Introduction to SolidWorks

Starting SolidWorks

Select the **SolidWorks** icon on the desktop or from the menu start

- Note that the lectures are based on the default settings

The window in figure (1) is going to appear When the SolidWorks program is loaded. The Menu Bar has the commonly used tools such as (New, Open, Save, etc). By default the Menu Bar menus are hidden. To display them, move the cursor over or click the SolidWorks logo.

Figure (1)

First thing to do is either to start new model or to open an existing one. For this lecture select the **New** icon from the Menu Bar 🗋

The dialog box in figure (2) will appear. Three options will appear, Part, Assembly and Drawings

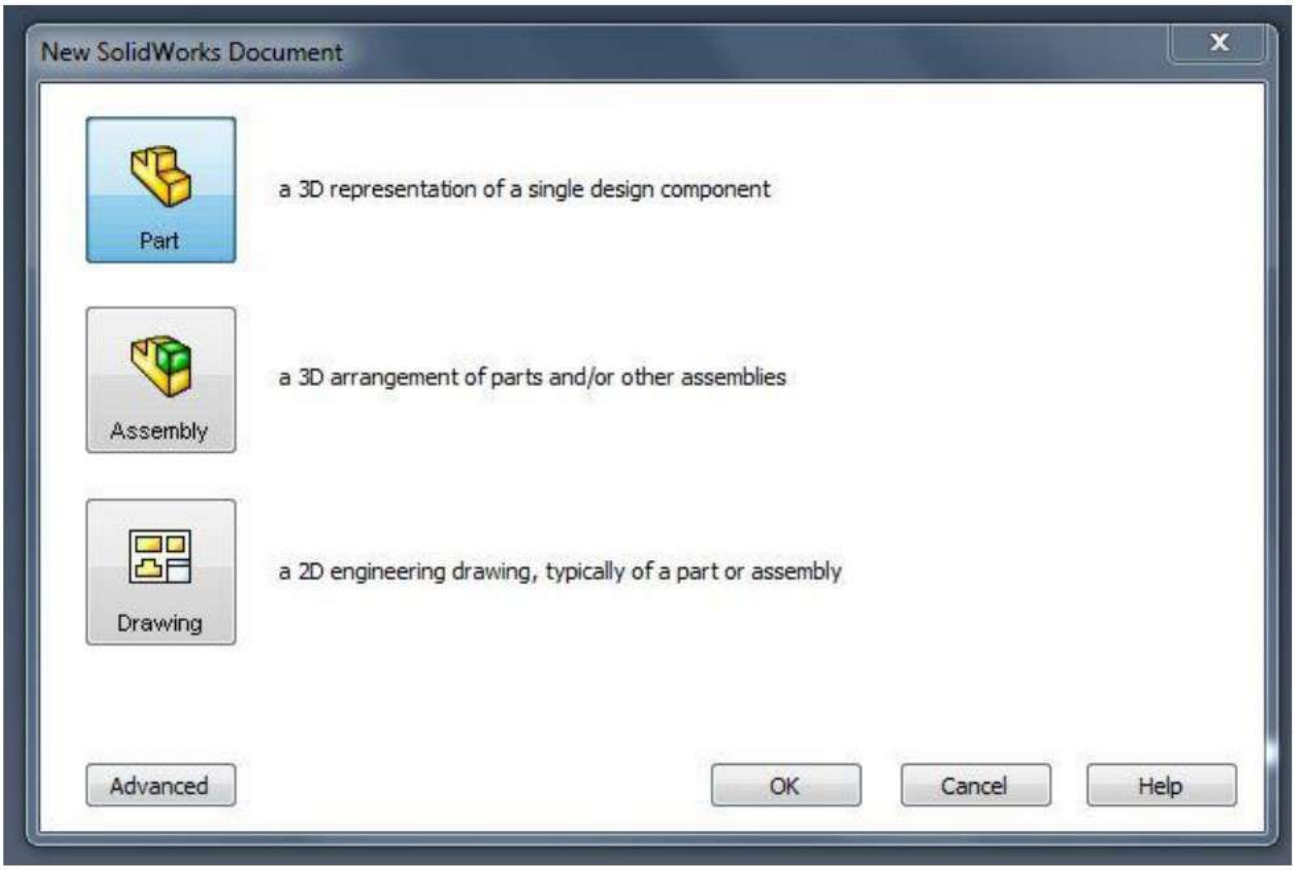


Figure (2)

- **Part** - this option will open a window that allow you to create a single solid model part either a two or three dimensional one.
- **Assembly** - this option will open a window that allows you to assemble the parts that was creating earlier using the Part option.
- **Drawing** - using this you can create a two dimensional representation for a part or assembly.

SolidWorks Layout

Window in figure (3) will appear, if you select Part in the previous step. SolidWorks Screen Layout has the Menu Bar, the Heads-up View toolbar, the Feature Manager Design Tree, the Feature toolbar, the Sketch toolbar, the graphics area, the Reference Triad, and the Status Bar.

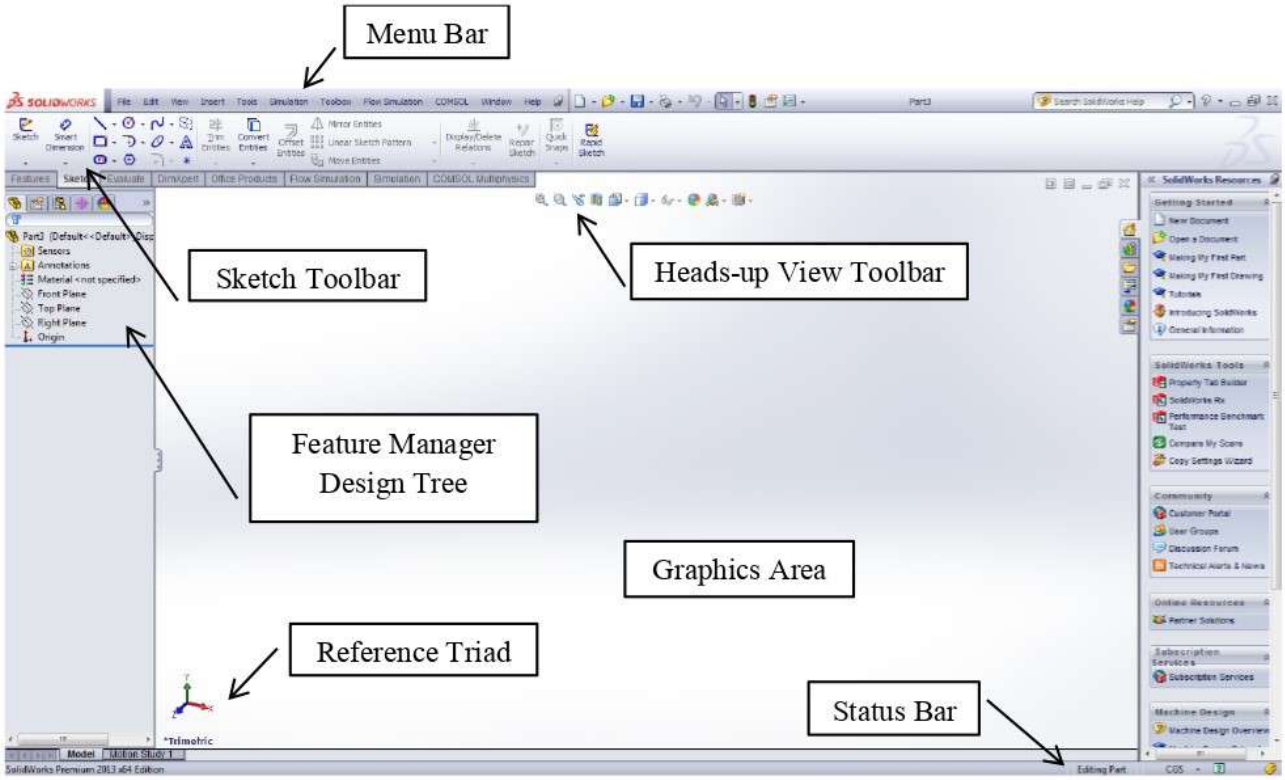


Figure (3)

Units Setup

The units of SolidWorks program can be changed using the following steps

- ⇨ Select the Options icon from the Menu Bar
- ⇨ Open Document Properties tab, and select Units
- ⇨ Select the Unit system you want and click OK, you can also select the degree of accuracy from the Decimals spin box located at the same window.

Sketch

The sketch is a two dimensional drawing which will be used to create the part. To create sketch you need to select **plane** from the Feature Manager Design Tree, the program have three default plans which are front, top, and right planes. You also have the possibility to create another planes based on what you need to model, we will talk about that in other lectures.

Tool used to create the Sketch

Figure (4)

Figure (4) shows the basic tools that can be used to create the sketch, which are Line, Rectangular, Circle, Arc, etc. It also has a Smart Dimension option that can be used to set the dimensions in an easy way. Other than the basic option the sketch tool bar have options such as

- Mirror Entities – used to duplicate the sketch over an axis
- Sketch Pattern – used to create liner or circular patterns
- Move Entities – used to move the sketch from one point to another
- Trim Entities – used to delete lines in the sketch

To practice the sketch tools we will create the sketch in figure (5). Note that you can create the same sketch in different steps, in SolidWorks program you have so many tools and options to create any model you can think of.

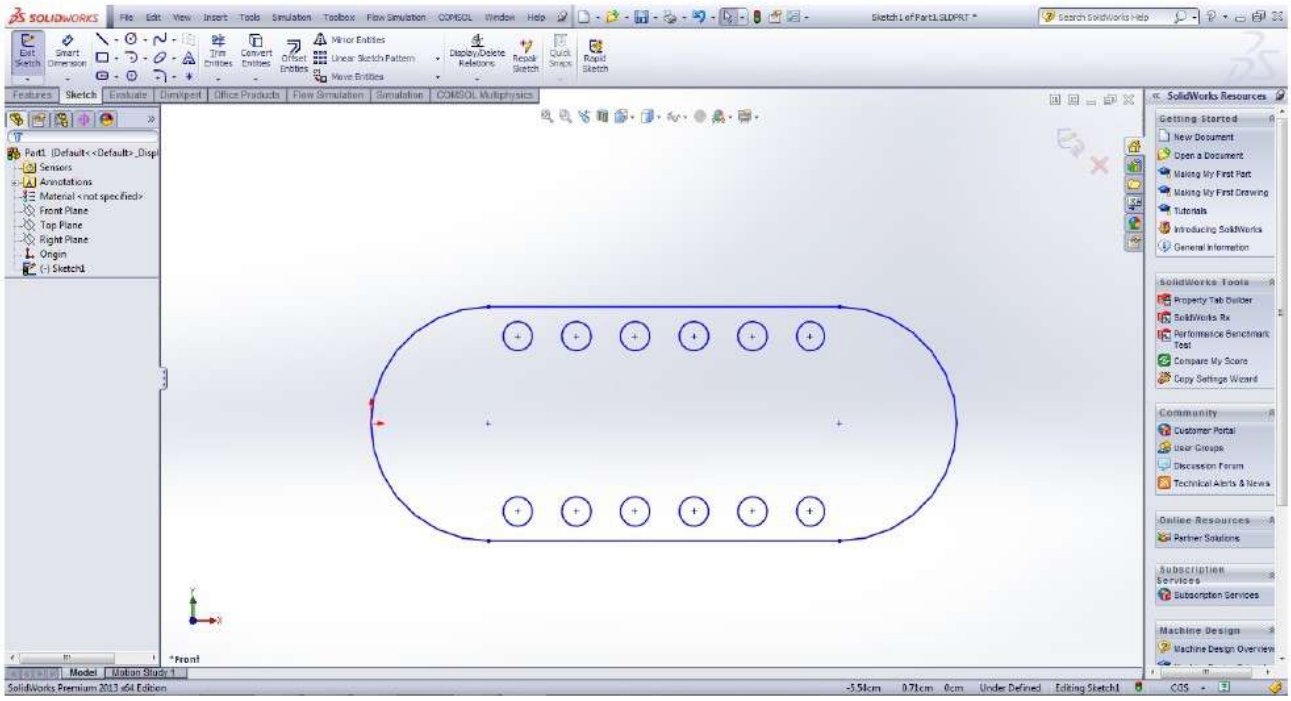


Figure (5)

⇨ Click on the front plane and select sketch
⇨ From the sketch tool bar select line
⇨ Click and drag in any point on the graphic area to create two lines as shown in figure (6)

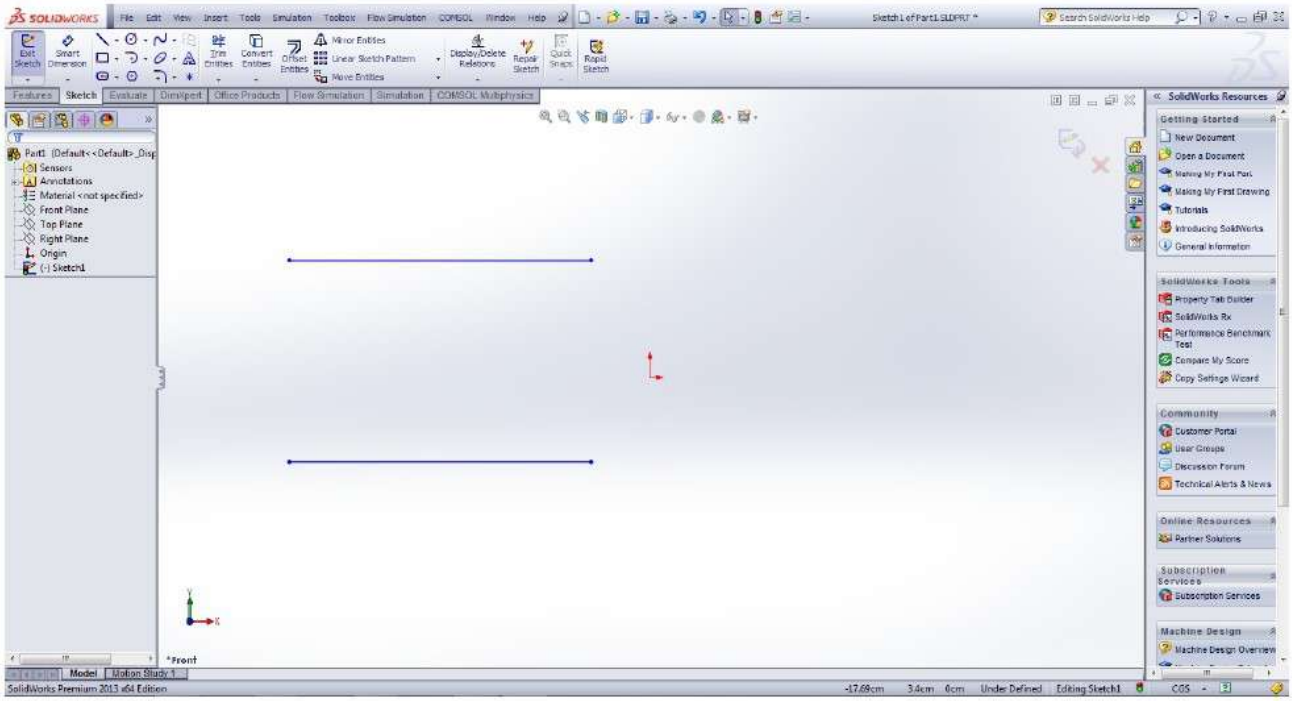


Figure (6)

⇨ Select Arc from the sketch tool bar, select Tangent Arc from The Arc Property Manager, and create two Arcs as shown in figure (7)

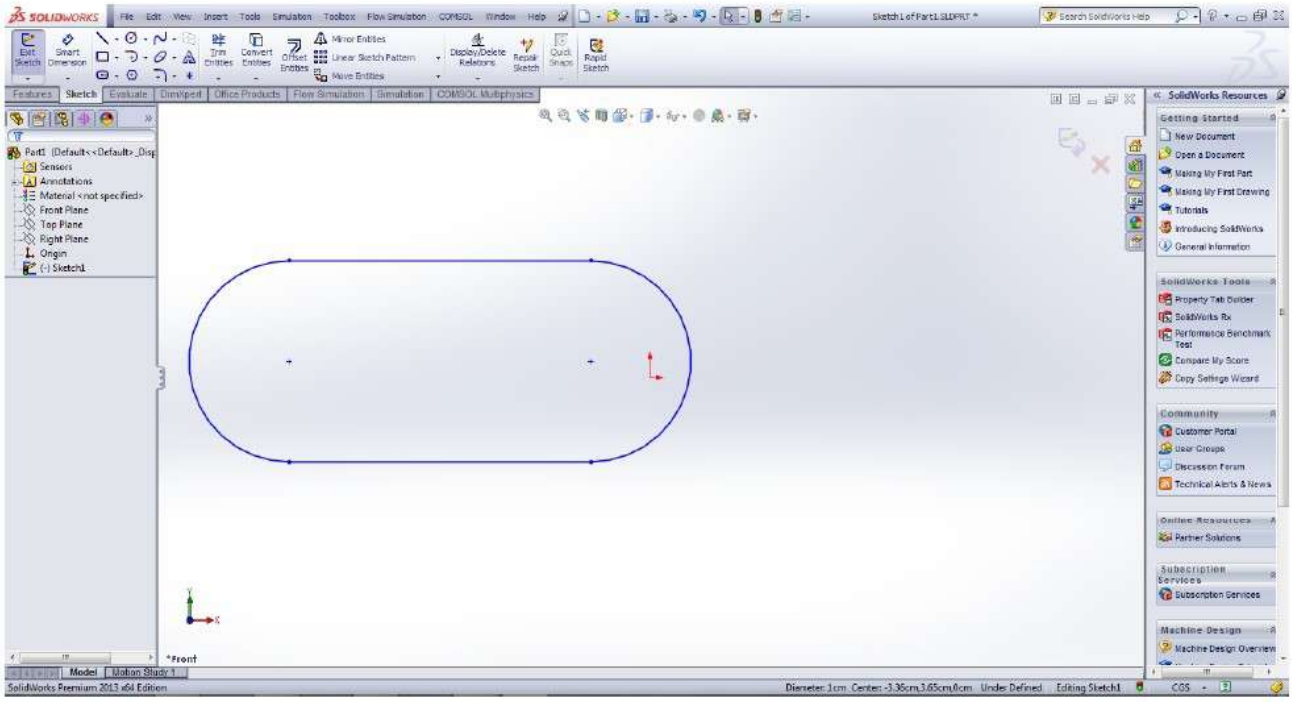


Figure (7)

⇨ Select circle from the sketch tool bar and create circle as shown in figure (8)

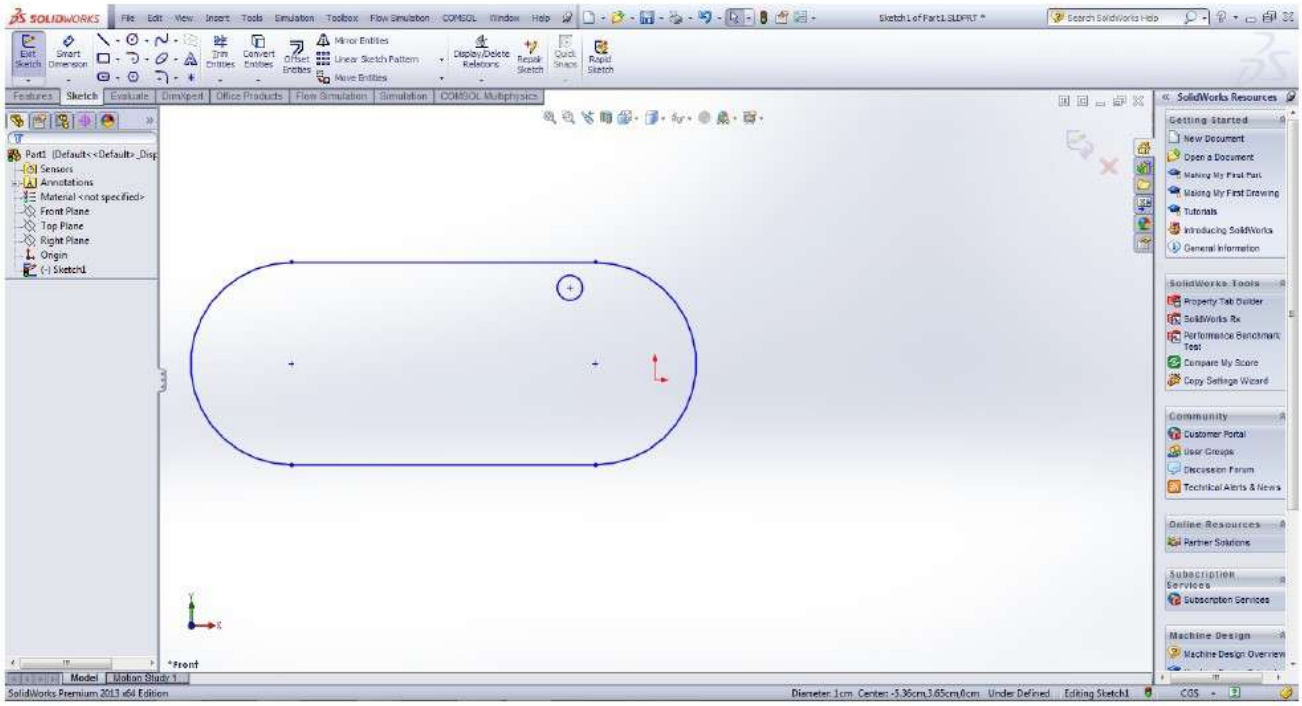


Figure (8)

- ⇨ Select Linear Sketch Pattern from the sketch tool bar
- ⇨ Select the circle you just created as Entities to pattern
- ⇨ In direction 1 set the number of circles you want to create and the spacing of that separates between their centers as shown in figure (9)
- ⇨ Click OK

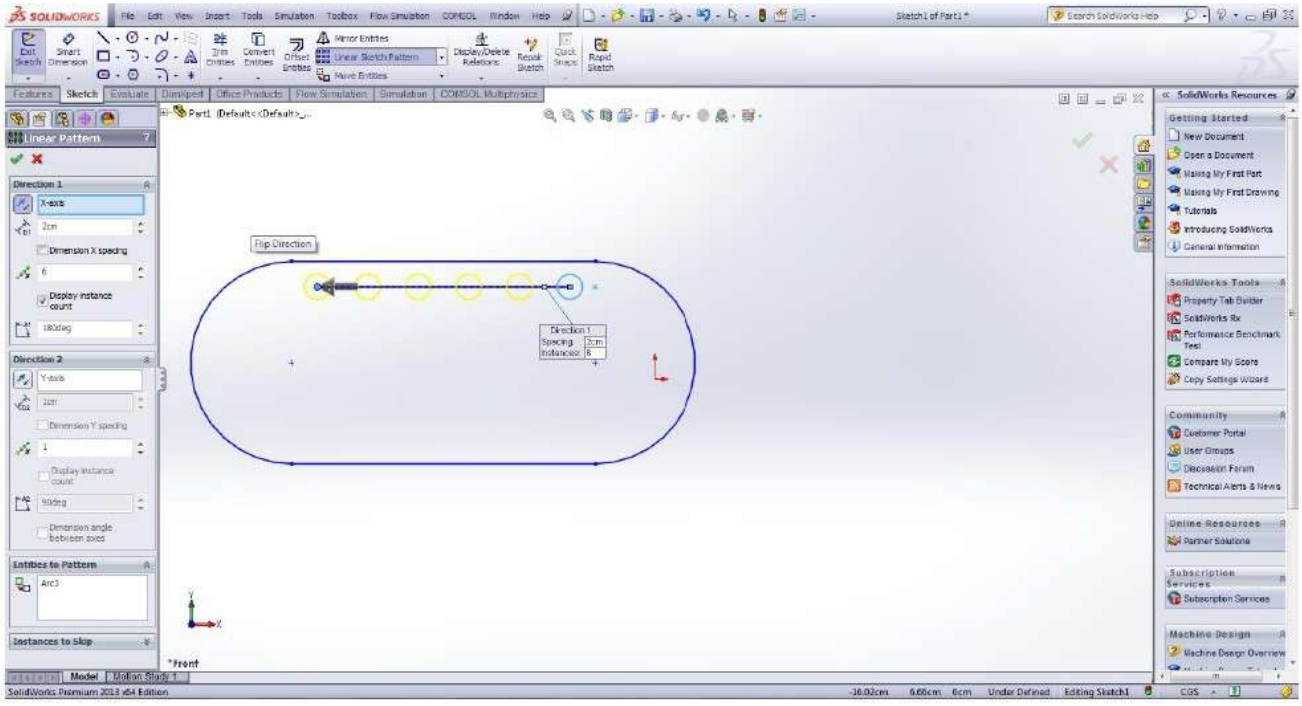


Figure (9)

- ⇨ Create axis at the centerline of your sketch as shown in figure (10)

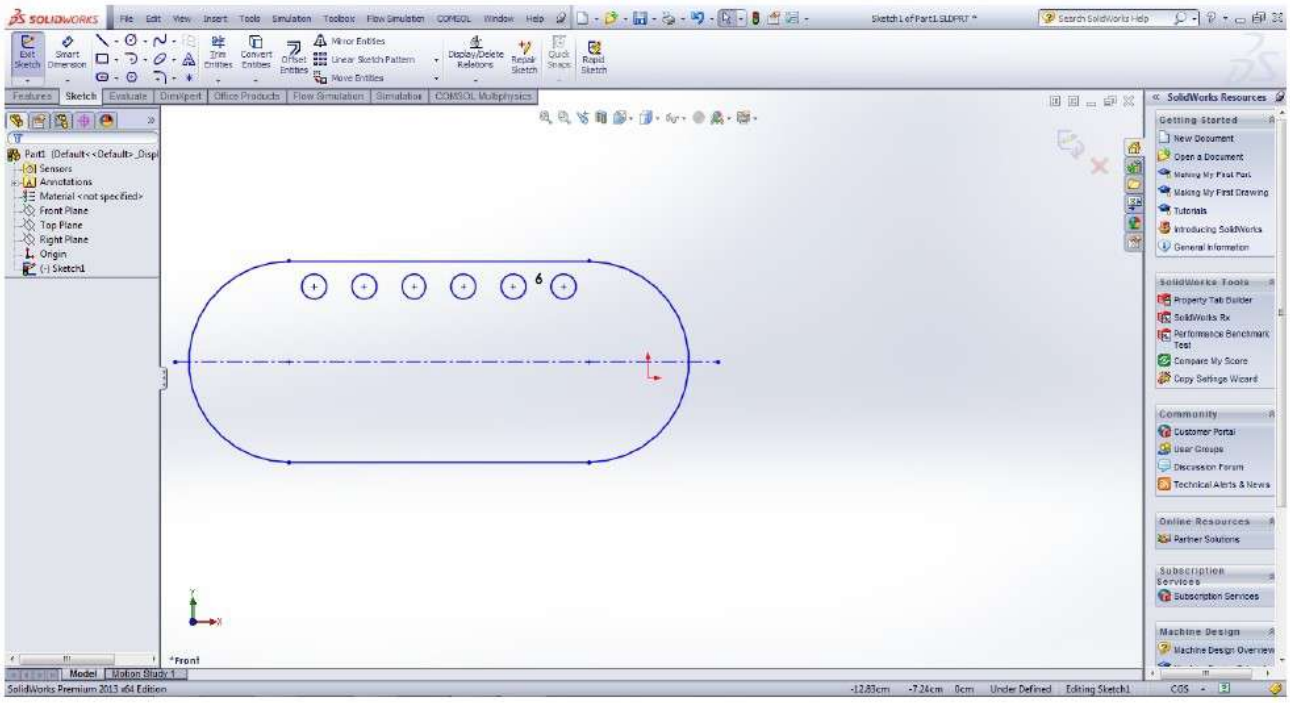


Figure (10)

- ⇨ Select Mirror Entities from the sketch tool bar
- ⇨ Select the small circles as the Entities to mirror
- ⇨ Select the centerline as the Mirror about line
- ⇨ Click OK

The result are shown in figure (11)

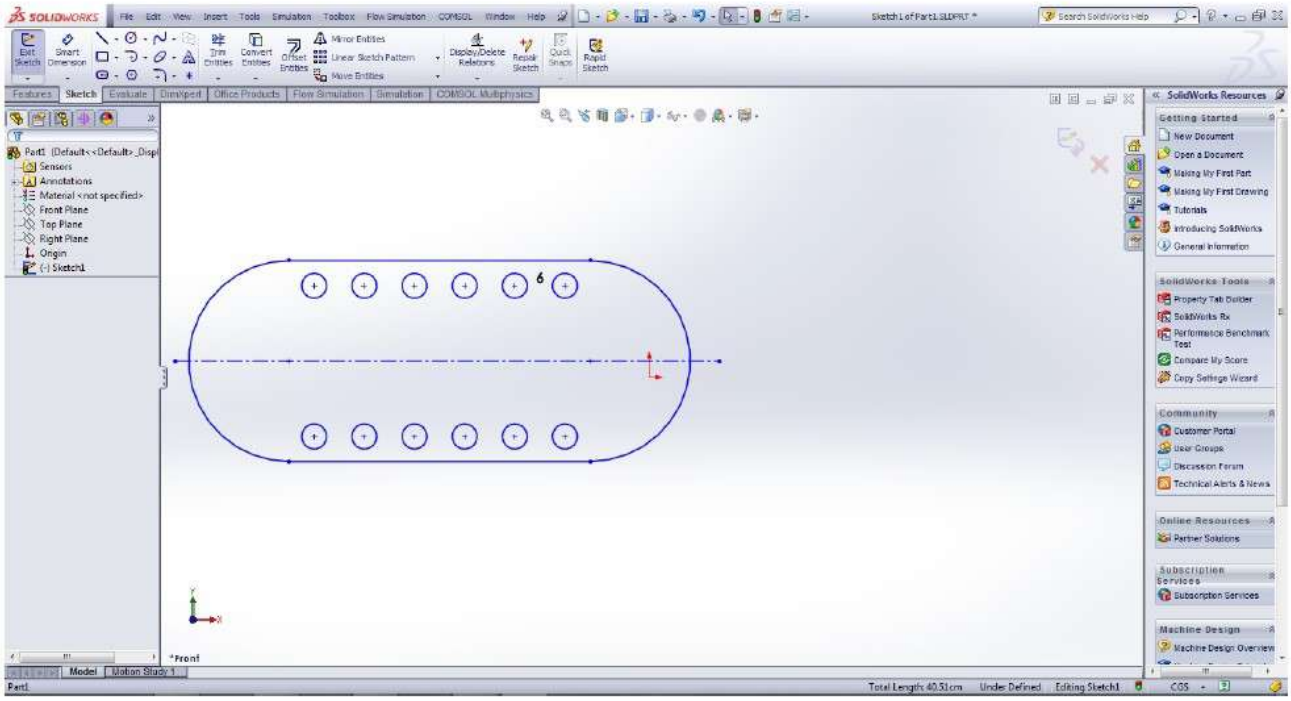


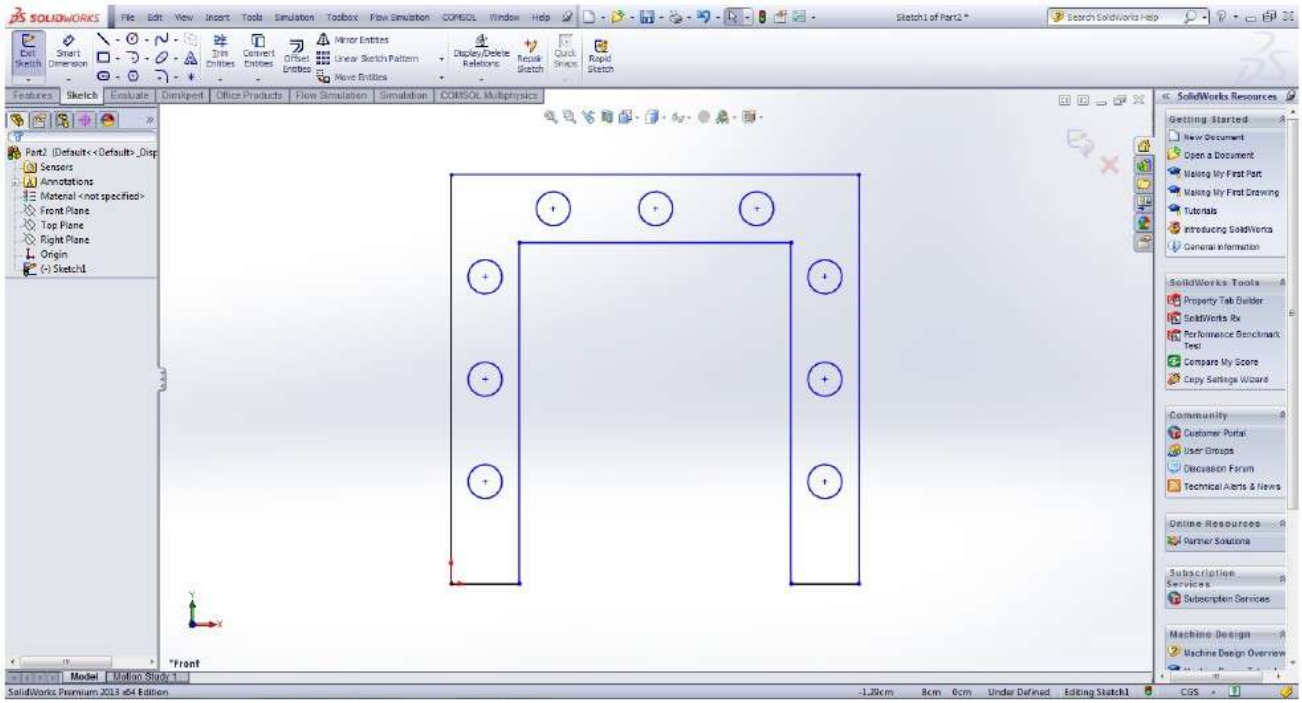
Figure (11)

- ⇨ To finish the sketch click on the centerline and hit delete from the keyboard or select Trim from the sketch tool bar and click on the centerline

- ⇨ Select Move Entities, to move the sketch you just created to the original reference point on the graph area
- ⇨ Select all the lines that created the sketch as Entities to Move
- ⇨ Select the top point on the arc as the point you want to start with
- ⇨ Select the original reference point on the graph area

The result should be as figure (5)

H.W



How to add dimensions and relations

For example we are going to make the sketch shown in figure 12

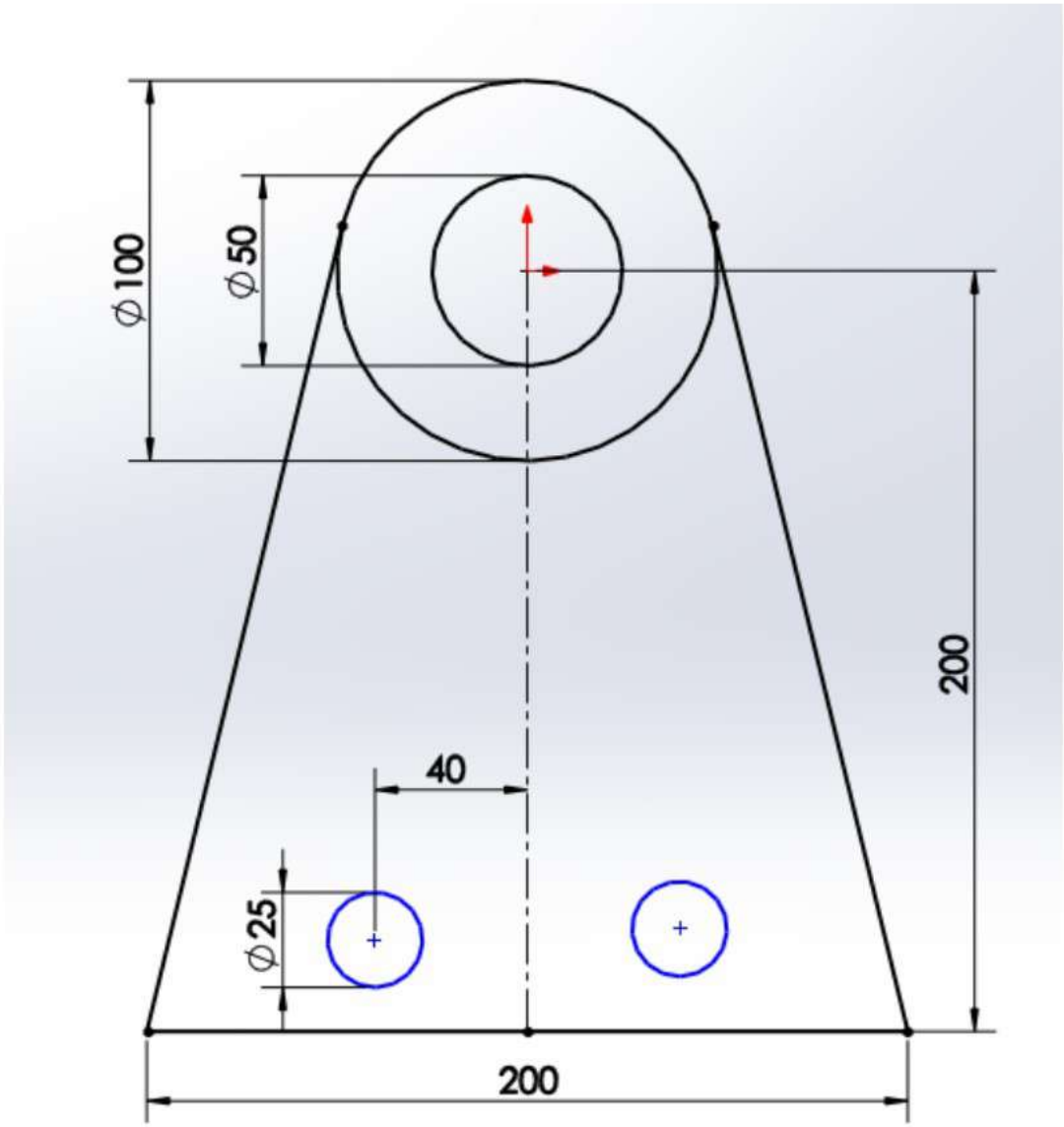
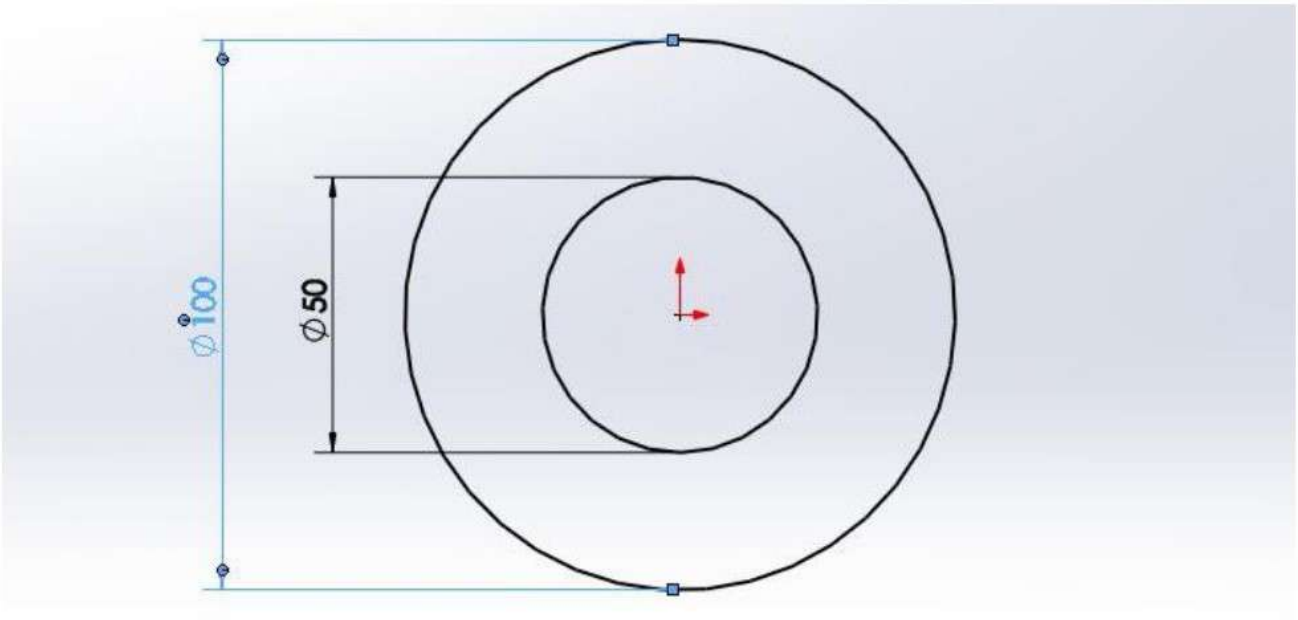


Figure (12)

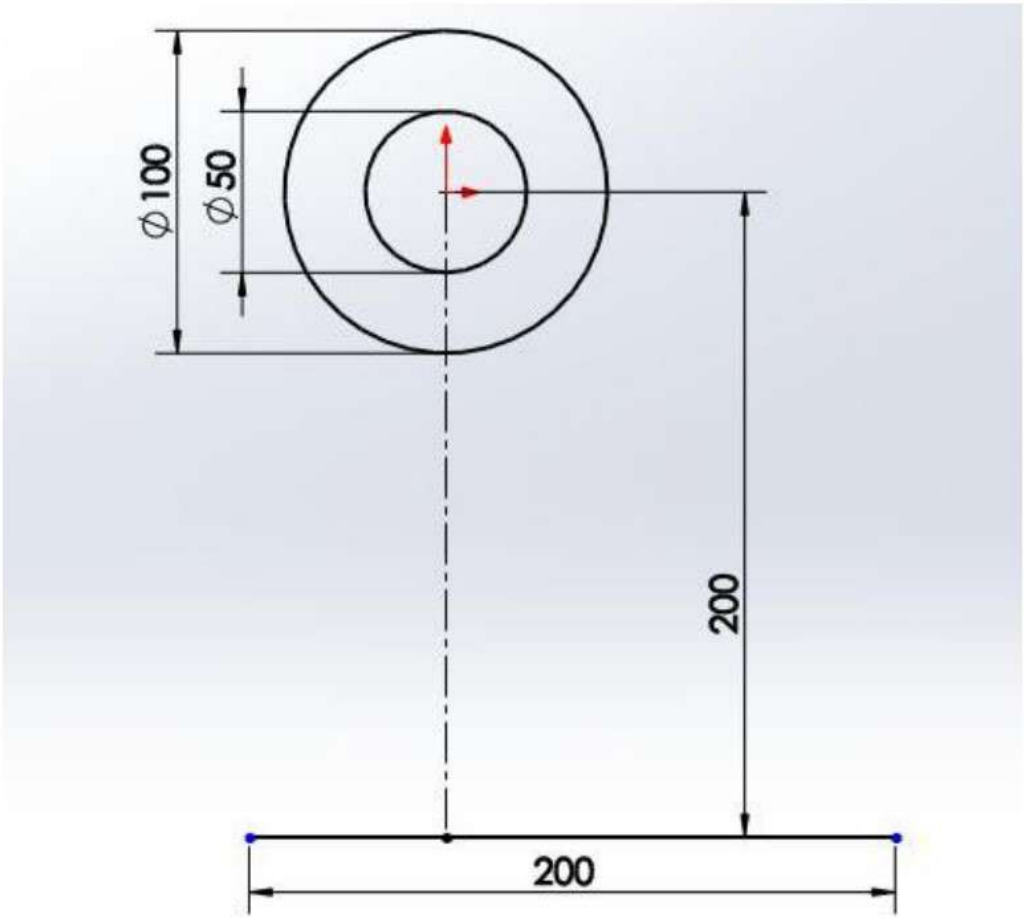
⇨ Create the two circles with 100 mm and 50 mm

To set the dimensions

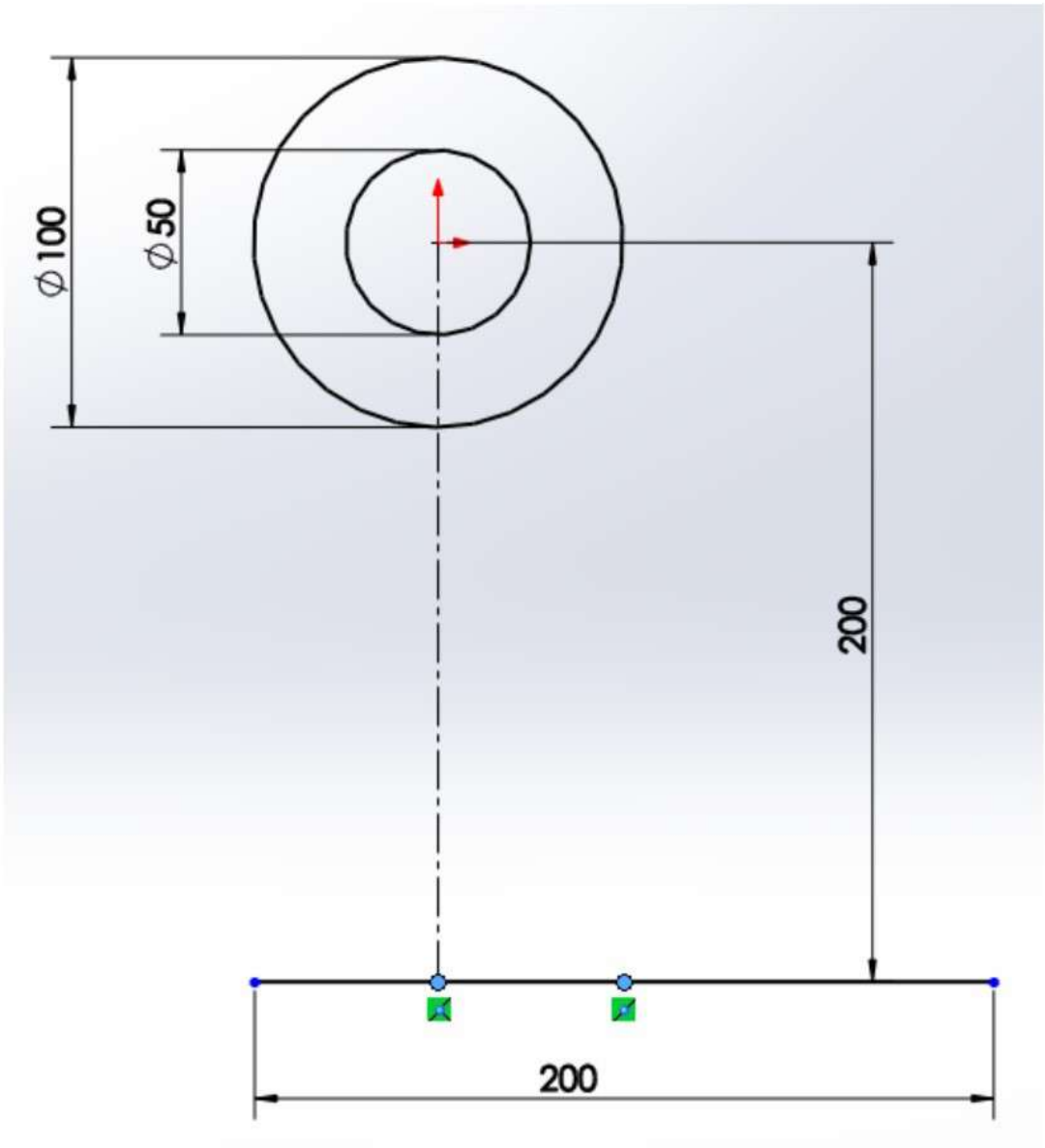
- Select Smart Dimension from the Sketch Toolbar
- Click on the circle
- Drag and click on clear area on the graph window
- Change the dimension and click Ok



⇨ Create a centerline that goes 200 mm from the center of the circles to the base
⇨ Create the base line which is 200 mm



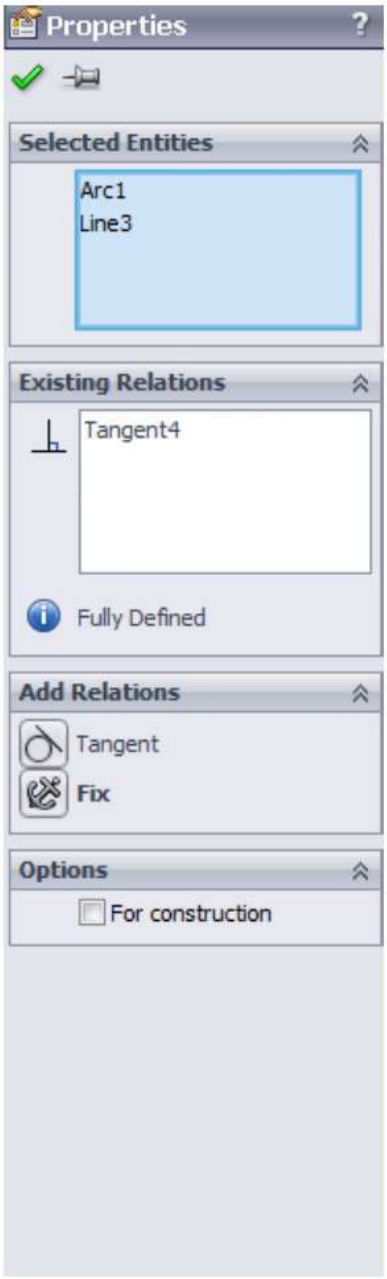
Make a relation between the centerline and the base line



- ⇨ Right click on the base line and select the midpoint
- ⇨ Hold ctrl from the keyboard
- ⇨ Select the end point of the centerline
- ⇨ From the menu on the side select coincident

- ⇨ Create the small circle on the side
- ⇨ Select Mirror option and make mirror of the small circle on the centerline

⇨ Create two lines from the two sides of the base line



⇨ Select the first side line
⇨ Hold ctrl from the keyboard
⇨ Select the top circle
⇨ From the menu on the side select tangent
⇨ Repeat the last four steps on the second side line
⇨ Use Trim option (Trim to closest) to delete the excess of the lines