SRI VENKATESWARA COLLEGE OF ENGINEERING AND TECHNOLOGY
(Autonomous)
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DEPATMENT OF MECHANICAL ENGINEERING

COMPUTER AIDED DRAFTING

LABORATORY MANUAL

Prepared by

DEPATMENT OF MECHANICAL ENGINEERING

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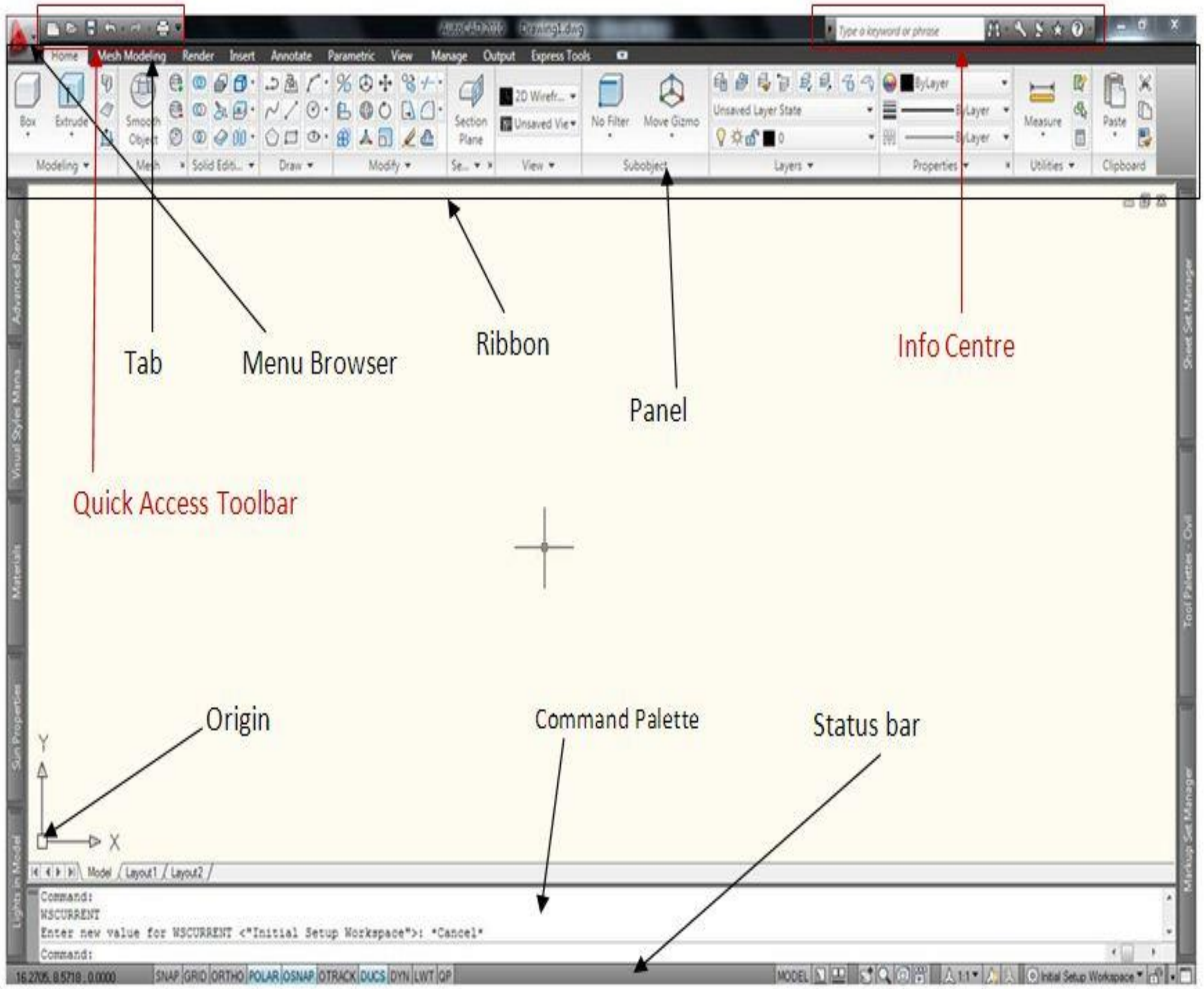
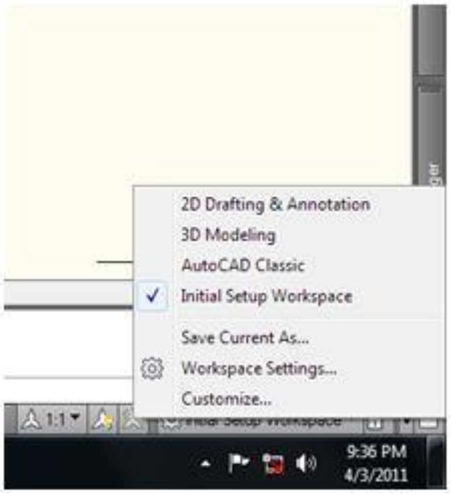
S. No	Date	Name of the Experiment	Marks Awarded					Initials
			Con. Of Exp. **(10)**	Observation **(10)**	Viva **(05)**	Record (05)	**TOTAL (30)**	

1.1Introduction:

First of all let's have a look at the AutoCAD 2010 interface.

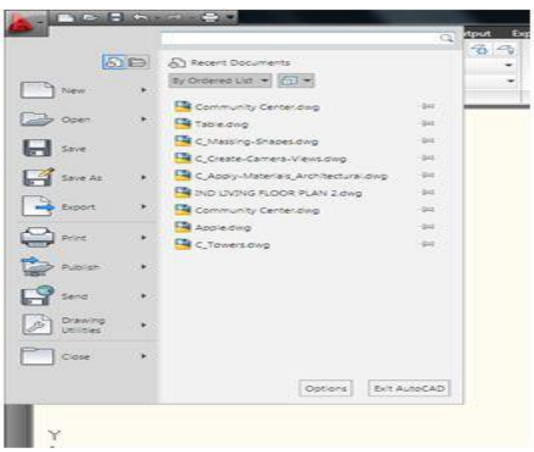


When you open it the initial workspace is at Initial Setup Workspace. You can change it to 2D Drafting & Annotation, 3D Modeling and AutoCAD Classic mode as necessary by clicking the Workspace Switching menu at lower right corner. We will proceed with 2D Drafting & Annotation for 2D drawings. In this workspace only the tools for 2D drawing are given. In 3D Modeling, all the tools for 3D drawing are given. But in   Initial Setup Workspace, both the 2D & 3D drawing tools are given. So sometimes it is preferable to work in Initial Setup Workspace. In AutoCAD Classic the interface of the software from 2000-2008 are given. If anyone, who is very much habitual with the previous versions, can use this workspace for convenience instead of the new one which was adopted from 2009.

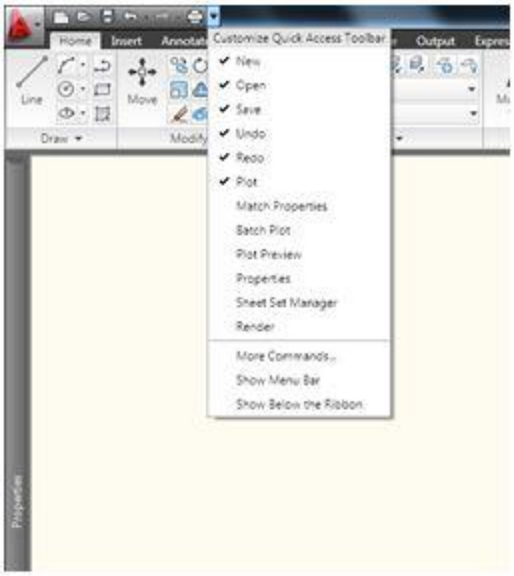


1.2 Different Parts of the Interface:

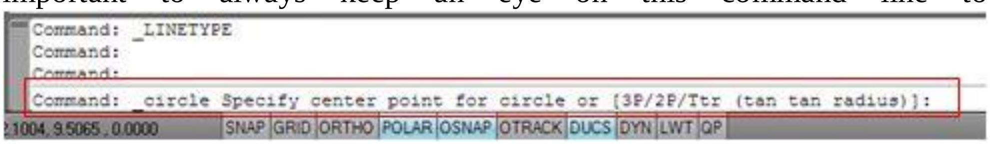
At first , at the upper left corner, you will find the Menu Browser (see the first figure). By clicking it, you will have the options to open new file, previous saved file, save option, print, publish etc. commands. You can see the recently opened documents. By clicking "Options" button, you can modify your drawing preferences.



In the Quick Access Toolbar, there is provided the same options like new file, save, open etc. By clicking the drop down arrow, you can add or deselect commands.



In the **InfoCentre**, the functions are similar. The functions are search, help, action centre etc.
Below these is the ribbon. This is made of several tabs. Again the tabs are made of several panels. These panels provide tools for drawing and modification.
In the **Command Palette**, in the upper part, the commands just entered are shown. In the lower part, the current command is shown. You can write command here and it shows what to do next. So it is very important to always keep an eye on this command line to give the correct inputs.

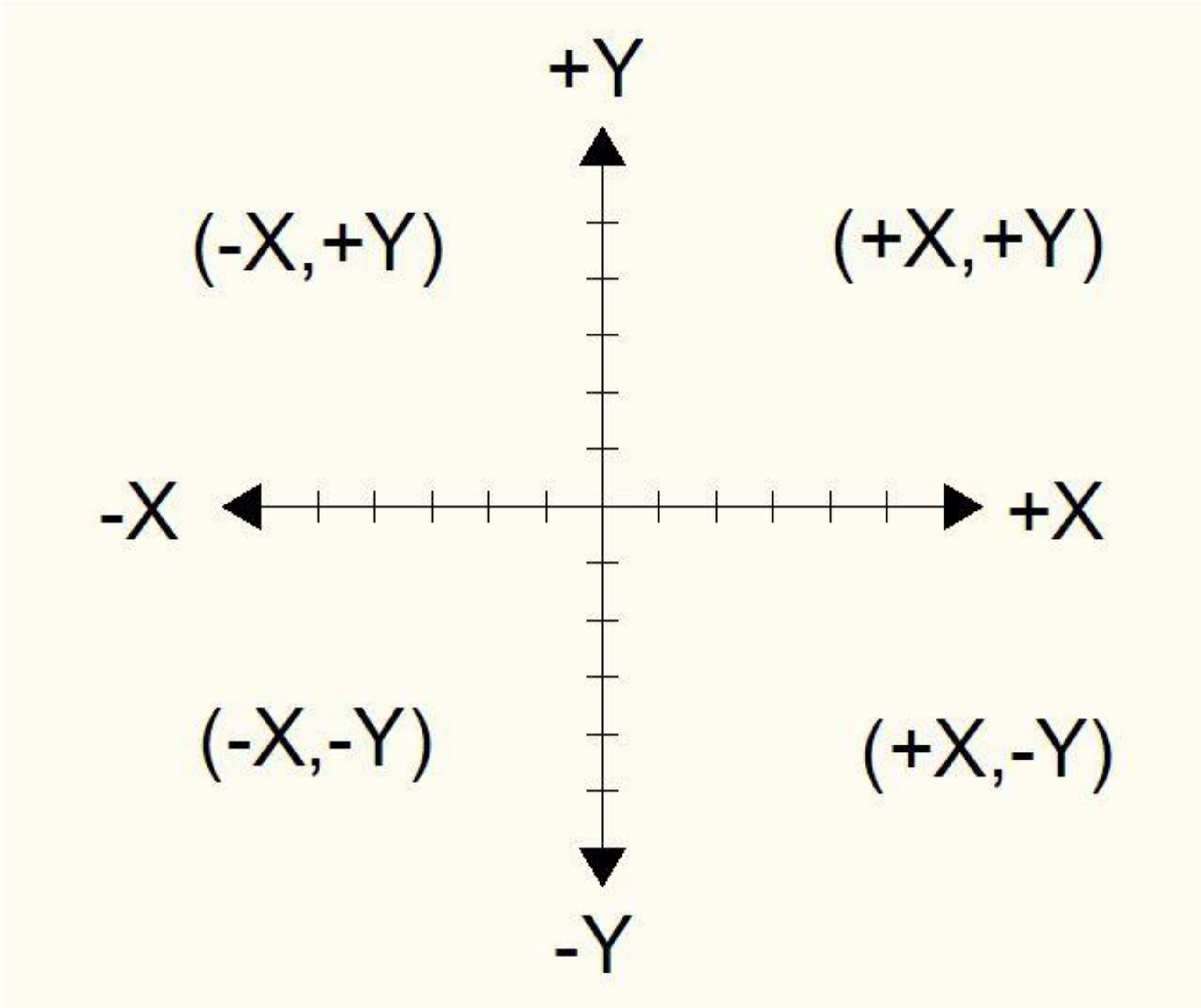


In the **Status bar**, there are some buttons to toggle (turning on/off) some functions like snap, polar, ortho etc. which are very necessary during drawing. And in the right side there are some helping buttons like switching workspace (which you have seen previously), pan, zoom, steering wheel, quick view layout, clear screen etc. the ribbons at right and left side of the interface are docked panels.

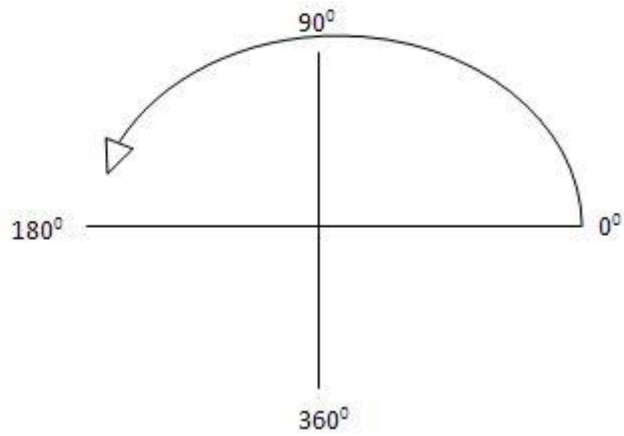
1.3InputSystem:

Everything drawn on AutoCAD are drawn at accurate scale. The drawings are placed on the sheet by co-ordinate system. In 2D, it is two axis -X,Y co-ordinate system and in 3D it is three axis- X,Y,Z co-ordinate system. You can see it while moving the cursor (which is actually called **Cross Hair** in AutoCAD) in the workspace, at the lower left corner the co-ordinates are shown simultaneously.

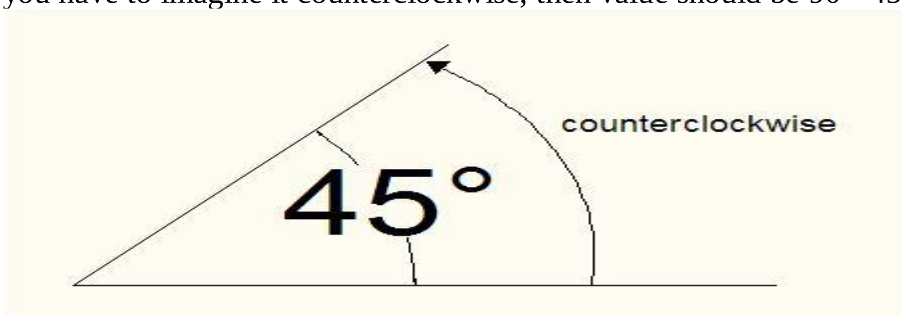
The default co-ordinate system is called WCS (World Co-ordinate System). Everything you draw, are drawn in **WCS**. Sometimes designers need to change the default co-ordinate system for drawing purpose. That is called **UCS** (User Co-ordinate System). So, when you are drawing a line 10 units long, the inputs should be (0,0) and (8,0) for horizontal length; or (0,0) and (0,8) for vertical length (The commands for line, circle, rectangle etc. are coming next. This is just to give the conception of using co-ordinate system). Now, the points you enter are the **absolute** points. If you need to put any point with respect to the entered last point, then you have to write the symbol “@” before the co-ordinate of the desired point. This point then will be called the **relative** point.

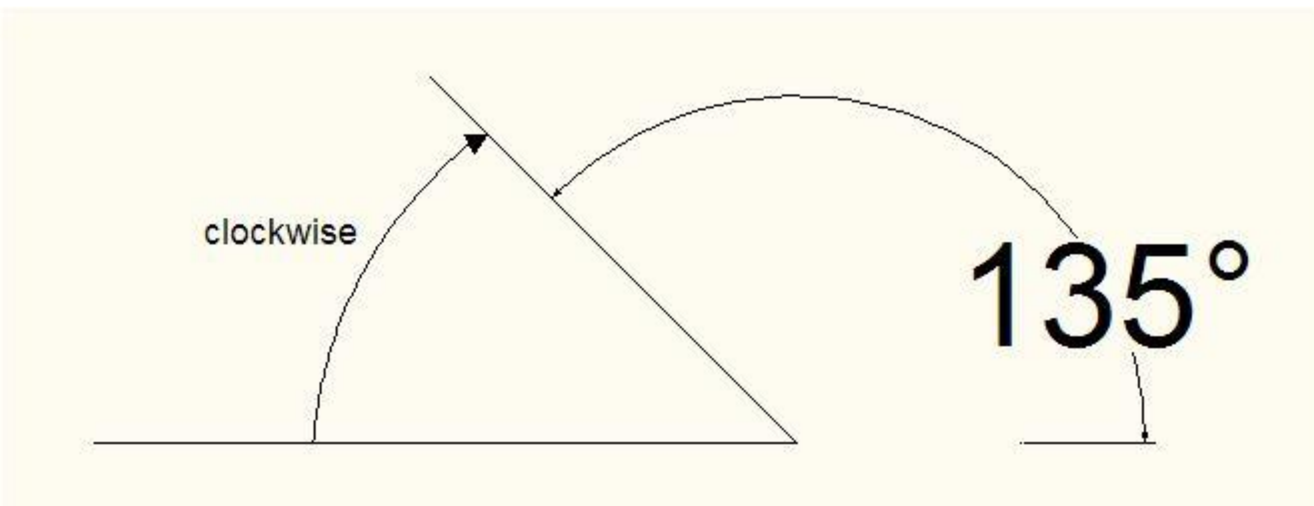


Those were all linear inputs. But sometimes you have to draw angular lines. In that case you have to mention the angle. In AutoCAD angle is measured counterclockwise. The 0 degree starts from 3 o'clock, then increases counterclockwise.



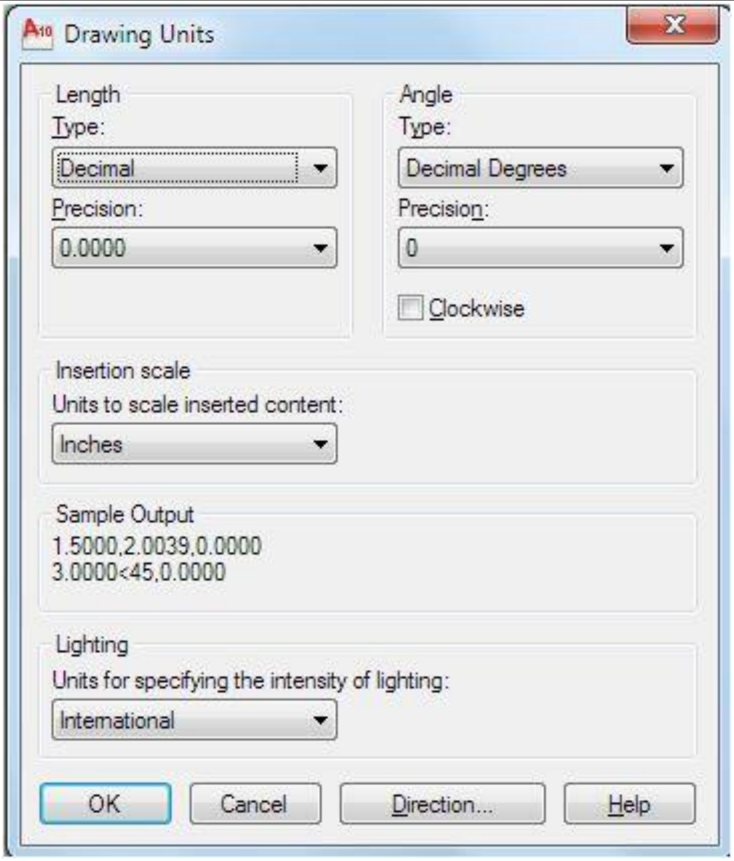
For example, you have a line and need to draw another line at the left point at 45^0 angle (Fig. 1). Now from the left point, upside the line, if you start measuring angle, it goes counterclockwise. So simply give the angle value 45^0. But if you have to draw the line from the right corner, the measurement goes clockwise. So you have to imagine it counterclockwise, then value should be $90^0+45^0= 135^0$.



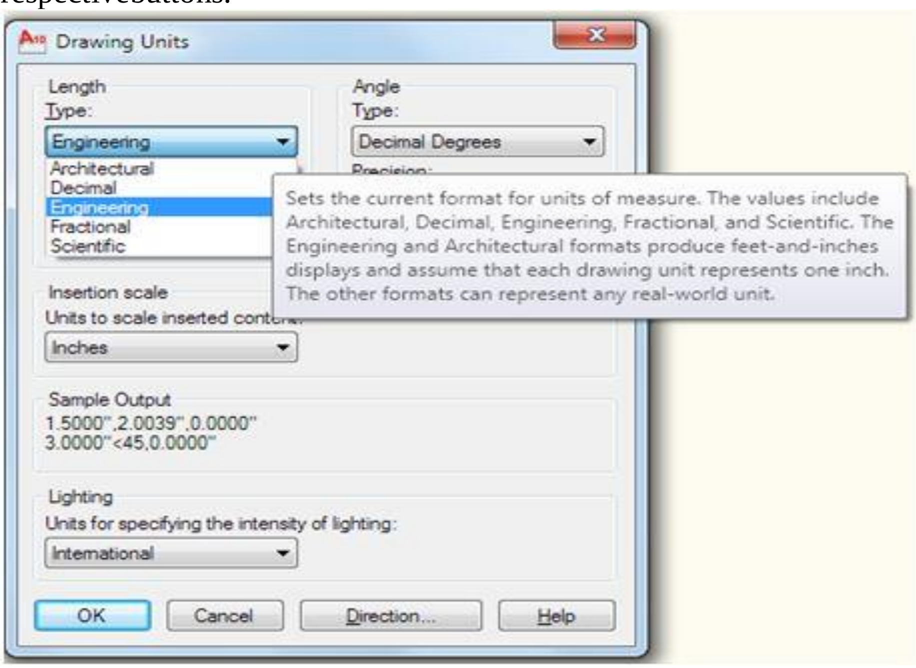


2.1Choosingunits:

To draw any plan, you need to draw some objects, like line, circle, rectangle etc. But before starting drawing, you have to choose the unit you want to work with. To set unit, Write-“**un**”, then press **enter**. You will have the dialog box:



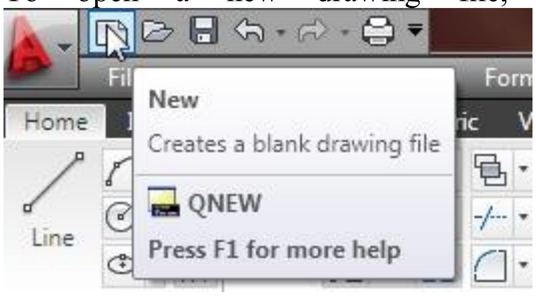
Select your length type, angle type, insertion scale and lighting intensity by clicking on the respectivebuttons.



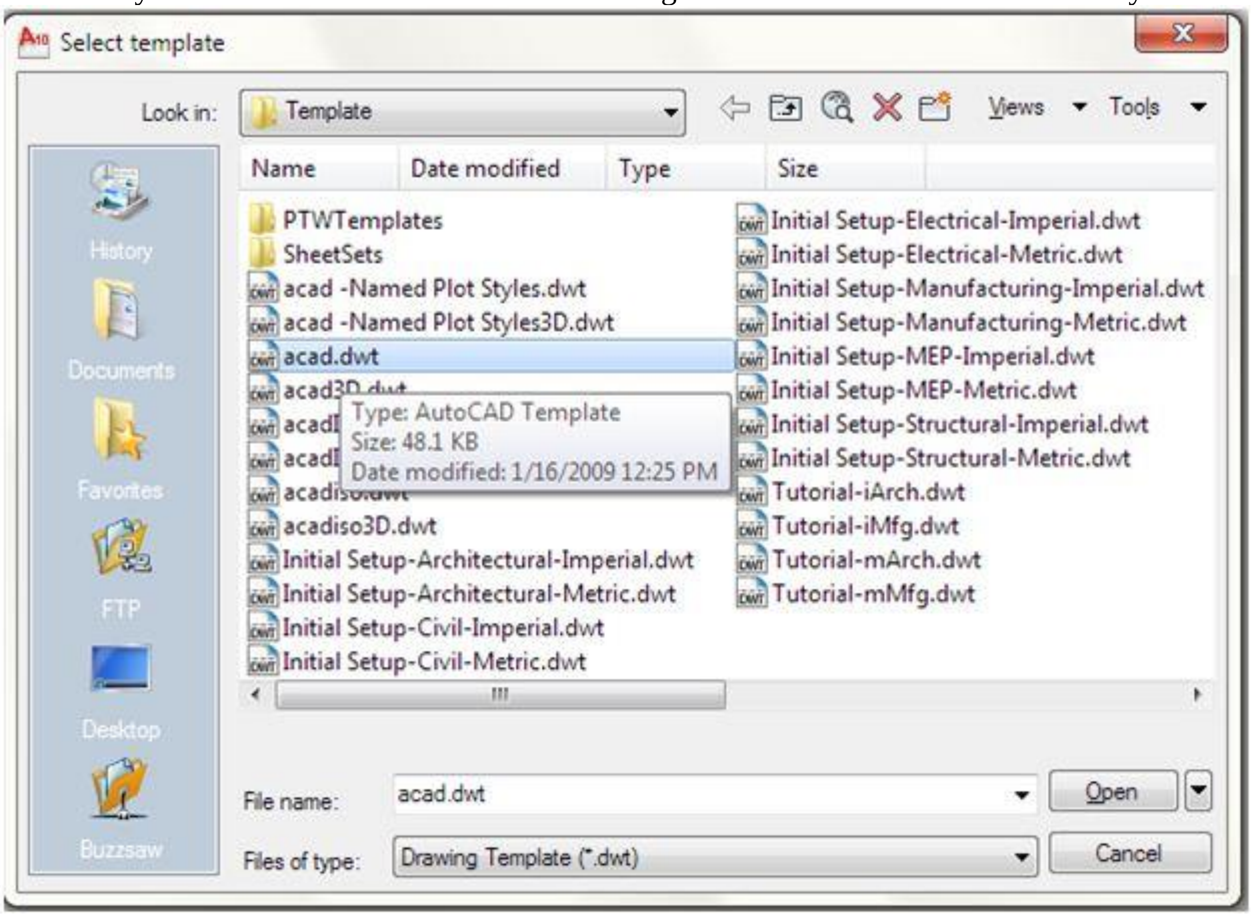
It is convenient to choose Engineering or Architectural unit with "inches" scale. So choose so and choose the angle unit as you need. The lighting intensity unit is applicable for 3D drawing.

2.2DrawingObjects:

To open a new drawing file, click the "New" button on Quick Access Toolbar.

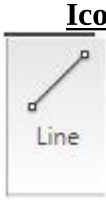


Then you will have a dialog box to choose your template.



Select "**acad.dwt**" and press **Open** or double click. Now type "Z" <Enter> "E" <Enter>; this will zoom into to the extents of the drawing area and make it easier to see what you are drawing.In the **Home** tab in **Draw** panel you will have the objects to draw.

Line

Keystroke	**Icon**	**location**
" L"		Home > Draw > Line

To construct a line from one point to another. The command should be co-ordinate based (as mentioned previously) or directly inputing the length (which is actually easiesr and used in practical field).
For example, if you need to draw a line 10 inch long, your command should be:

- "L"
- Enter (orclicktheicon)
- Specify first point : Click at the starting point and hold cursor towards the desired direction
- Specifynextpoint:10"
- Enter
- Esc

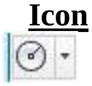
If you need to draw a line 10mm away at the right of your last entered point, then you can do it by first drawing a 10mm line, or you can give direct input. For that you have to use "@" symbol. The starting point should be indicated by co-ordinate (@10,0) then the length either byco-ordinateorbygivinglinelength(**relativeco-ordinate**).

Now, if the line needs to be drawn at a specefic angle, such as 45°, then the command should be:

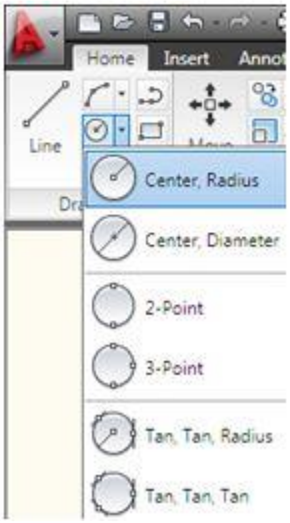
- "L"
- Enter (orclicktheicon)
- Specify first point : Click at the starting point and hold cursor towards the correct direction
- Specifynextpoint:10"
- <45
- Enter
- Esc

So, you can see that, during introducing angle the command is actually "Length < Angle" (**polar co-ordinate**).

<u>Circle</u>

<u>Keystroke</u>	**<u>Icon</u>**	**<u>location</u>**
"C"		Home > Draw > Circle

Draw a circle based on a center point and a radius. But there are other inputs to draw. Click the arrow in the icon, there will be a drop down menu showing you the other options like center & diameter,2point,3pointetc.



For example, to draw a circle of radius 4", center at (1',1'), the commands should be:

- “C”
- Enter(orclickicon)
- Specifycenterpointofcircle:1' ,1'
- Enter
- 4"
- Enter

Rectangle

Keystroke	**Icon**	**location**
"R"		Home > Draw > Rectangle

Can be draw in 2 ways:
No.1, showing two opposite diagonal points of the rectangle.
No.2, giving the arm lengths of the rectangle (first along X axis, secondly along Y axis)
For example, to draw a rectangle 10" X 5" , the commands should be:

- “R”
- Enter(orclickicon)
- SpecifyfirstcornerpointSelectanypoint
- @
- 10mm ,5mm
- Enter

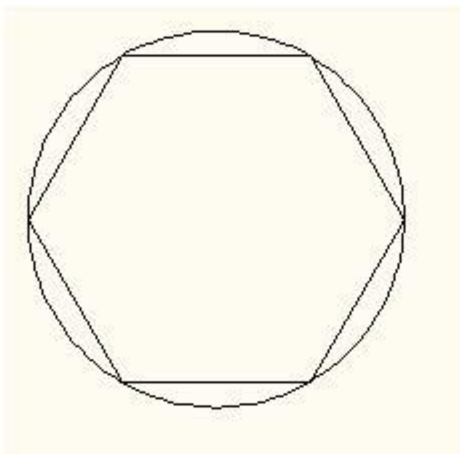
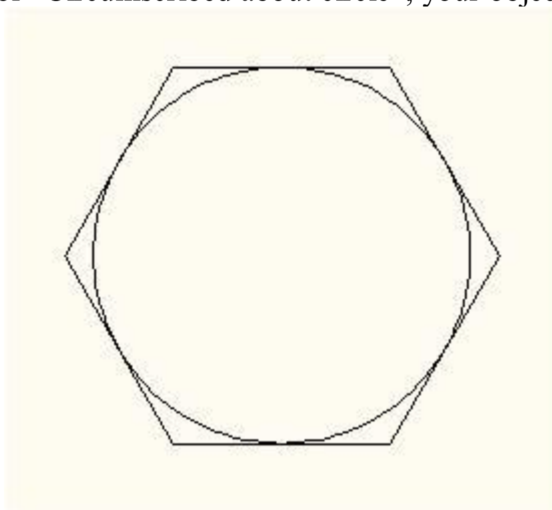
Polygon

Keystroke	**Icon**	**location**
"POL"		Home > Draw > Polygon

Used to construct polygon, rectangle, triangle or any object consisting as many sides as needed ofequallength.

Example:
- "pol"
- Enter(orclickicon)
- Enterno.ofsides:6
- SelectCenterofpolygon:20,12
- Selectoneoption: **Inscribedincircle** or **Circumscribedaboutcircle**
- Enterradiusofcircle: 4

For the command "Inscribed in circle" or "Circumscribed about circle"; your object will be like

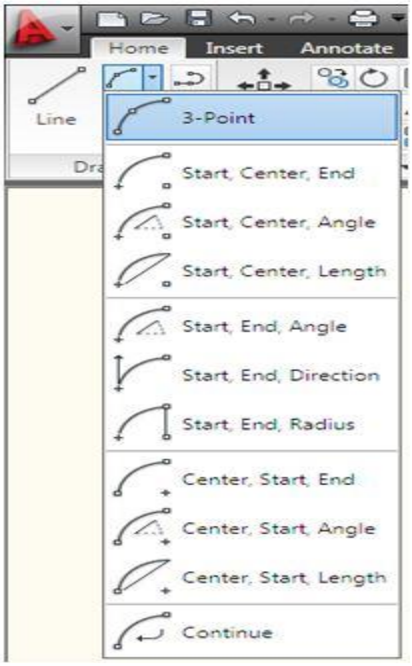
 or 

Remember, if you choose "Inscribed in circle", the length of each sides of the object will be equal to the value of radius. On the other case, the lengths will be greater than the radius.

<u>Arc</u>

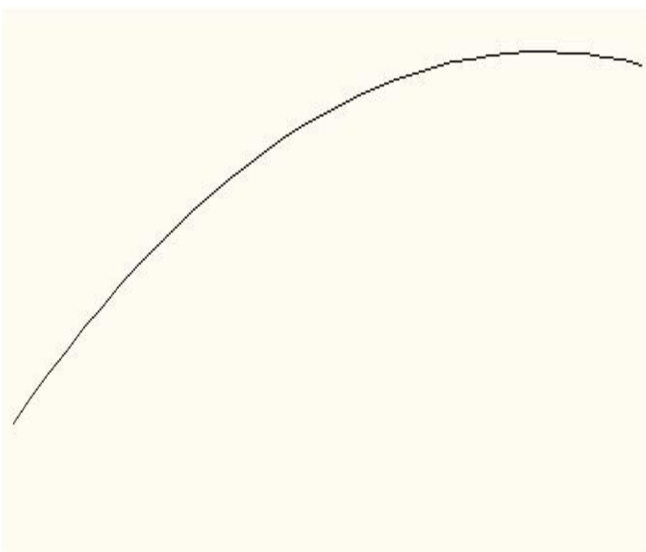
<u>Keystroke</u>	**<u>Icon</u>**	**<u>location</u>**
"A"		Home > Draw > Arc

Draws an arc of specefic dimension. Can be drawn in various ways. Click the arrow button on theicon.Theoptionswillappear.



Fora3pointarc,thecommandshouldbe:

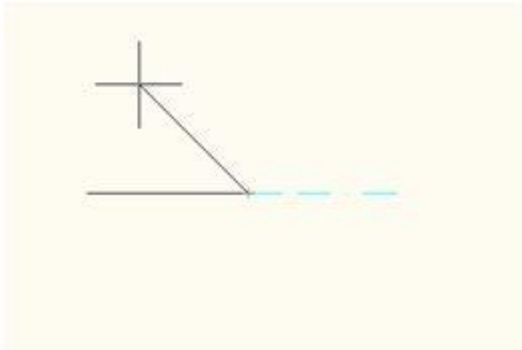
- “a”
- Enter(orclickicon)
- Speifystartpointofarc:20,10
- Enter
- Specifysecondpointofarc:5,2
- Enter
- Specifyendpointofarc:5,2
- Enter

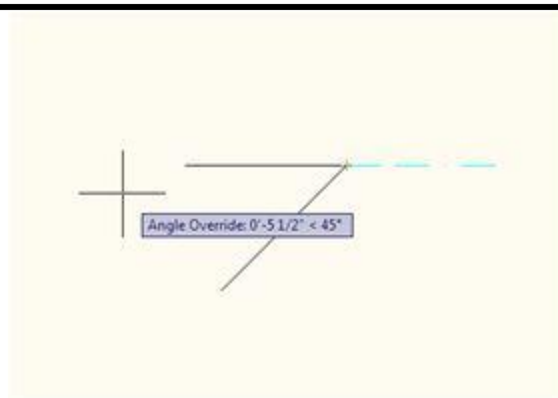


3.1 Details on Line:

In the previous lesson, you have learnt how to draw a line. But still problem can arise regarding placing a line at a certain direction. So clear your conception.

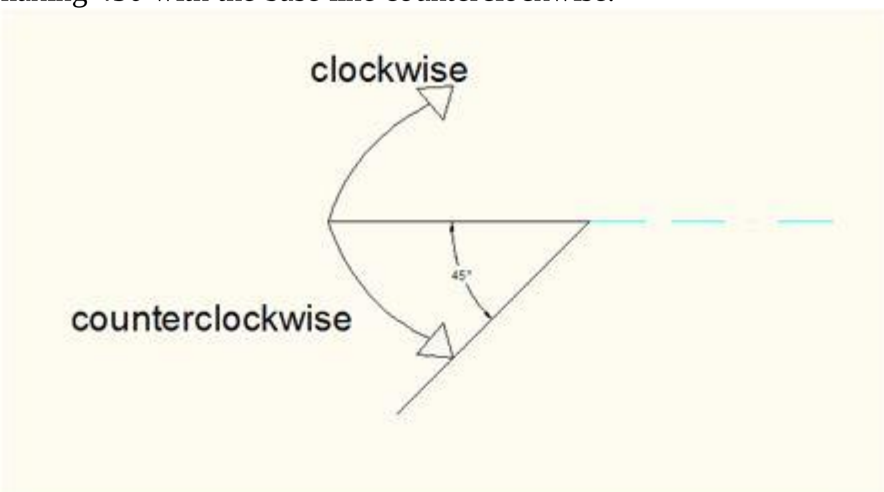
Lookatthetwopictures:



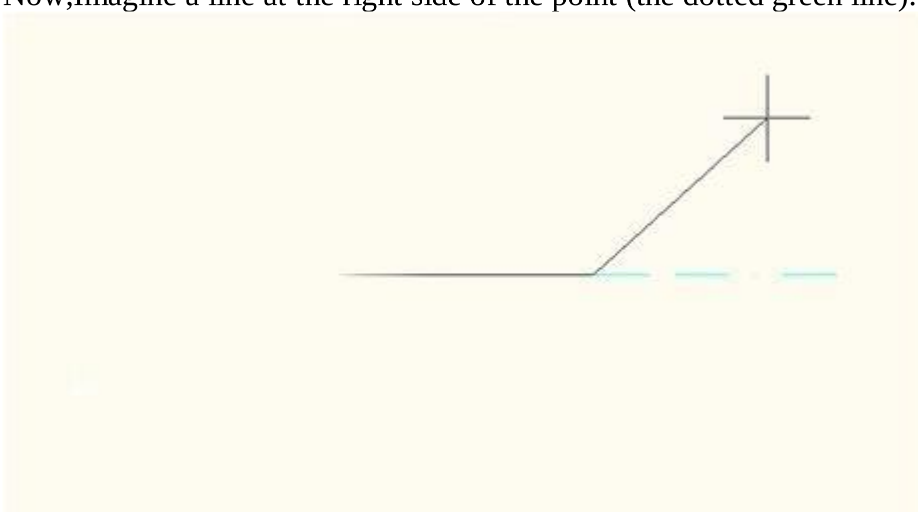


Here, you want to draw a line at 45 degree angle with the horizontal line at the upper side. But when you give the command for 450, the line goes downward. Why?

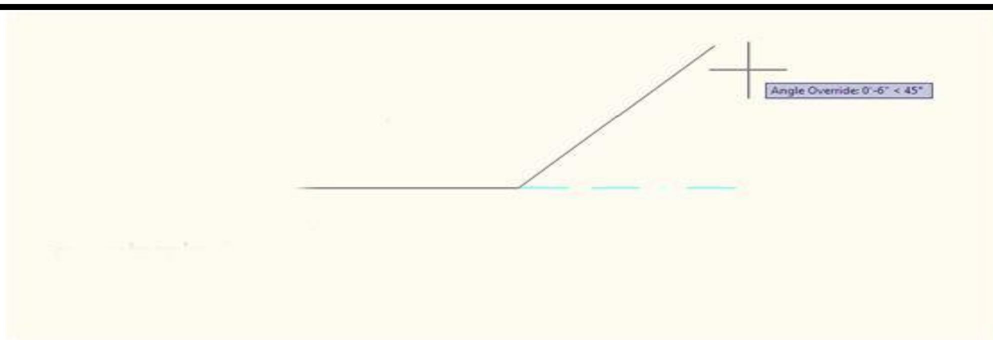
Because, If you keep the starting point fixed and rotate the base line, it goes clockwise at the upper side and counterclockwise at the downward side. As a result, if you give input of 450 angled line, it goes downward making 450 with the base line counterclockwise.



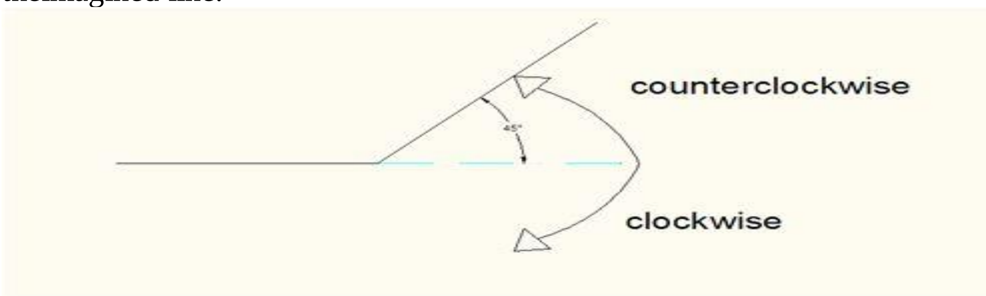
Now,Imagine a line at the right side of the point (the dotted green line).



Give line command and hold your cursor above the green line - as you want to draw a line above it at 45 degree angle. Now give input for 45 degree.



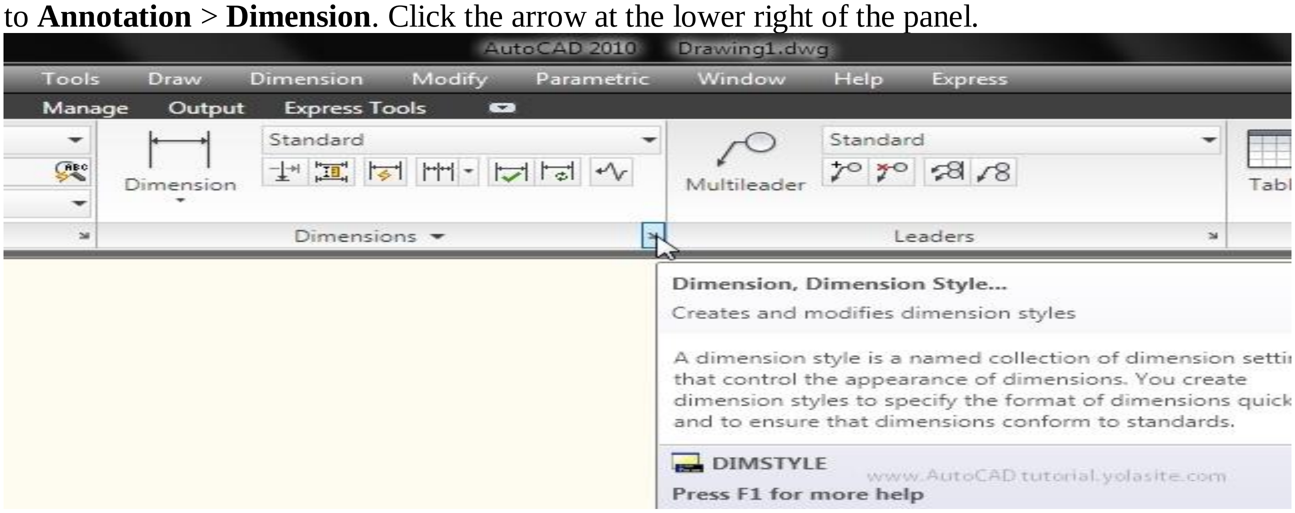
This time the line stays at the desired side. Because that side is the counterclockwise side of theimagined line.



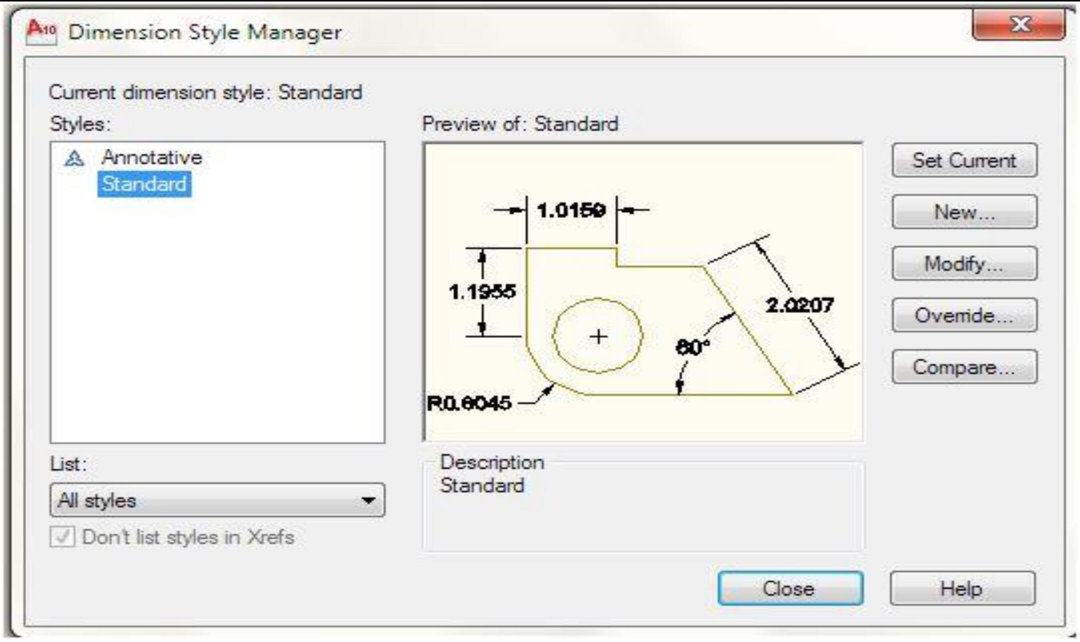
But this line is not actually drawn there. So why didn't the line go downward?
Because AutoCAD considers **the line (to measure angle with respect to) towards which you hold your cursor, even if it is not drawn on the paper; And of course only the horizontal line.**
Watch video tutorial on the three exercises given in the previous lesson. Please watch carefully the input values.

4.1Dimension:

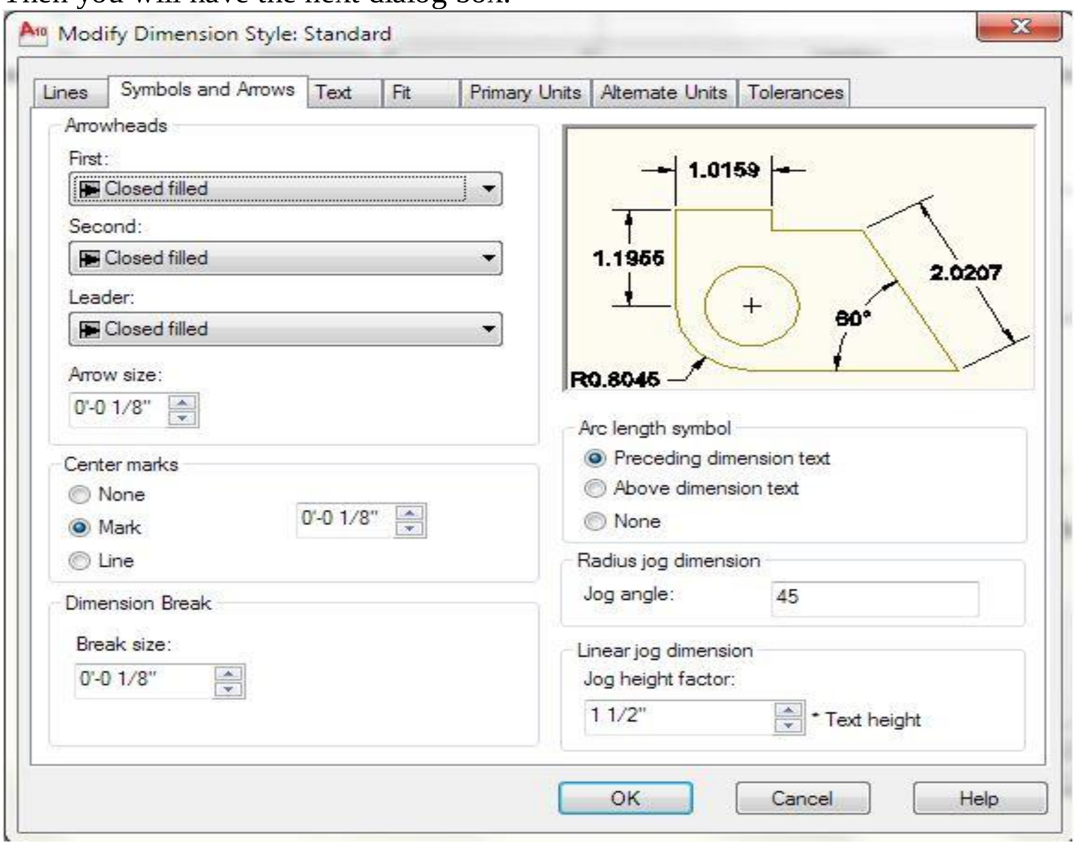
After you have drawn any object, you will need to show the dimension in the drawing. To do that go to **Annotation** > **Dimension**. Click the arrow at the lower right of the panel.



You will have the dialog box as below. Click Modify button.



Then you will have the next dialog box:



Then do your modification for Text size, Primary Units, Arrow size etc. After you have finished, click OK. You will be redirected to the previous dialog box. There click SetCurrent, then Close. Then you can use both Home > Annotation ; or Annotation > Dimension to show any dimension.

4.2 Drawing Objects:

Polyline

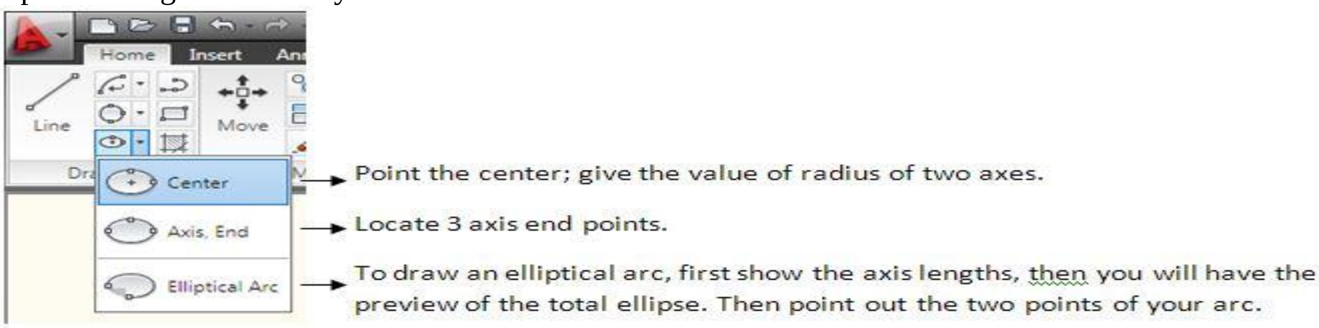
Keystroke	Icon	location
PL		Home > Draw > Polyline

In case of line, you can construct more than one line under one command. But all the lines will be considered as independent lines. But if you use polyline, all the lines constructed under one command will be considered as a single line. The input is same as line.

Ellipse

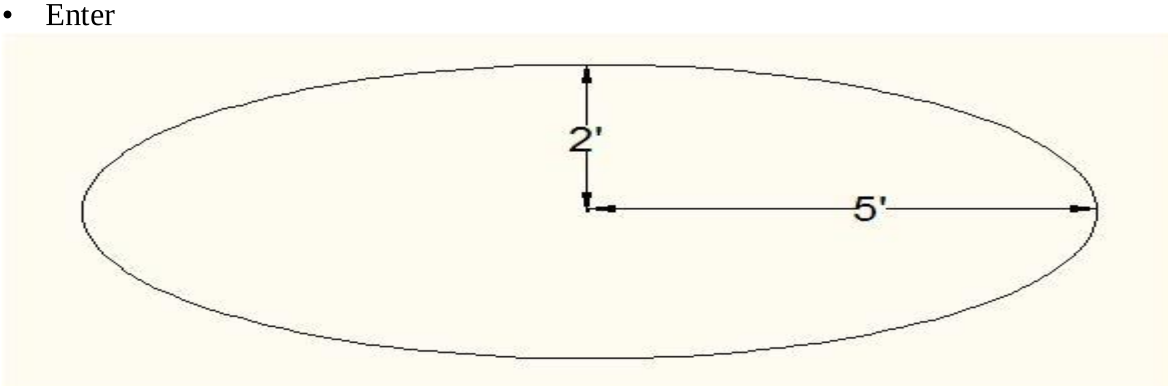
Keystroke	Icon	location
EL		Home > Draw > ellipse

Input can be given in 3 ways:

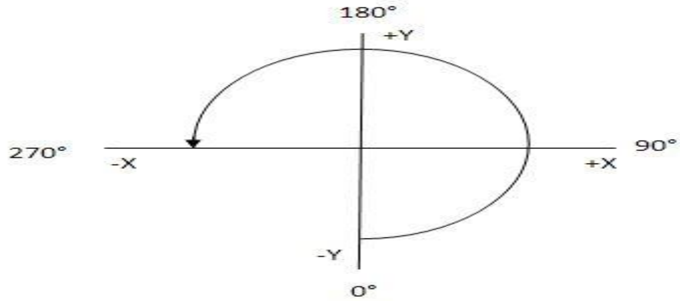


To draw the below ellipse, the command should be:

- "el"
- Enter (or click icon)
- Select center by clicking giving co-ordinate
- 5
- Eneter
- 2
- Enter



You have seen how to define angle in AutoCAD. But during you construct an elliptical arc, you have work with a slightly different conception. In conventional drawing 0° is started from 3 o'clock and then proceeds counterclockwise. But during drawing elliptical arc, the 0° starts from 6 o'clock or negative Y axis.

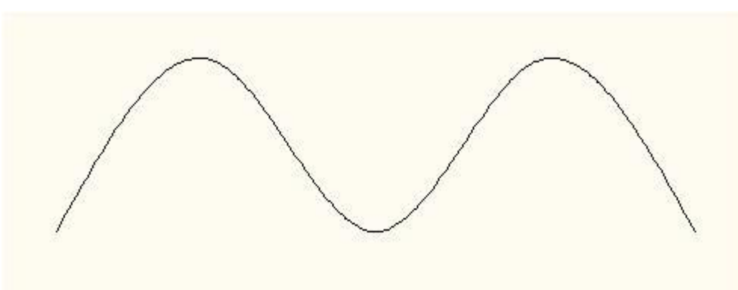


Spline

Keystroke	**Icon**	**location**
SPL		Home > Draw > Spline

In arc, there is only one bending in the curve. If you need more than one bending, then you will need Spline. The command is like that of "3 point arc". Every 3 points will make an arc and it continues as you need.
Example:

- "spl"
- Enter (or click icon)
- Select start point (click on the workspace or enter co-ordinate)
- Specify next point: "5<45
- Specify next point: "5<315
- Specify next point: "5<45
- Specify next point: "5<315
- Specify next point: Enter
- Specify start tangent: Enter
- Specify end tangent: Enter



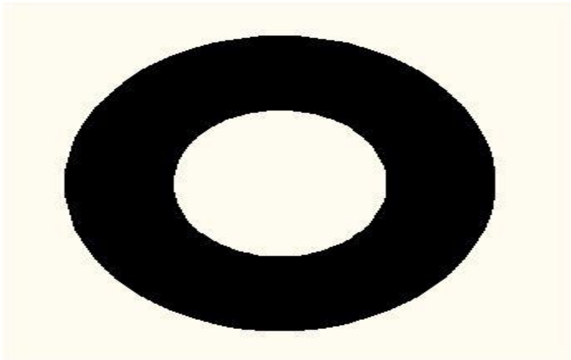
Donut

Keystroke	**Icon**	**location**
DO		Home > Draw > Donut

Constructs a donut like shape of specified radius.
Example:

- "do"

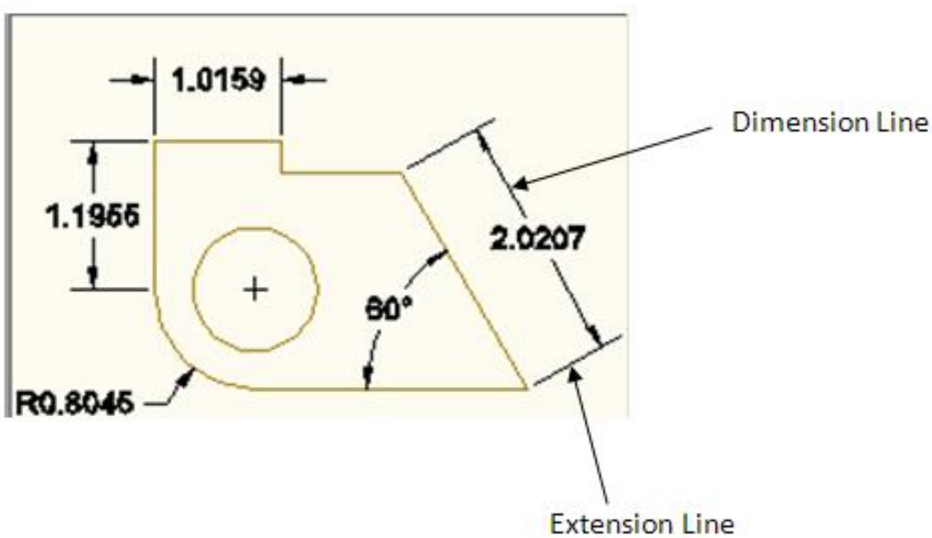
- Enter (or click icon)
- Specify inside diameter of donut: 3
- Specify outside diameter of donut: 6
- Specify center of donut: click any point



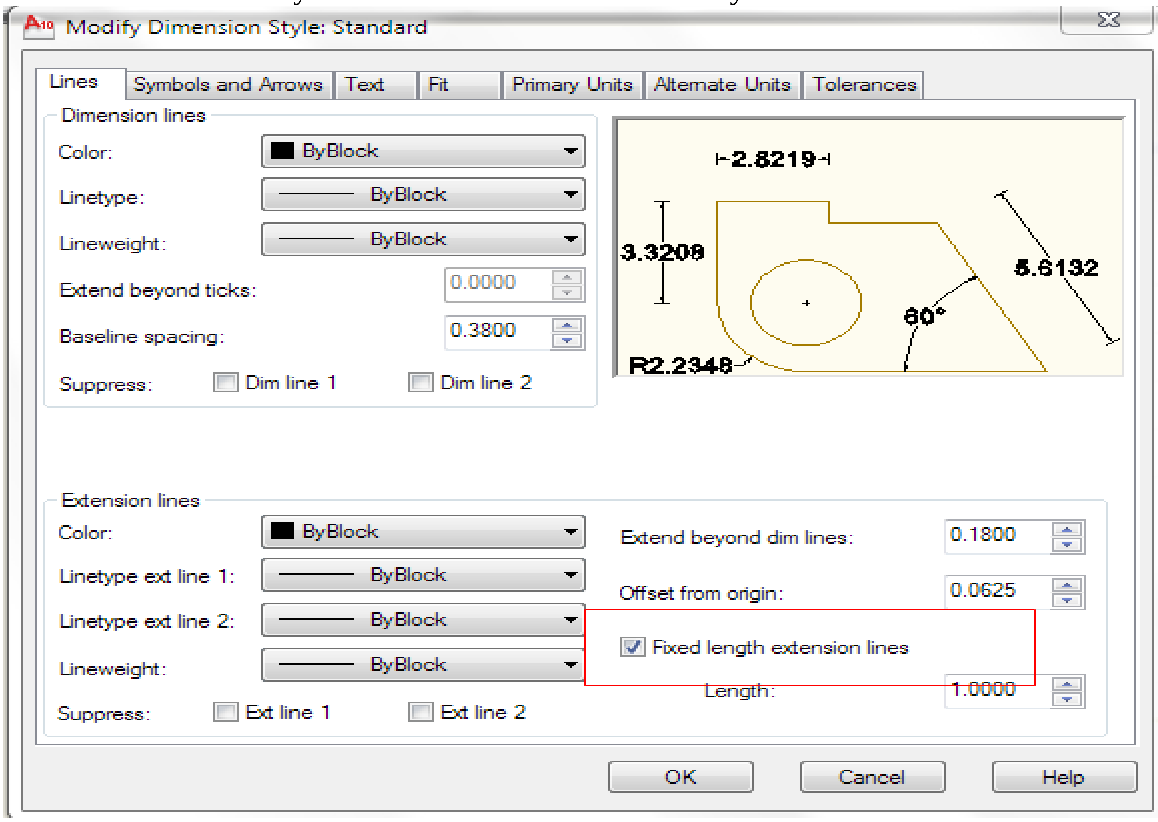
5.1 Detail on Dimension

You now know how to show dimension of your drawing, but there is some thing more you can do to modify the dimension style.
Open **Dimension Style Manager** and go to **Modify** or **Set a new style.**

Line:
The first tab is called line. here you can see that there are two types of lines: Dimension lines & Extension lines.



The dimension lines are the one with arrow heads. you can change their color, line type, line lines; or you can give a fixed lenth by clicking the checkbox.weight etc.if you click the check boxes (Suppress: dim line 1 & dim line 2) the dimension lines will disappear. The extension lines are the lines extended from the desired point on the object up to the dimension line.Modify them in the same manner.you can extent these lines beyond the dim.



Symbols & Arrows:

>Change arrow shape or style and also size of arrow in this tab.
>Choose center mark.
>Modify break size, jog dimension etc.

Text:

Select text style, color,height (or size with respect to your drawing unit).you can also modify text placement and alignment.

Select a reasonable value for text height so that it is consistent with the drawing and easily viewable.

Fit:

This is to fit both text and arrow within the dimension area.Change settings to set up the display as you need.

You can change the placement of text and arrow,scale for dimension to display etc.

Primary units:

Before start drawing, you did a setup for dimension. This time you need to choose the dimension again to display.

In the same manner, select your dimension unit,rounding (if you need) etc.

Alternate units:

If you need to show the dimensions in more than one unit (actually two), then use this tab to set the other unit.

Tolerance:
Suppose you have drawn a line of 10 mm. but in practical field, it can be 10-0.5 mm or 10+0.5 mm.To show this tolerance limit in the drawing sheet, you have to use this tab.

6.1 Statusbar:

The status bar tools are essential parts of AutoCAD drawing. Suppose, you need to draw a line from the midpoint of another line, or endpoint of another line; or you need to draw absolutely straight lines. In these cases you can't do a precise work by eye estimation. The tools in status bar (polar, snap, osnap, ortho etc.) make these selectionsprecisely.

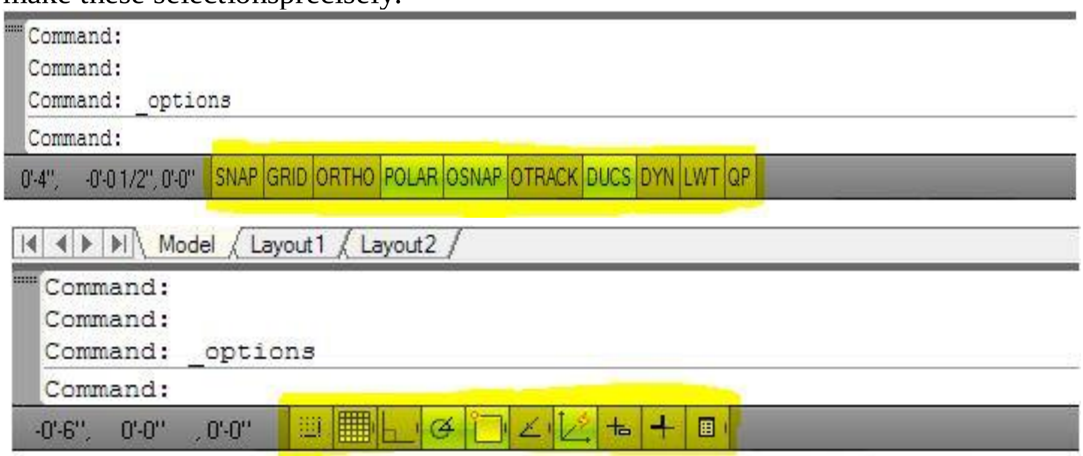
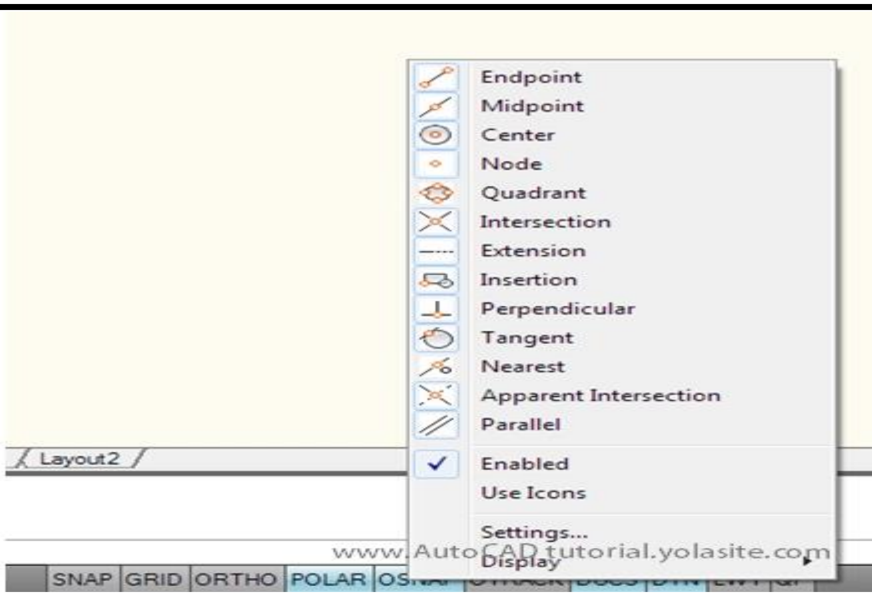


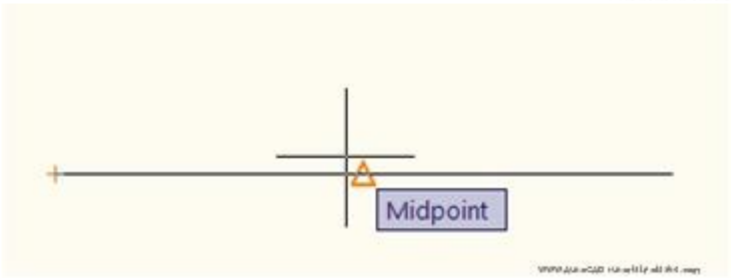
Fig.: using icon view.

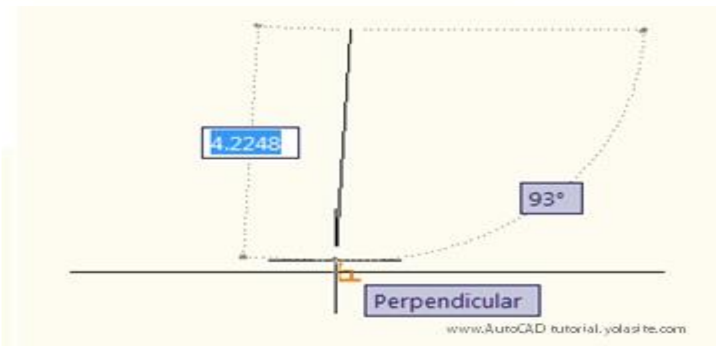
First, let's have a quick look on the functions of the tools:
SNAP: also toggled using the F9 key. When snap on, the cursor under mouse control can only be moved in jumps from one snap point to another.
GRID: also toggled using the F7 key. When set on, a series of grid points appears in the drawing area.
ORTHO: also toggled using the F8 key. When on, lines, etc. can only be drawn vertically or horizontally.
POLAR: also toggled using the F10 key. When set on, a small tip appears showing the direction and length of lines, etc. in degrees and units.
OSNAP: also toggled using the F3 key. When set on, an osnap icon appears at the cursor pick box. Which means an end point, mid point, centre etc. is indicated and attracts the cross hair.
OTRACK: when set on, lines, etc. can be drawn at exact coordinate points and precise angles.
DUCS: Dynamic UCS. Also toggled by the F6 key. Used when constructing 3D solid models.
DYN: Dynamic Input. When set on, the x, y coordinates and prompts show when the cursor hairs are moved.
LWT: when set on, lineweights show on screen. When set off, lineweights only show in plotted/printed drawings.
QP : if any object is selected, its basic properties are shown.

OSNAP: this is probably the most important tool among the ten. Right click on the button and select the points you want to snap on an object.

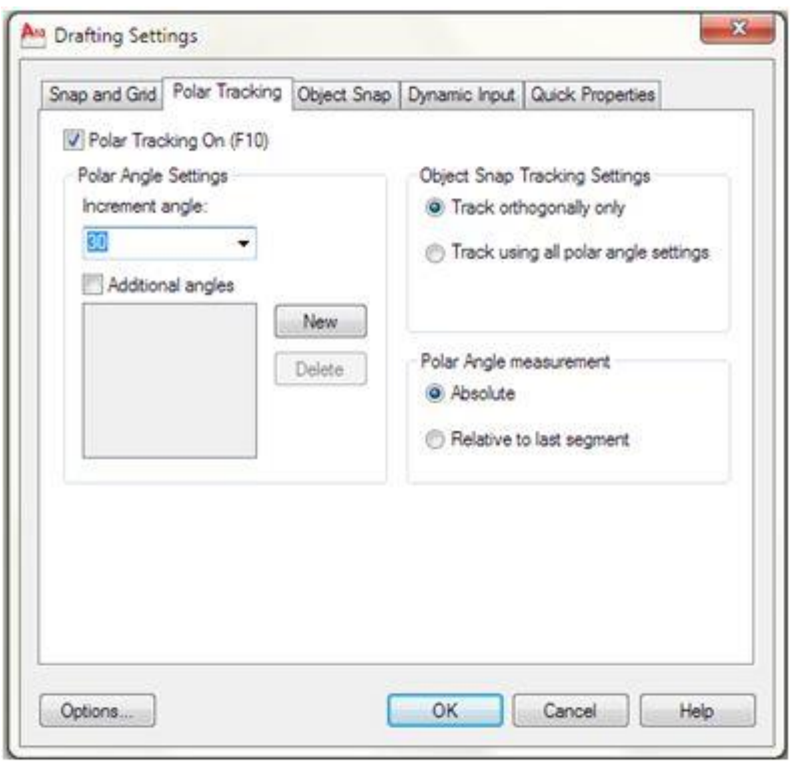


Endpoint - snaps to either the beginning or the end of an object such as a line
Midpoint - snaps to the exact middle of a line or an arc
Center - snaps to the center-point of a circle or arc
Node - snaps to 'nodes'
Quadrant - snaps to any of the four quadrants of a circle
Intersection - snaps to the point where two object cross
Extension - Snaps to the phantom extension of an arc or line
Insertion - snaps to the insertion point of an object (such as a block or text)
Perpendicular - will snap so that the result is perpendicular to line selected
Tangent - snaps to create a line tangent to a circle or arc
Nearest - will find the closest point an object and snap to that point
Parallel -Snaps parallel to a specified line





POLAR: Set it to show a certain angle. Right click on the button and select "**Settings**". Then specify the angle you want to be shown.



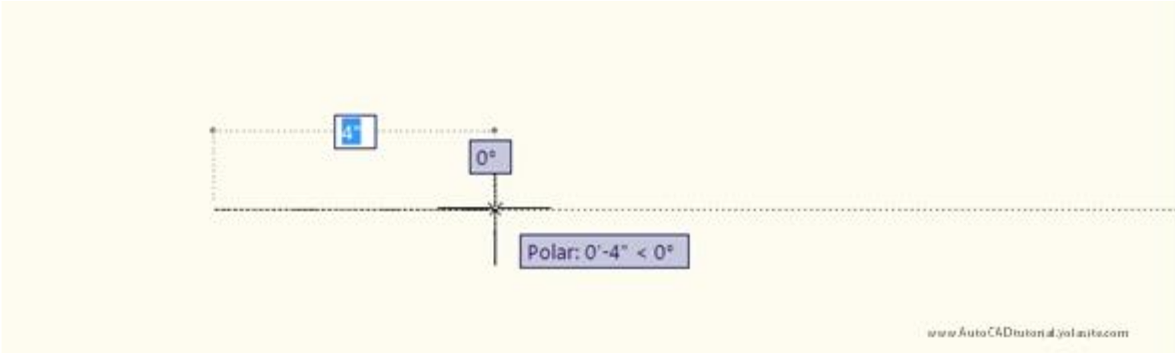
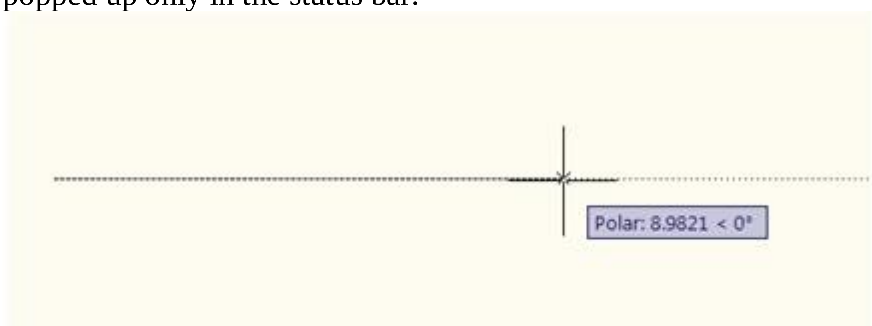
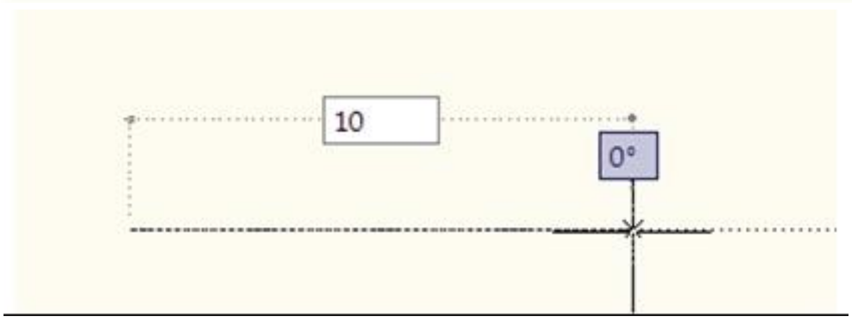


Fig.: 0° angle shown by polar.

In the same manner (right click > settings) set all other buttons you need for a drawing.

DYN: if turned on, your given inputs are seen simultaneously on screen. So helps to avoid wrong command and makes commands easier with less calculation. If turned off, commands are not seen simultaneously, popped up only in the status bar.





In the second image, both line length and angle with horizontal axis are shown. Use "**TAB**" button to toggle between these two value. This will make your commands mucheasier.
(Try the Exercise-3 in lesson-2 with Dynamic Input system)

LWT: you can choose different line width (or line weight) during drawing. But if **LWT**is turned off, no line weights can be seen. Any weight you select will appear as the same as the default weight.

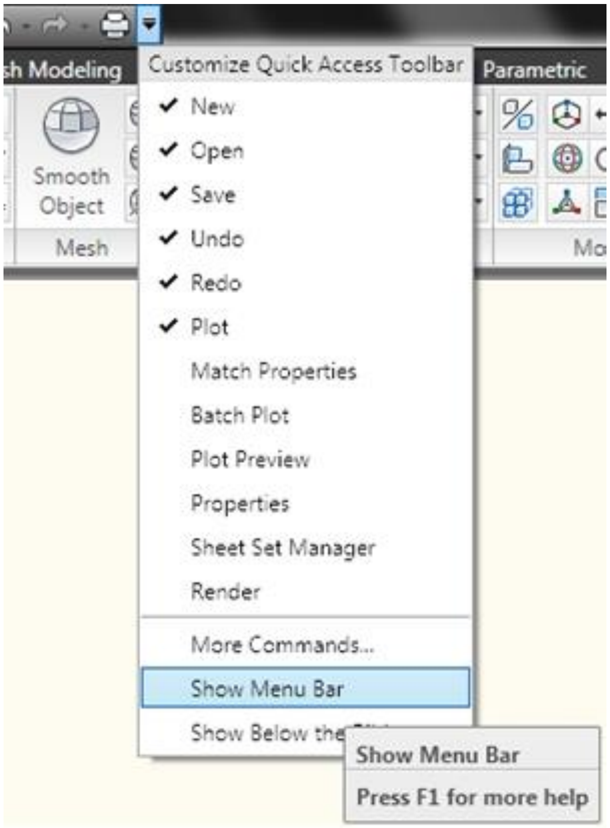
7.1 Pan:

During using AutoCAD you will need to move your workspace several times at different directions, which is called pan. This is done simply by pressing the mouse wheel (the cross hair will turn into a hand) and then moving the workspace as you are grabbing it.

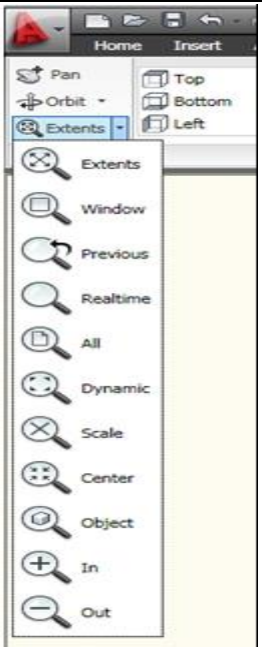
Another way is to clicking the icon below the workspace 

7.2 Zoom:

A very useful and frequently used tool. This is done simply by scrolling the mouse wheel. Scrolling up results zooming in and scrolling down results zooming out. But you will need some other zooming options. To use those click the arrow on **Quick Access toolbar**, select **Show Menu Bar**. On the Menu Bar, click **View** > **Zoom**. You will have the tools like below.



Or go to the "View" tab, on the "Navigate" panel you will find the zooming tools.



Realtime:

normal zoom operation to the whole workspace. Click the left mouse button and drag the ursor downward, it will cuase zoom out.inversely click the left mouse button and drag upward to zoom in. This zoom action is done within the extent of the current view.

Previous:

Shows the previou zoomed view. in this case, up to 10 views are saved.So that the last 10 views can be recalled.

Window:

This option prompts the user to pick two corners of a box on the existing view and enlarges that area to fill the display.

Dynamic:

Permits very quick movement around the drawing. Once selected, this option redraws the graphics area of the screen and displays two rectangles. The larger box shows the extents of the current drawing. The smaller box shows the current view with an "X" in the middle. This moves with the mouse. This view box should be positioned so that its lower left corner is at the lower left corner of the view required. By pressing the left button on the mouse, the "X" is replaced by an "> " pointing to the right side of the view box. This allows you to change the magnification. As the mouse is moved, the view box shrinks and expands so that the size of the required view can be set. The left mouse button toggles between PAN "X" and ZOOM "> " mode so that fine adjustments can be achieved. When the view required has been selected, press <ENTER> or right click to let AutoCAD display it.

Scale:

Creates zoomed view for a given magnification number. If you give a value "n", then a view of n times of the current view will be displayed.So clearly any number below 1 will cause zoom out, and above 1 will zoom in.Now, If "X" is inserted after the number (like 2x) then the factor is applied to the current view. If "XP" is inserted after the scale factor, then the view is scaled relative to paper space.

Center:

This option requires two inputs: a point that is to be the center of the new display and a value to be its new height in drawing units.The existing height is the default for the new height to allow for panning across the drawing. If the new height value is followed by "X" (like 2x), then it is taken as a magnification factor relative to the current height. If followed by "XP", then it is taken as a scale factor relative to paper space and can be used for scaling the contents of paper space viewports.

Object:

zooms only selected objects on the workspace to cover the whole screen. This is useful when you need to work on a specific object.

In:

Clicking this icon will zoom in the drawing by about 50%.

Out:

Clicking this icon will zoom out the drawing by about 50%.

All:

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).

Extents:

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

Material Adjustments

Adding materials to objects greatly increases the realism of a model. In the context of rendering, materials describe how an object reflects or transmits light. Within a material, maps can simulate textures, bump effects, reflections, or refractions.

From the Advanced Render Settings palette, you can turn materials on or off, turn material filtering on or off, and affect how the surfaces of an object are rendered. Materials that you’ve created and attached to objects in the model are normally turned on when you start the rendering process. If you turn them off, all the objects in the model assume the characteristics of the GLOBAL material.

Using Lighting in Rendering

When there are no lights in a scene, the scene is shaded with default lighting. Default lighting is derived from two distant sources that follow the viewpoint as you move around the model. All faces in the model are illuminated so that they are visually discernible. You can control brightness and contrast, but you do not need to create or place lights yourself. When you insert custom lights or add sunlight, you can disable the default lighting. You can apply default lighting to the viewport only.

You add lights to give the scene a realistic appearance. Lighting enhances the clarity and three-dimensionality of a scene. You can create point lights, spotlights, and distant lights to achieve the effects you want (see Figure 4). You can move or rotate them with grip tools, turn them on and off, and change properties such as color and attenuation. The effects of changes are visible in the viewport in real time. Spotlights and point lights are each represented by a different light glyph. Distant lights and the sun are not represented by glyphs in the drawing because they do not have a discrete position and affect the entire scene. You can turn the display of light glyphs on or off while you work. By default, light glyphs are not plotted.

For more precise control over lighting, you can use photometric lights to illuminate your model. Photometric lights are physically correct lights that use photometric values, which enable you to define lights more accurately—as they would be in the real world. You can create lights with various distribution and color characteristics or import specific photometric files available from lighting manufacturers. Photometric lights can use manufacturers' IES standard file format. By using manufacturers' lighting data, you can visualize commercially available lighting in your model. Then you can experiment with different fixtures and, by varying the light intensity and color temperature, you can design a lighting system that produces the results you want.

The sun is a special light similar to a distant light. The angle of the sun is defined by the geographic location that you specify for the model and by the date and time of day that you specify. You can change the intensity of the sun and the color of its light. The sun and sky are the primary sources of natural illumination. With the sun and sky simulation, you can adjust their properties. In the photometric workflow, the sun follows a more physically accurate lighting model in both the viewport and the rendered output. In the photometric workflow, you can also enable sky illumination, which adds soft, subtle lighting effects caused by the lighting interactions between the sun and the atmosphere.
Light fixtures can be represented by embedding photometric lights in blocks that also contain geometry. A luminary assembles a set of light objects into a light fixture.

Light types

Light adjustme

Exp: No: 1

AIM: To develop the given model by using auto cad 2D commands and to specify its Dimension.

SOFTWARE REQUIRED: - AUTOCAD 2010 Database.

COMMANDS IN USE: - LIMITS, ZOOM, LINE, AND DIMLINEAR.

PROCEDURE: - In order to obtain given model the following procedure will be followed…

COMMAND: - Limits:

Specify lower left corner: (0,0)

Specify upper right corner :

(150,100)

Command: ZOOM: [All/Center/Previous/Scale/Window/Object] : All

Command: LINE:

Specify first point: 0, 0

Specify next point or (undo): 100[0^0]

Specify next point or (close/undo): 20[90^0]

Specify next point or (close/undo):

40[180^0] Specify next point or

(close/undo): 120[90^0] Specify next point

or (close/undo):20[180^0] Specify next point

or (close/undo):120[270^0] Specify next

point or (close/undo): 40[180^0] Specify

next point or

(close/undo): C Specify next point or

(close/undo): ESC

Command: DIMLINEAR

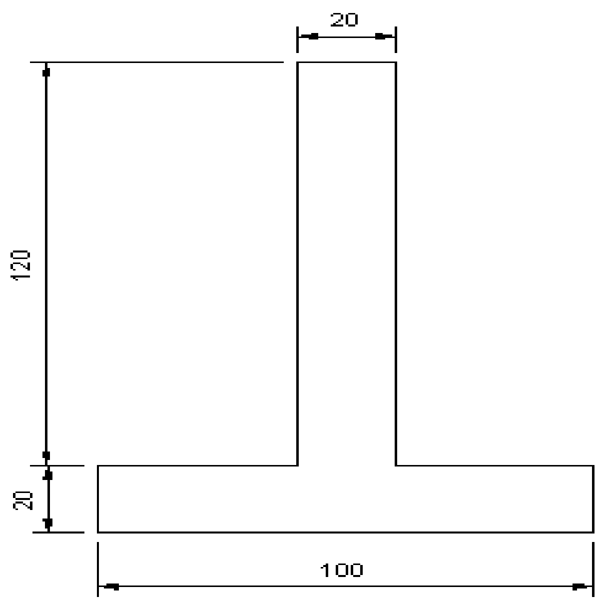
By using this command give dimensions linearly for drawn object to indicate its linear dimensions.

PRECAUTIONS:-

1. Limits should be given before drawing the object.
2. Object should be drawn from a specific point of location only.
3. Ensure that proper sequence should be followed to draw an object.

RESULT:

Hence by using auto cad 2010 2D commands we have drawn the object model and Dimensions are specified.



EXP NO: 2

AIM: To develop the given model by using auto cad 2D commands and to specify its Dimension.

SOFTWARE REQUIRED: - AUTOCAD 2010 Database.

COMMANDS IN USE: - LINE, CIRCLE, DIMLINEAR, DIMDIA.

PROCEDURE: - In order to obtain given model the following procedure Will be followed.

COMMAND: - Limits:

Specify lower left corner: (0,0) Specify

upper right corner : (150,150)

Command: ZOOM:[All/Center/Previous/Scale/Window/Object] : All

Command: LINE:

Specify first point: 0, 0

Specify next point or (undo): 150[0^0] Specify

next point or (close/undo): 150[90^0] Specify

next point or (close/undo): 150[270^0] Specify

next point or (close/undo): C Specify next point

or (close/undo): ESC

Command: CIRCLE

Specify centre point for circle (3p/2p/ttr): 30,30

Specify radius of circle or (diameter): d Specify

diameter of the circle: 10

Command: CIRCLE

Specify centre point for circle (3p/2p/ttr): 120, 30

Specify radius of circle or (diameter): d

Specify diameter of the circle: 10

Command: CIRCLE

Specify centre point for circle (3p/2p/ttr):

120,120 Specify radius of circle or (diameter): d

Specify diameter of the circle: 10

Command: CIRCLE

Specify centre point for circle (3p/2p/ttr):

30,120 Specify radius of circle or (diameter): d

Specify diameter of the circle: 10

Command: DIMLINEAR

By using this command give dimensions linearly for drawn object to indicate its

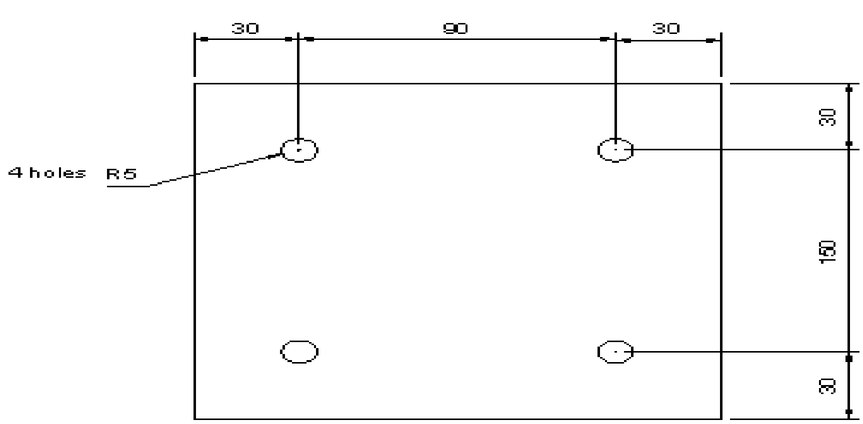
Linear dimensions.

PRECAUTIONS:-

1. Limits should be given before drawing the object.
2. Object should be drawn from a specific point of location only.
3. Ensure that proper sequence should be followed to draw an object.

RESULT:

Hence by using auto cad 2006 2D commands we have drawn the object model and Dimensions are specified.



EXP NO: 3

AIM: To develop the given model by using auto cad 2D commands and to specify its Dimension.

SOFTWARE REQUIRED: - AUTOCAD 2010 Database.

COMMANDS IN USE: - LIMITS,ZOOM,LINE, DIMLINEAR.

PROCEDURE: - In order to obtain given model the following procedure will be followed.

First draw the axis line(center line),X and Y ,

COMMAND:LINE

COMMAND: LIMITS

Specify lower left corner: 0, 0 Specify

upper left corner: 300,300

ZOOM:[All/Center/Previous/Scale/Window/Object] : All

Command: LINE.

Specify the first point: 0,0

Specify the next point: @150<0

Specify the next point: @40<90

Specify the next point: @20<180

Specify the next point:@20<270

Specify the next point:@110<180

Specify the next point:@30<90

Specify the next point:@20<180

Specify the next point: C

Create chamfer for inner side corners.

Command: CHAMFER

Select first line or (poly line/distance/angle/trim/method):d

Specify first chamfer distance:10 mm

Specify the second chamfer distance:10 mm

Select two corner lines.

Create fillet for outsides corners.

Command: FILLET

Select first object or [undo/poly line/radius/trim/multiple]: r

Specify fillet radius: 10

Select two corner lines

Command: MIRROR

Select objects

Specify the first point of the mirror line:

Specify the second point of mirror line:

Give enter

Delete source objects?(yes/no):n

Command: DIMLINEAR

By using this command give dimensions linearly for drawn object to indicate it's

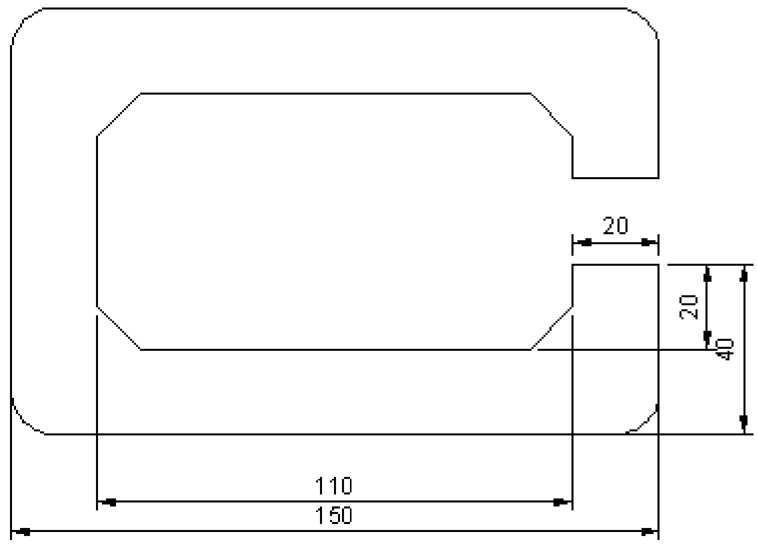
Linear dimensions.

PRECAUTIONS:-

1. Limits should be given before drawing the object.
2. Object should be drawn from a specific point of location only.
3. Ensure that proper sequence should be followed to draw an object.

RESULT:

Hence by using auto cad 2010 2D commands we have drawn the object model and Dimensions are specified.



EXP NO: 4

AIM: To draw the isometric drawings by using AutoCAD 2010 3D commands and the Dimensions can be determined by counting the no of grids.

SOFTWARE REQUIRED: - AutoCAD 2010 Database.

COMMANDS USED: LIMITS, ZOOM, LINE, DIMLINEAR.

PROCEDURE: Highlight the grid option.

COMMAND: LIMITS

Specify lower left corner: 0, 0 Specify
upper left corner: 300,300

ZOOM:[All/Center/Previous/Scale/Window/Object] : All

Draw Axis lines(X and Y)

Command: CIRCLE

Specify centre point for circle (3p/2p/ttr): 0,0

Specify radius of circle or (diameter): d

Specify diameter of the circle: 36 mm

Command: CIRCLE

Specify centre point for circle (3p/2p/ttr): 0,0

Specify radius of circle or (diameter): r

Specify radius of the circle: 30 mm

Command: CIRCLE

Specify centre point for circle

(3p/2p/ttr): 0,125 Specify radius of circle or
(diameter): d

Specify diameter of the circle: 12 mm

Command: CIRCLE

Specify centre point for circle (3p/2p/ttr): 0,125

Specify radius of circle or (diameter): r

Specify radius of the circle: 10 mm.

Command: ARC

Specify start point of arc or [center]: C

Specify the center point of arc: 64

Specify start point and end point of the arc.

Draw the inclined axis line with dimension of 30°.

Command: CIRCLE

Specify centre point for circle (3p/2p/ttr): 0, 64

Specify radius of circle or (diameter): d

Specify diameter of the circle: 12 mm

Command: ARRAY

Select the polar array

Select the object (12 mm circle)

Select the center point of the base circle and give number of items 2 and angle 30° enter.

Command: FILLET

Select first object or [undo/poly line/radius/trim/multipule]: r

Specify fillet radius: 12

Select two circles.

Command: FILLET

Select first object or [undo/poly line/radius/trim/multiple]: r

Specify fillet radius: 54

Select two circles and enter.

Command: TRIM

Give enter and click unknown lines, arcs. Etc.

Command: MIRROR

Select the mirroring objects and enter

Specify the first point of the mirror line: click start point of the vertical axis line

Specify the second point of the mirror line: click second point of the vertical axis line.

Command: DIMLINEAR

By using this command give dimensions linearly for drawn object to indicate it’s

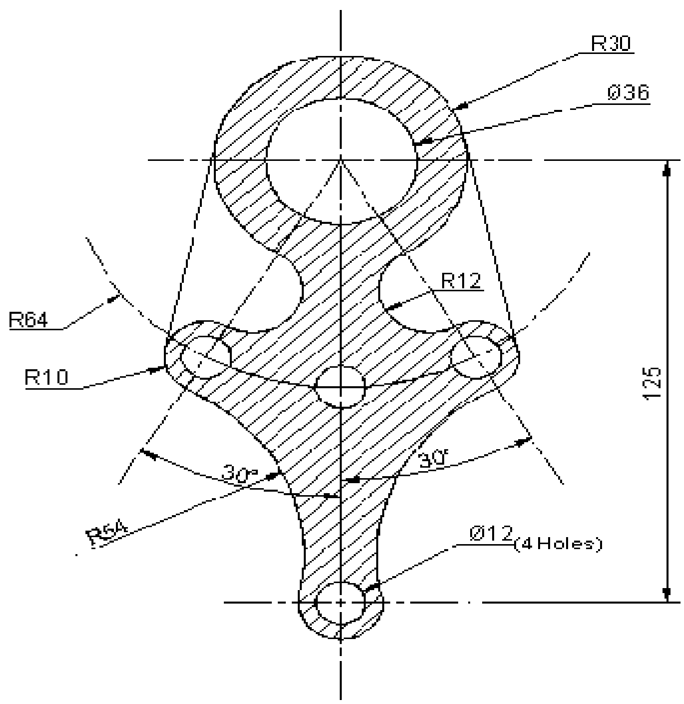
Linear dimensions.

PRECAUTIONS:-

1. Limits should be given before drawing the object.
2. Object should be drawn from a specific point of location only.
3. Ensure that proper sequence should be followed to draw an object.

RESULT:

Hence by using auto cad 2010 2D commands we have drawn the object model and Dimensions are specified.



EXP NO: 5

AIM: To draw the isometric drawings by using AutoCAD 2010 3D commands and the Dimensions can be determined by counting the no of grids.

SOFTWARE REQUIRED: - AutoCAD 2010 Database.

COMMANDS USED: LIMITS, ZOOM, LINE, DIMLINEAR.

PROCEDURE: Highlight the grid option.

COMMAND: LIMITS

Specify lower left corner: 0, 0 Specify

upper left corner: 300,300

ZOOM: [All/Center/Previous/Scale/Window/Object] : All

Command: LINE.

Polar: on setting 30 ^{c}
Specify the first point: 0,0
Specify the next point: 16[90^0]
Specify the next point: 24[150^0]
Specify the next point: 16[90^0]
Specify the next point: 24[150^0]
Specify the next point: 16[90^0]
Specify the next point: 24[150^0]
Specify the next point: 48[-90^0]
Specify the next point: c
Specify the first point: 0, 0
Specify the next point: 80[30^0]
Specify the next point: 16[90^0]
Specify the next point: 24[150^0]
Specify the next point: 16[90^0]
Specify the next point: 24[150^0]
Specify the next point: 16[90^0]
Specify the next point: 24[150^0]
Specify the next point: C
Join the all edges.

Command: DIMLINEAR

By using this command give dimensions linearly for drawn object to indicate it's

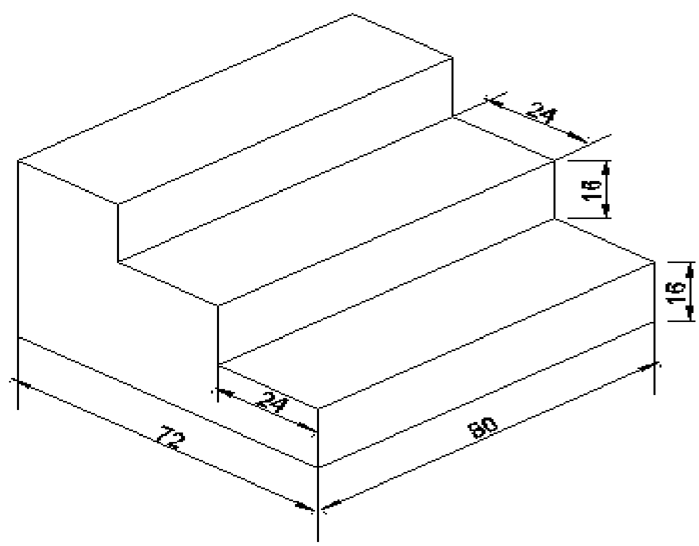
Linear dimensions.

PRECAUTIONS:-

1. Limits should be given before drawing the object.
2. Object should be drawn from a specific point of location only.
3. Ensure that proper sequence should be followed to draw an object.

RESULT:

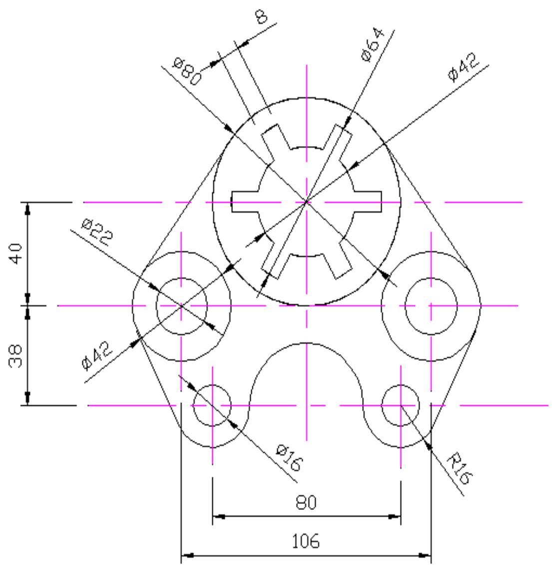
Hence by using auto cad 2010 3D commands we have drawn the object model and Dimensions are specified.



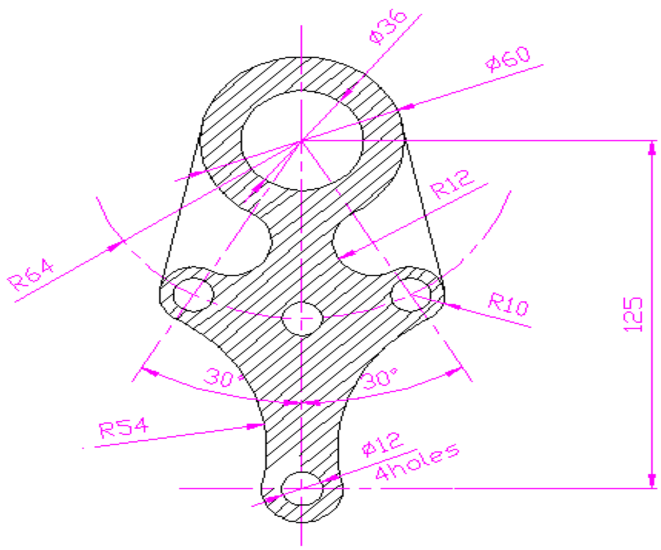
Extra exercises:

2D Sketcher practicing general components

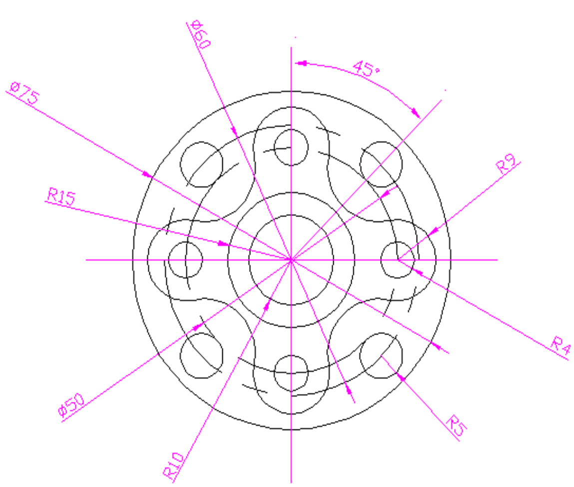
1. Draw a 2D view of object cover



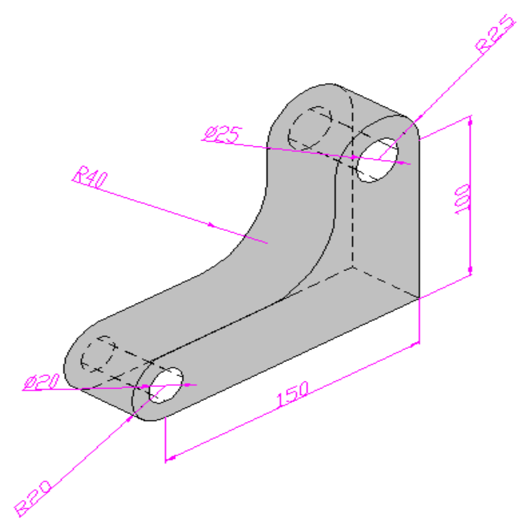
2.Draw a 2D view of Suspension



3. Draw a 2D view of Object-1



4.Draw a 3D view of Link



Solid Works

Basics of Solids Modeling with Solid Works

Introduction

Solid Works is the state of the art in computer-aided design (CAD). Solid Works represents an object in a virtual environment just as it exists in reality, i.e., having volume as well as surfaces and edges. This, along with exceptional ease of use, makes Solid Works a powerful design tool. Complex three-dimensional parts with contoured surfaces and detailed features can be modeled quickly and easily with Solid-Works. Then, many parts can be assembled in a virtual environment to create a computer model of the finished product. In addition, traditional engineering drawings can be easily extracted from the solids models of both the parts and the final assembly. This approach opens the door to innovative design concepts, speeds product development, and minimizes design errors. The result is the ability to bring high-quality products to market very quickly.

CONSTRAINT-BASED SOLIDS MODELING

The constraint-based solids modeling used in Solid Works makes the modeling process intuitive. The 3-D modeling begins with the creation of a 2-D sketch of the profile for the cross section of the part. The sketch of the cross section begins much like the freehand sketch of the face of an object. The initial sketch need not be particularly accurate; it needs only to reflect the basic geometry of the part's cross-sectional shape. Details of the cross section are added later. The next step is to constrain the two-dimensional sketch by adding enough dimensions and parameters to completely define the shape and size of the two-dimensional profile. The name constraint-based modeling arises because the shape

of the initial two-dimensional sketch is "constrained" by adding dimensions to the sketch. Finally, a three-dimensional object is created by revolving or extruding the two-dimensional sketched profile. Figure 1 shows the result of revolving a simple L-shaped cross section by 270° about an axis and extruding the same L-shaped cross section along an axis.

In either case, these solid bodies form the basic geometric solid shapes of the part. Other features can be added subsequently to modify the basic solid shape. Once the solids model is generated using Solid Works, all of the surfaces have been automatically defined, so it is possible to shade it in order to create a photorealistic appearance. It is also easy to generate two-dimensional orthographic views of the object. Solid modeling is like the sculpting of a virtual solid volume of material. Because the volume of the object is properly represented in a solids model, it is possible to slice through the object and show a view of the object that displays the interior detail (sectional views). Once several solid objects have been created, they can be assembled in a virtual environment to confirm their fit and to visualize the assembled product. Solids models are useful for purposes other than visualization. The solids model contains a complete mathematical representation of the object, inside and out. This mathematical representation is easily converted into specialized computer code that can be used for stress analysis, heat transfer analysis, fluid-flow analysis, and computer-aided manufacturing. Getting Started. In Solid Works Introduction and Reference Solid Works Corporation developed Solid Works as a three-dimensional, feature-based, solids-modeling system for personal computers.

Solid modeling represents objects in a computer as volumes, rather than just as collections of edges and surfaces. Features are three-dimensional geometries with direct analogies to shapes that can be machined or manufactured, such as holes or rounds. Feature-based solid modeling creates and modifies the geometric shapes of an object in a way that represents common manufacturing processes. This makes Solid Works a very powerful and effective tool for engineering design. As with other computer programs, Solid Works organizes and stores data in files. Each file has a name followed by period (dot) and an extension. There are several file types used in Solid Works, but the most common file types and their extensions are

Part files .prt or .sldprt

Assembly files .asm or .sldasm

Drawing files .drw or .slddrw

Part files are the files of the individual parts that are modeled. Part files contain all of the pertinent information about the part. Because Solid Works is a solids-modeling program, the virtual part on the screen will look very similar to the actual manufacture part. Assembly files are created from several individual part files that are virtually assembled (in the computer) to create the finished product.

awing files are the two dimensional engineering drawing representations of both the part and assembly file. The drawings should contain all of the necessary information for the manufacture of the part, including dimensions, part tolerances, and so on. The part file is the driving file for all other file types. The modeling procedure begins with part files. Subsequent assemblies and drawings are based on the original part files. One advantage of Solid Works files is the feature of dynamic links. Any change to a part file will automatically be updated in any corresponding assembly or drawing file.

Tool bars:

The Sketch toolbar contains tools to set up and manipulate a sketch.

• The Sketch Tools toolbar contains tools to draw lines, circles, rectangles, arcs, and so on.

• The Sketch Relations toolbar contains tools for constraining elements of a sketch by using dimensions or relations.

The Features toolbar contains tools that modify sketches and existing features of a part.

• The Standard toolbar contains the usual commands available for manipulating files (Open, Save, Print, and so on), editing documents (Cut, Copy, and Paste), and accessing Help.

The Standard Views toolbar contains common orientations for a model.

• The View toolbar contains tools to orient and rescale the view of a part.

in Solid Works Introduction and Reference Solid Works Corporation developed Solid Works as a three-dimensional, feature-based, solids-modeling system for personal computers.

Solid modeling represents objects in a computer as volumes, rather than just as collections of edges and surfaces. Features are three-dimensional geometries with direct analogies to shapes that can be machined or manufactured, such as holes or rounds. Feature-based solid modeling creates and modifies the geometric shapes of an object in a way that represents common manufacturing processes. This makes Solid Works a very powerful and effective tool for engineering design. As with other computer programs, Solid Works organizes and stores data in files. Each file has a name followed by a period (dot) and an extension. There are several file types used in Solid Works, but the most common file types and their extensions are Part files .prt or .sldprt,Assembly files .asm or .sldasm,Drawing files .drw or .slddrw Part files are the files of the individual parts that are modeled. Part files contain all of the pertinent information about the part. Because Solid Works is a solids-modeling program, the virtual part on the screen will look very similar to the actual manufacture part. Assembly files are created from several individual part files that are virtually assembled (in the computer) to create the finished product. Awing files are the two dimensional engineering drawing representations of both the part and assembly file. The drawings should contain all of the necessary information for the manufacture of the part, including dimensions, part tolerances, and so on. The part file is the driving file for all other file types. The modeling procedure begins with part files. Subsequent assemblies and drawings are based on the original part files. One advantage of Solid Works files is the feature of dynamic links. Any change to a part file will automatically be updated in any corresponding assembly or drawing file.

Translation to SolidWorks

One huge advantage that SolidWorks has over most CAD software is the ability to open nearly every CAD file type on the market today. Its automatic translation tools allow you to convert 2D drawings or 3D solids into SolidWorks, quickly define them, and then use them for your own purposes. This ability can save a lot of time and give you an advantage over your competition in that you don't have to ask your customer to provide data in a specific format to be able to use it.

Data Translation

SolidWorks 2011 features built-in translators that let you exchange CAD data created in a wide variety of software applications and file formats. Below is a list of the file types that SolidWorks can open:

File Type(s)	File Extensions
SolidWorks Part Files	.prt, .sldprt
SolidWorks Assembly Files	.asm, .sldasm
SolidWorks Drawing Files	.drw, .slddrw
DXF	.dxf>
DWG	.dwg
Parasolid	.x_t, .x_b, .xmt_txt, .xmt_bin
IGES	.igs, .iges
STEP	.step, .stp
ACIS	.sat
VDAFS	.vda

VRML	.wfl
STL	.stl
Catia Graphics	.cgr
Pro/Engineer Part	.prt, .prt.*, .xpr
Pro/Engineer Assembly	.asm, .asm.*, .xas
Unigraphics II	.prt
Inventor Part	.ipt
Solid Edge Part	.par, .psm
Solid Edge Assembly	.asm
CADKEY	.prt, ckd
Add-Ins	.dll
IDF	.emn, .brd, .bdf, .idf
Rhino	.3dm
Adobe Photoshop	.psd
Adobe Illustrator	.ai
Library Feature Part	.lfp, .sldlfp
Template	.prtdot, .asmdot, .drwdot

CAMERA BODY

AIM:

To model the given object using the Extrusion feature as per the dimensions given.

Description of Extrusion Future:

Base Feature:

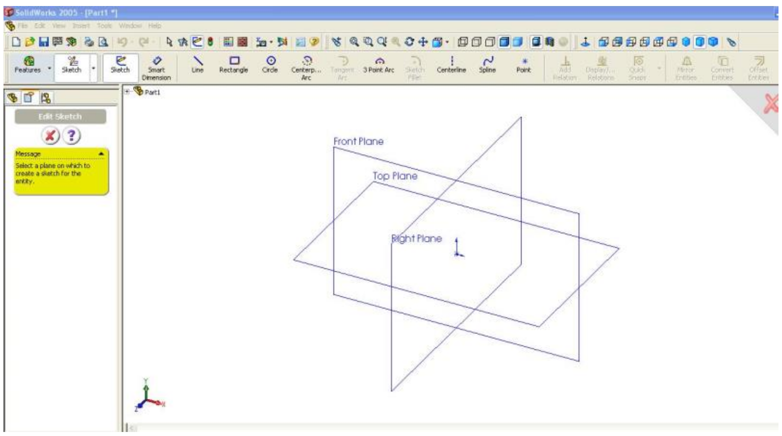
The first feature that is created.

The foundation of the part.

The base feature geometry for the box is an extrusion.

To create an Extruded Base Feature:

1. **Select a sketch plane.**



1. **Sketch a 2D profile of the square model as shown in the figure**

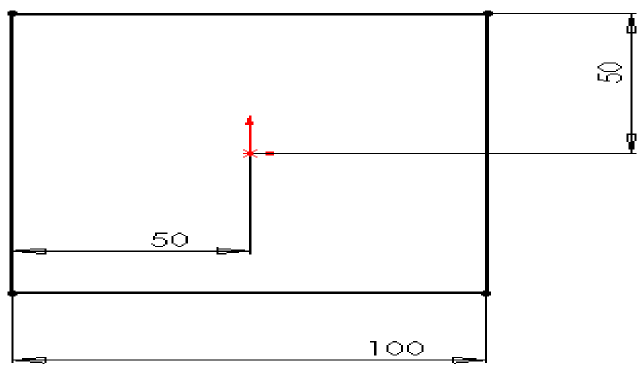
Select front plane →

Right click → select insert sketch→

Sketch a 2D profile of the square model →

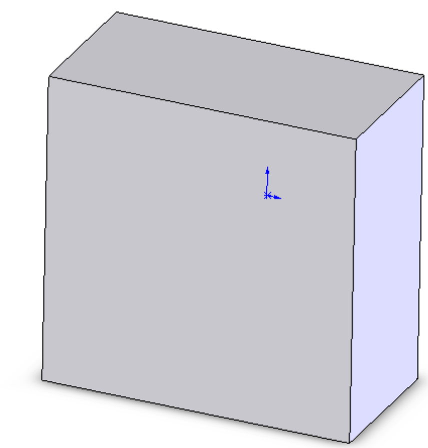
Give the dimension and it should come fully define →

ok → exit sketch



2. Extrude the sketch perpendicular to sketch plane

(Select → Insert → Boss/Bass → Extrude) → ok



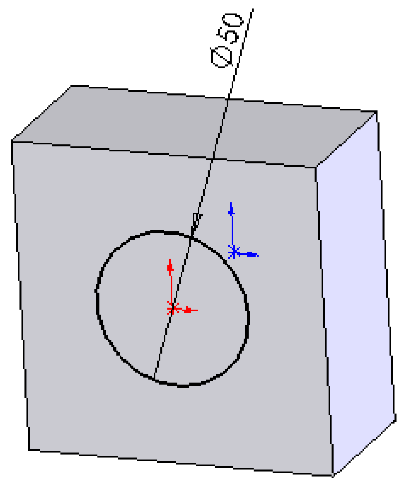
3. Draw a circle of Diameter 50 mm,

Select front plane to draw the 2D circle of dia 50 >

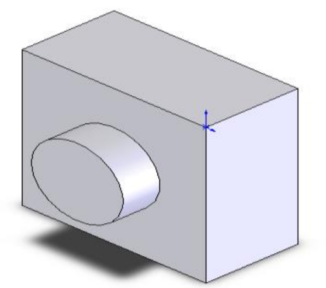
Give the dimension and it should come fully define >

Ok → exit sketch

It adds material to the part and requires a sketch.



1. **Extrude the sketch perpendicular to sketch plane**
   Select the sketch and then (select →Insert → Boss/Bass →Extrude→ok



4. Draw the circle for the Diameter of 35 mm and

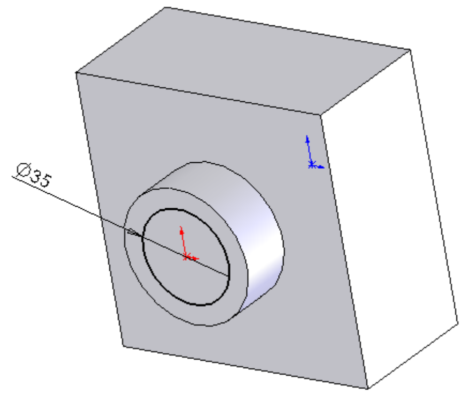
Select front plane to draw the 2D circle of dia 35 >

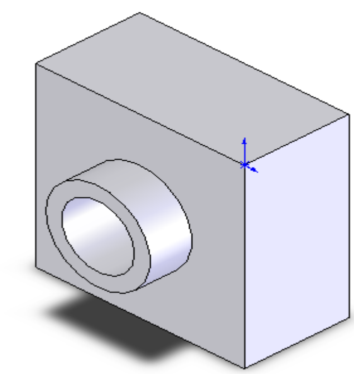
Give the dimension it should come fully define >

Ok > exit sketch

(Select →Insert →cut →Extruded)Feature:

It Removes material from the part and also it requires a sketch.

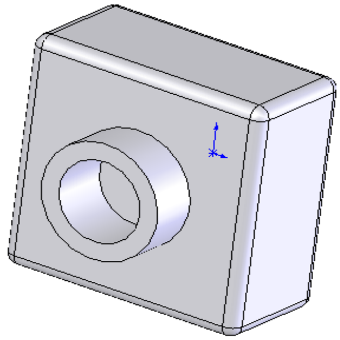




5. Fillet Feature:

Rounds the edges of faces of a part to a specified radius.

(→Select→insert→Feature→fillet)



Procedure:

1. Select a sketch plane(Front,top or side)
2. Sketch a 2D profile of the model.
3. Dimention the model using smart Dimensions icon.
4. Check the sketch is fully defined
5. Extrude the sketch perpendicular to sketch plane.
6. Use extrude cut feature to cut the solid as given in the drawing.

Result:

Thus the given model is extruded.

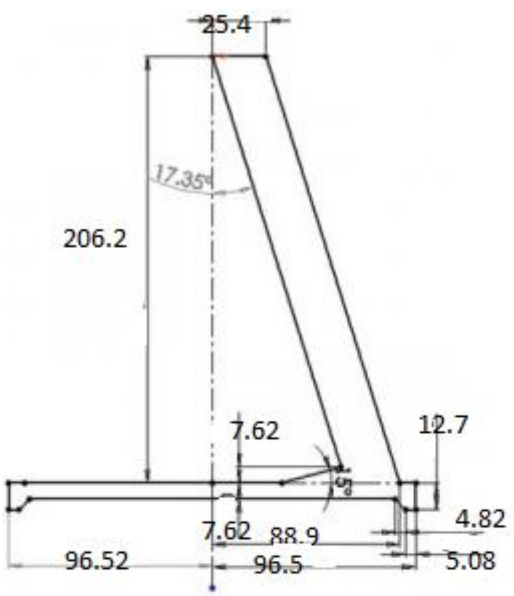
AUTOMOBILE WHEEL

AIM:

To model the given object using the Revolve and circular pattern feature as per the dimension given.

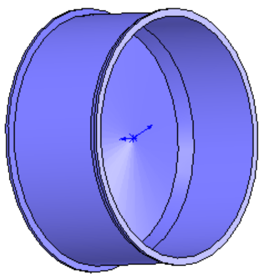
1. Create a 2D sketch on Front Plane as shown in the figure.

   (Right click the Front plane>insert sketch and draw the 2D sketch)



Note: All the 2D sketches drawn should be fully Defined and there should not be any under defined) and use (click Add Relation and Smart Dimensions)

2. Revolve, the sketch to 360 degree on top sketched line, by (Insert> Boss/Base>Revolve) ok.

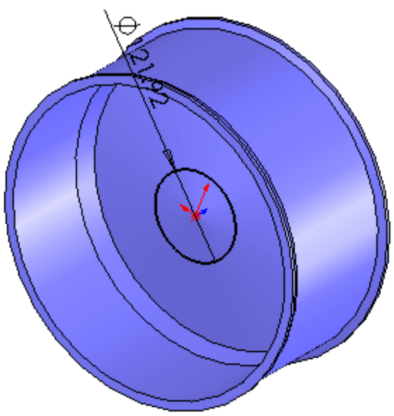


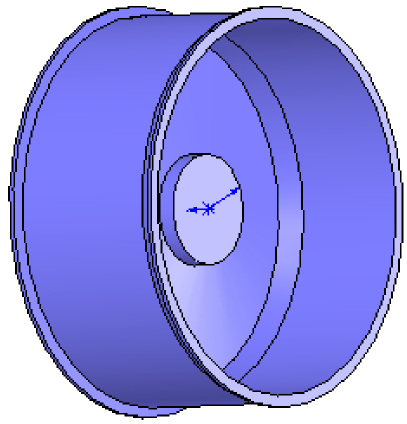
3. Create circle of 2D sketch of Diameter of 121.92 mm, on right plane and extrude to 50.8mm

   (Select the face by (Enter Space bar> double click the Normal plane)

   And Draw the 2D sketch as given above

   Extrude by (Insert>Boss/Base>Extrude)) ok.

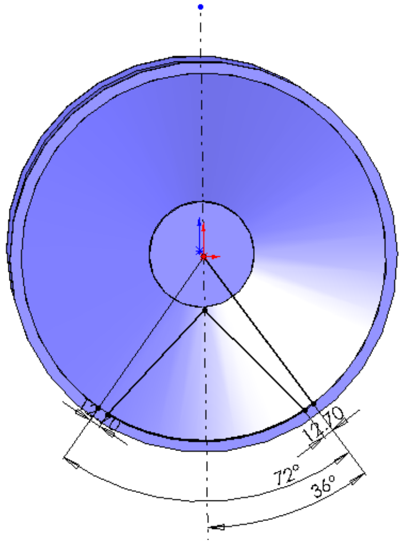




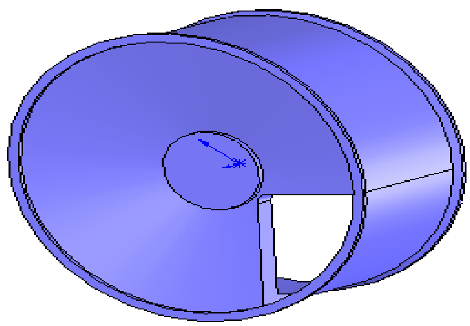
4. Draw the sketch on edge wheel face, sketch for arm hole ,

   Select the face by (Enter Space bar> double click the Normal plane)

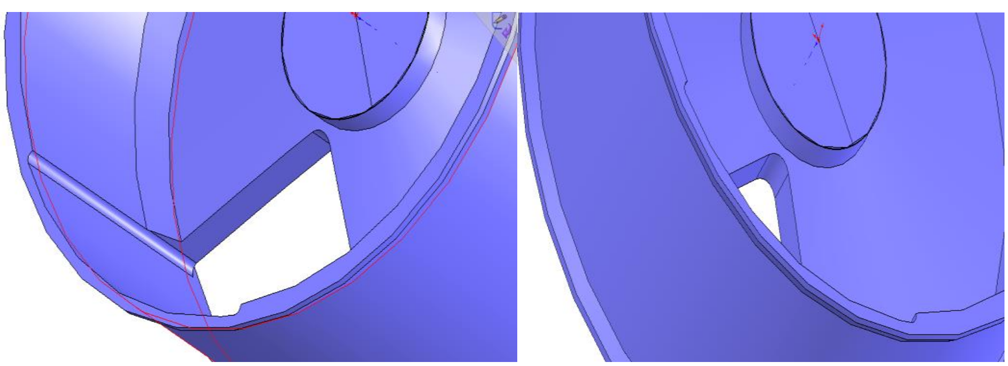
   And Draw the 2D sketch as given below



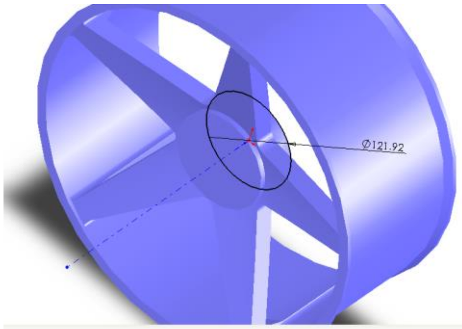
5. And remove the material by (Insert>cut>Extrude), through all, OK.



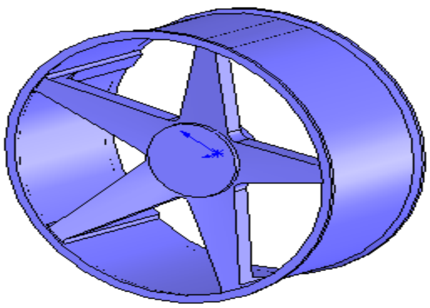
6. Add fillet R12.7mm inner(Insert>Features>Fillet/Round),add fillet 5.08mm for corner ok



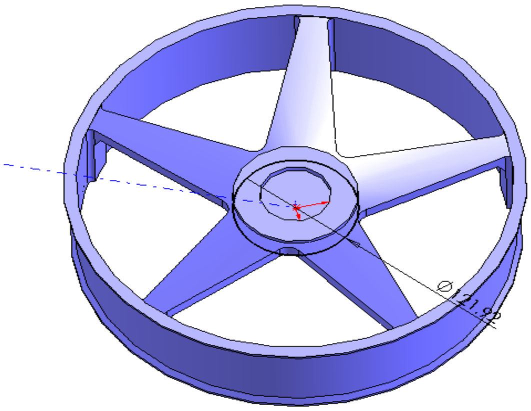
7. Click Circular Pattern ,click (View>Temporary Axes,) select center axis as rotation axis



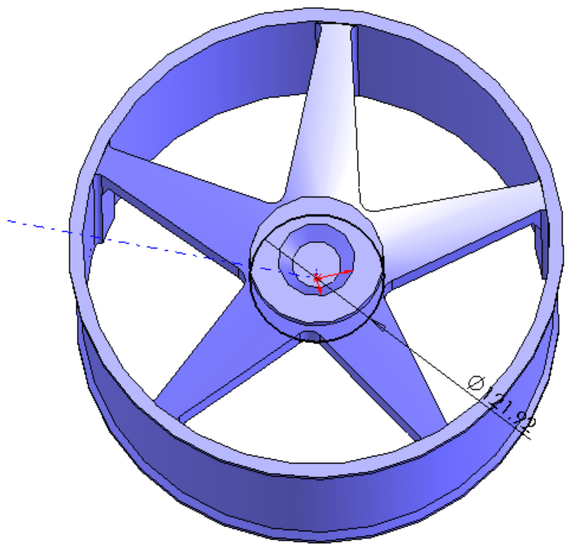
8.(Give 360 degree and 5 equal spacing) , Select Cut-Extrude1, Fillet1 and Fillet2 as a Features to Pattern. OK.



8. Click on hub face, insert sketch, sketch center circle diameter 69.85mm. Extrude Cut to 12.7mm deep.(Insert>cut>Extrude).



9. Add chamfer 12.7mm to inner cut and add chamfer 6.35mm to wheel edge ok done. (Insert>Feature>chamfer).



Result:

Thus the given model is completed.

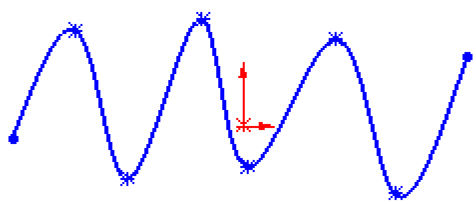
THREE LAYER ROPE

AIM:

To model the given object using the Sweep and circular step and repeat feature as per the dimensions given**.**

1. Create a spline curve sketch on Front Plane.

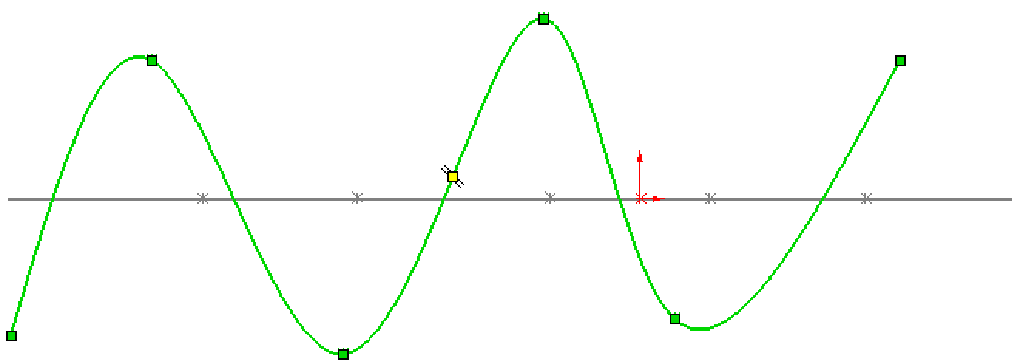
(Front plane→Insert Sketch→Draw the spline using spline tool bar →right click→end spline)



(Select →close dialog box and→exit sketch.)

2. And another spline curve sketch on top plane

(select →Top plane→Insert Sketch→click Space bar and double click the Normal plane →Draw the spline using spline tool bar→right click→end spline)

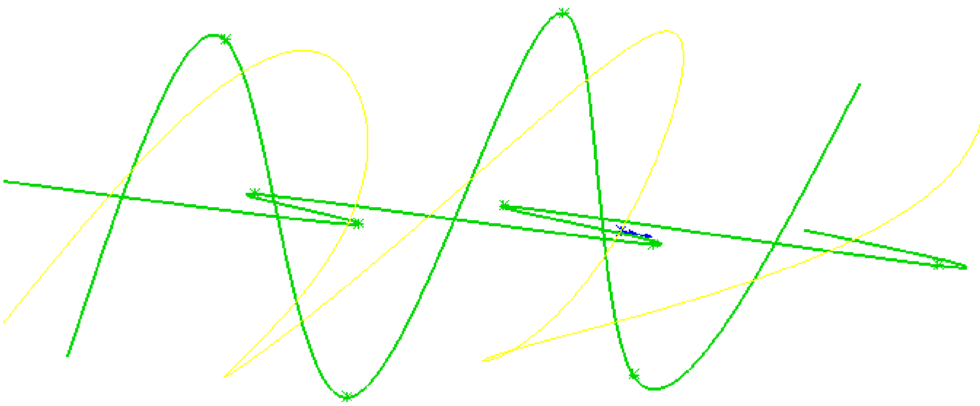


(Select →close dialog box and→exit sketch.)

3. Create 3D plane curve

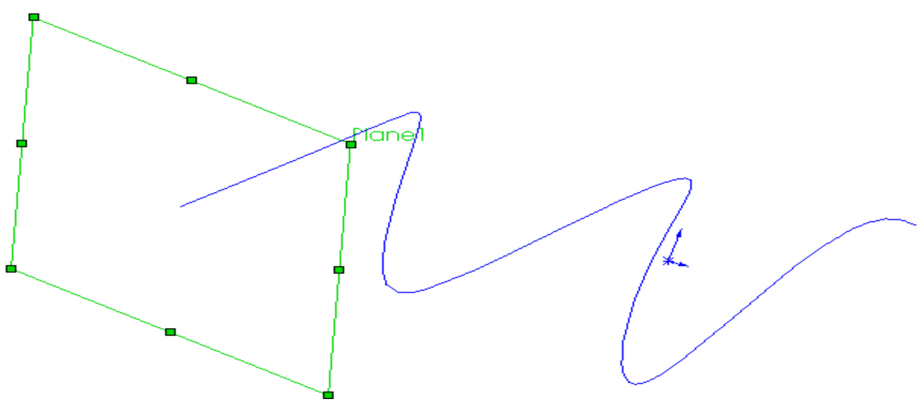
(Using, insert→ curve→projected, select Two (cures) sketches,

When we rotate the geometry it will show 3D curve. → Click OK.

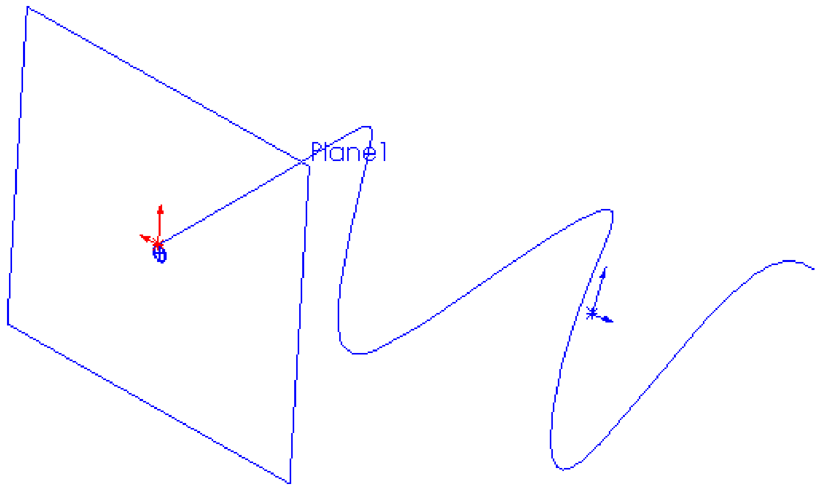


4.at one end of the 3D curve, create new plane.

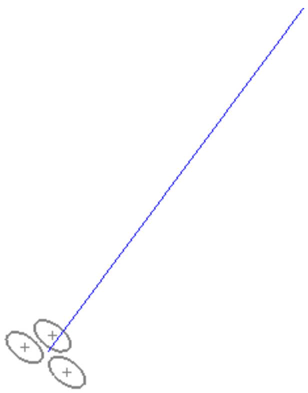
(Insert→Reference Geometry→plane),select curve end point and edge curve and OK.



5. Create a circle with suitable dimensions, nearby end of the spline curve.



6. Before finishing sketch (select →Tools→sketch tools→circular step and repeat) from that given number of rope layers.(3 or 4). OK and select finish sketch.



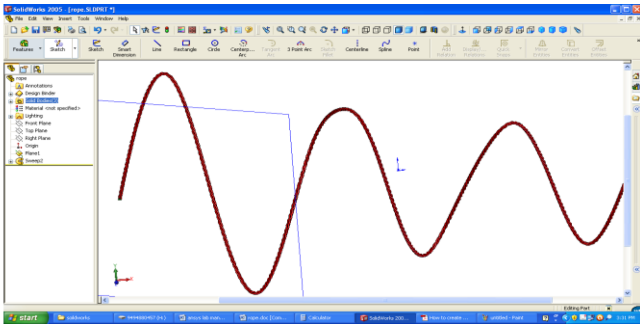
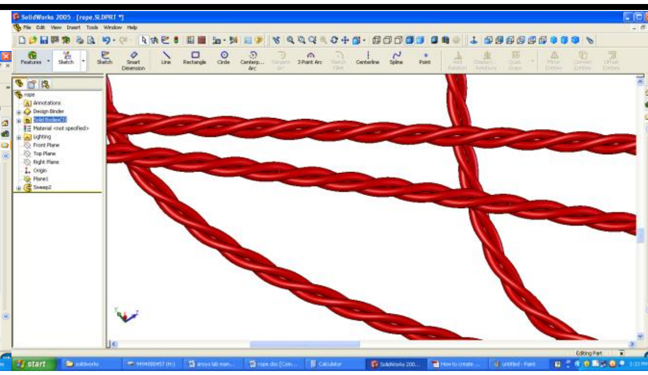
7. Take sweep (insert→boss/base →sweep) command and give

Select profile and there relative circle, and

Select path there relative curve,

Options→orientation /twist type (select →along path),

Define by→select turns →give the valve of 50 to 100). →Ok done.

Result:

Thus the given model is completed.

<u>**Automobile Spring**</u>

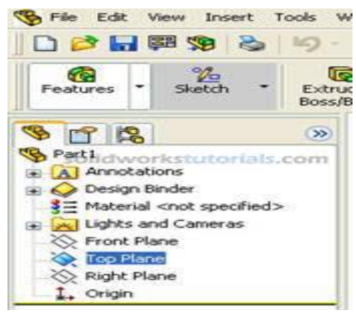
<u>**AIM:**</u>

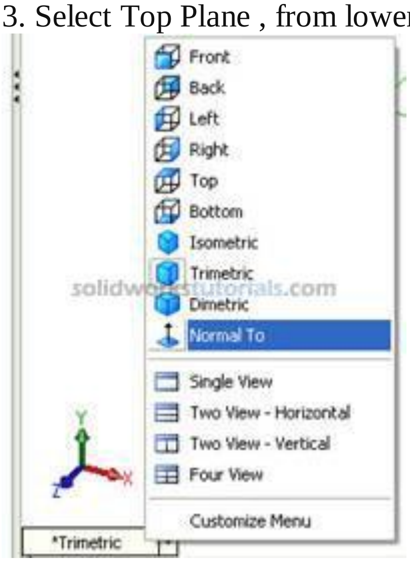
To model the given object using the SWEEP feature as per the dimensions given.

1. Click New  (File>New) , click Part 

, OK .

2. Click Option  (Tools>Option…) , select Document Properties tab. Select Units , under Unit System select IPS (inch, pound, second) OK.

3. Select Top Plane , from lower left menu select Normal To. 



4. Click Sketch 

in Command Manager, click Circle 

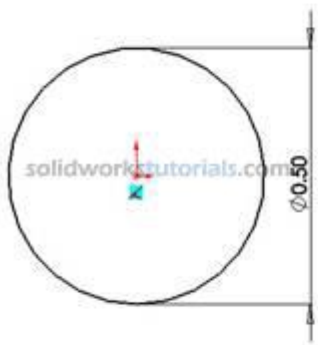
. As you can see on upper right corner sketch icon appear indicate that you're on sketch mode .

5. Pick Origin   point as starting point, drag to right hand side 

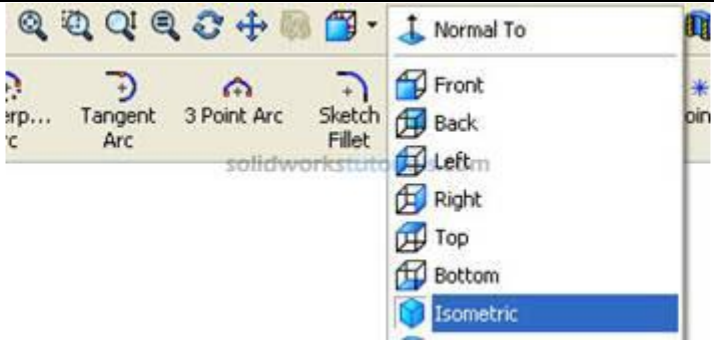
no need to be exact the size will define in later step. Press keyboard ESC to end circle sketch.

Note: There is two type line generated by in sketching, the one with black line and blue line. Black line is line that fully defined and blue line is under defined..

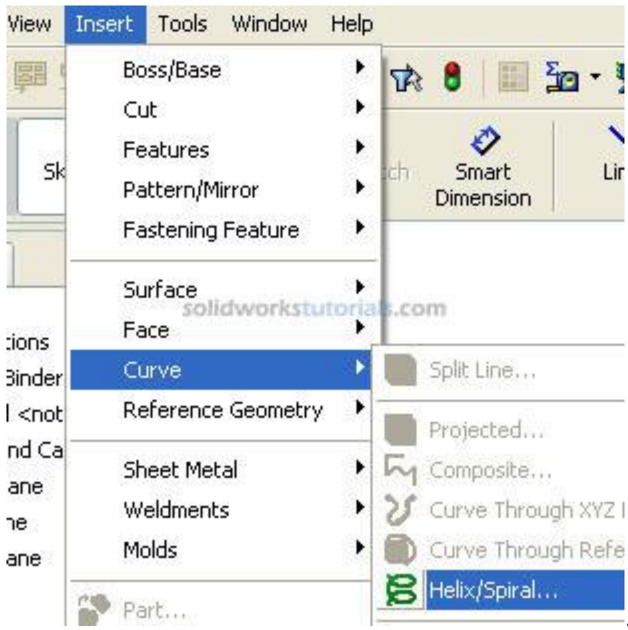
6. Define sketch with dimension. Click Smart Dimension 

, and start dimensioning pick circle edge and set to 0.50in 

. Press keyboard ESC to end smart dimension.

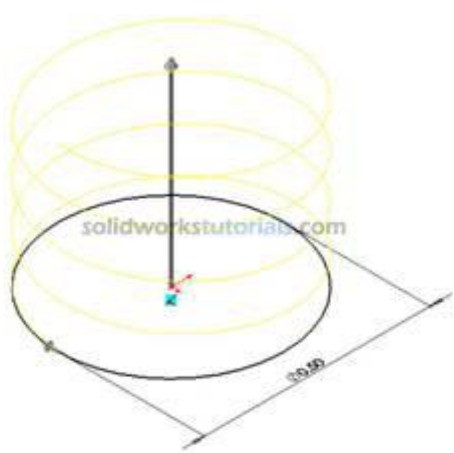


7. Change display to Isometric view.

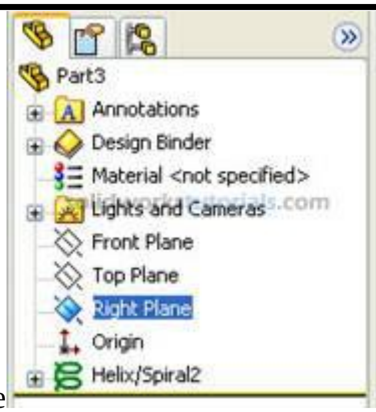


8. Insert coil, Click Insert>Curve>Helix/Spiral.

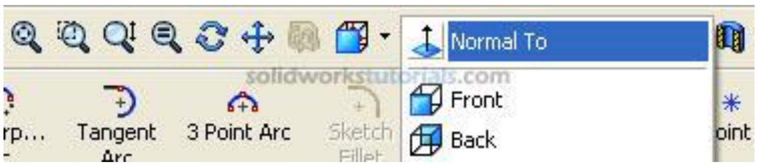
9. Press F to zoom fit, set Parameters Constant Pitch , Pitch 0.10in Revolutions 4 , Start angle 0.0deg



and  .

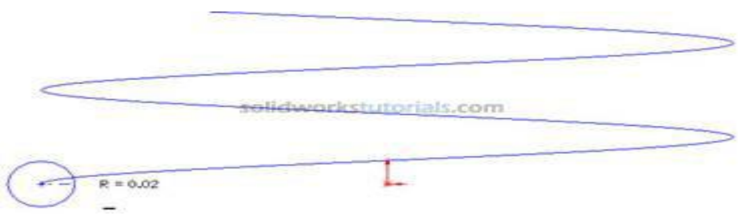
10. Click to Right Plane 

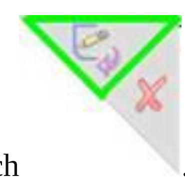
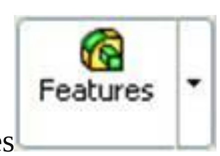
, click Normal



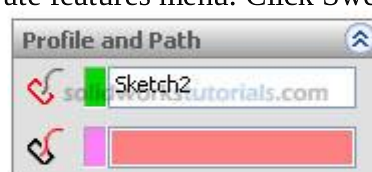
To .

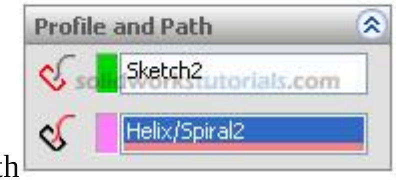
11. Click Sketch , click Circle . Sketch circle at start point, then click Smart dimension

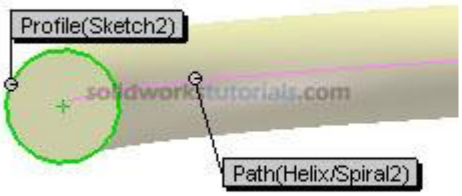
set circle diameter to 0.05in . 

12. Click exit sketch  . Click Features 

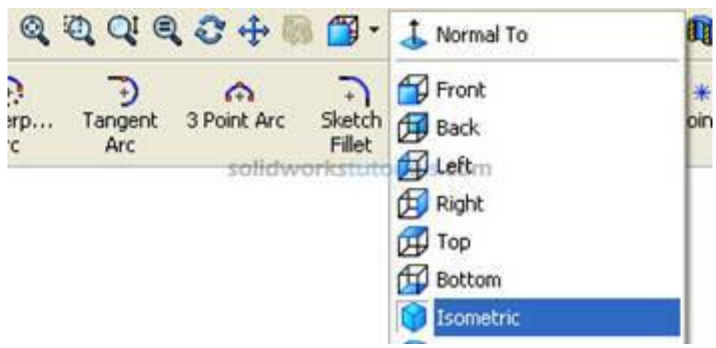
and activate features menu. Click Swept Boss/Base 

and set Profile to Sketch2 by click on circle sketch 

and set Path by click helix path 



and  .



13. Change display to Isometric view.

14. Press F to zoom fit.