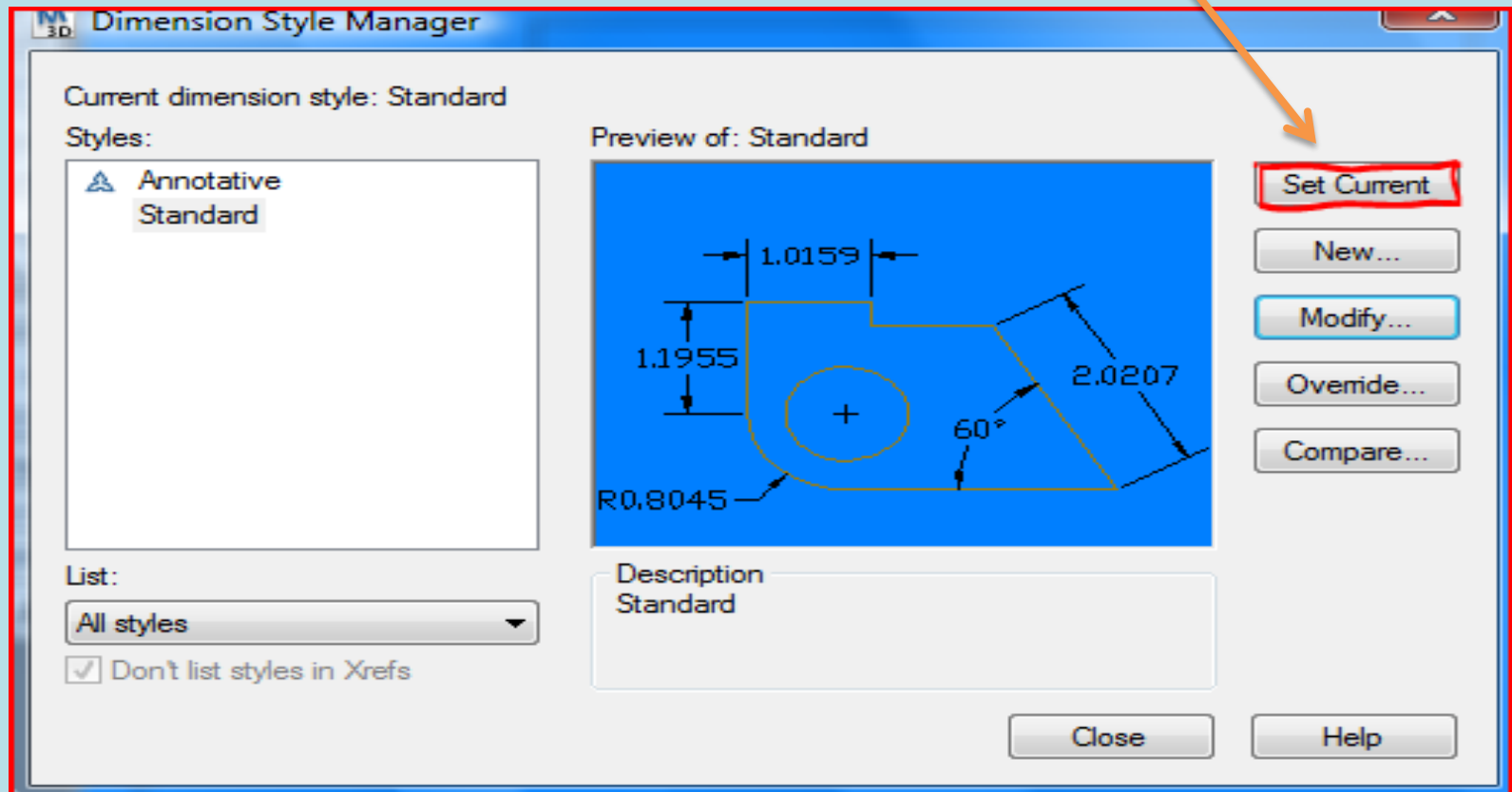
To create a new dimension style click new



To modify dimension style click **modify**:

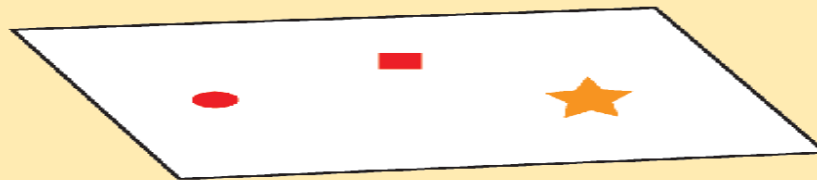


To validate new style, click **set courant**



Layers end their manipulation

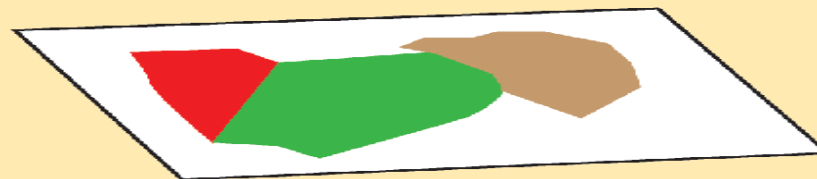
LAYERS



Symbols



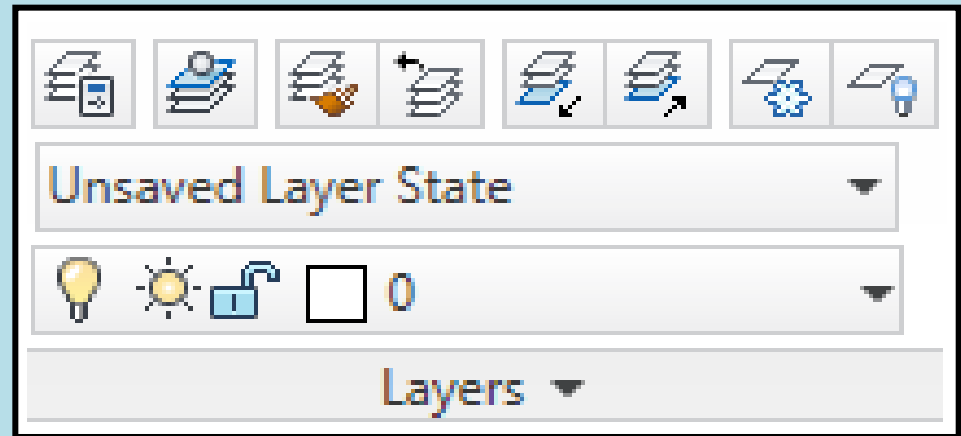
Lines



Polygons

Create layer

Home/AutoCAD Layers



Command line area:

TYPE : **LA**  **enter**

Procedure

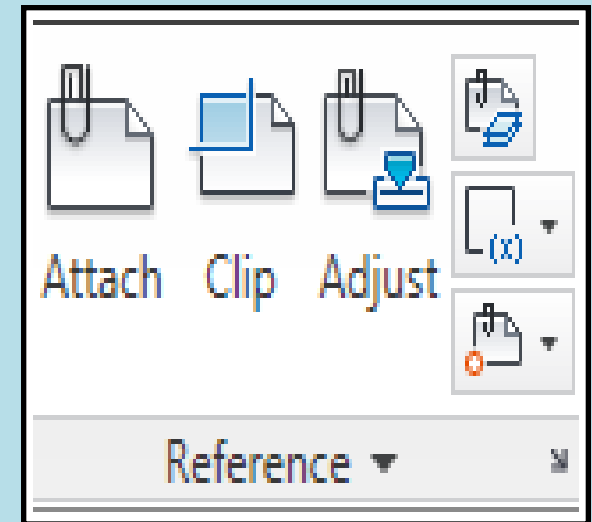
- 1 Layer <enter> (or LA <enter>) to start the LAYER command
- 2 In the layer properties, click the layer button
A layer name, such as layer1, is automatically added to the layer list.
- 3 Enter a new layer name by typing
- 4 To change the proprieties, click icons
- 5 Click in the description column and enter text
- 6 Click apply to save your changes, or click OK to save and close

Merge layers

In AutoCAD, you can merge unused layers into another used layer by using the **LAYMRG** command. It is helpful in cleaning up your layers and organizing them, especially when you need to convert your drawing standard to a different given standard. That might happen if you are working with someone else or another firm who has a different layer naming convention. To merge layers:

1. Type **LAYMRG** in the command prompt.
2. Select the *layer* you need to merge.
3. Hit *enter* or the *spacebar* on your keyboard.
4. Select the *target layer* you want to merge to.
5. Select or type “yes” when asked.
6. Press *enter* or the *space bar* on your keyboard.

INSERT IMAGE



Insert/AutoCAD



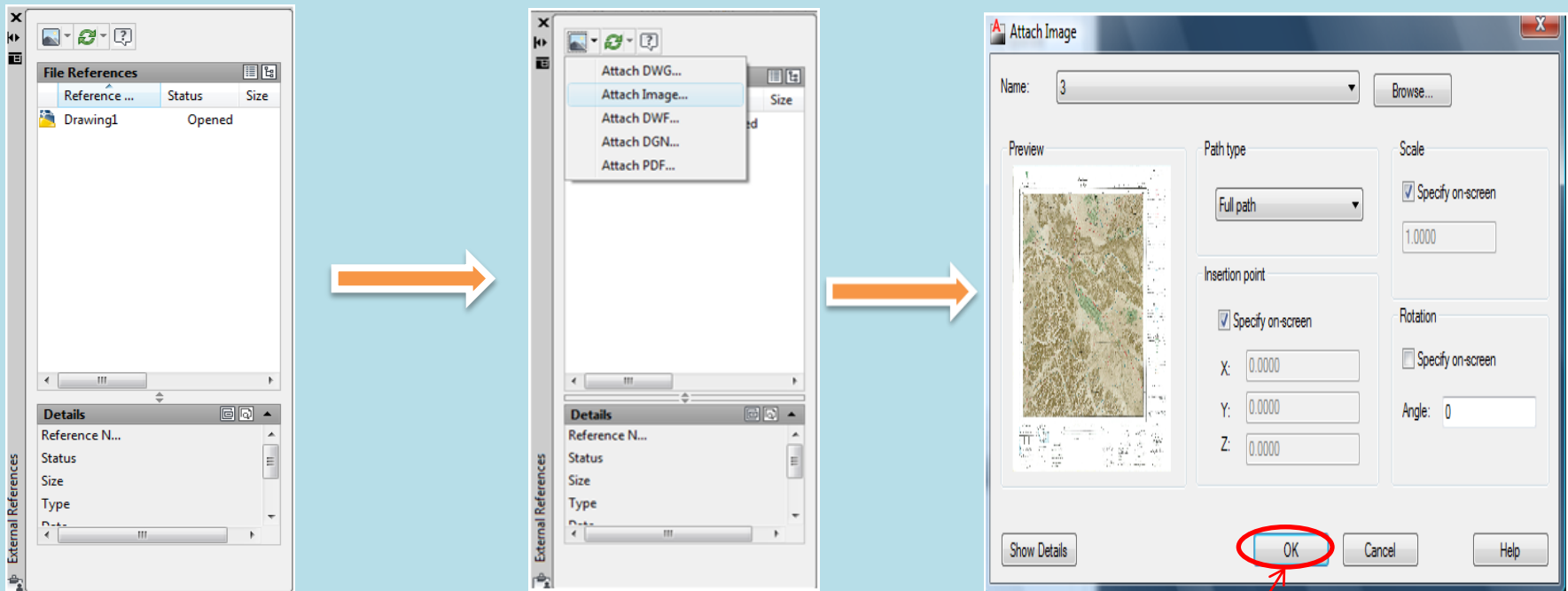
Command line area:

TYPE : **IM**



ENTER

Procedure



CLICK OK

PLOTTING

Application menu: **print**  **plot**

Quick access toolbar

Command line area: Ctrl+p

Plotting steps

Before plotting you should specify:

1/ The printer name

2/ The paper size

3/ The plot area

4/ The plot offset

5/ The plot scale

6/ The drawing orientation

Plotting steps

