1.) Open the SolidWorks program

2.) To create the new part design, on the menubar click **File>New>Part>OK**

3.) Start drawing the 2D sketch, on Sketch tab click **“line”**. It will show all three views of sketch, **select the plane** by moving mouse on the right plane and click mouse when it turns orange.

4.) Click on the **view orientation** and change to right plane, Start drawing the **outline sketch** of the part

5.) Using **“Smart Dimension”** to modify the length of the sketch

6.) If the decimal of the dimension is not properly displayed, on the menubar click **Tools>Options**. On Document Properties tab, click on **“Units”** on the left-hand side menu. You can edit the **drawing units and decimal place** here then click OK

7.) After finish drawing the outline sketch with correct dimensions, Click **“Exit Sketch”**

8.) Select the finished sketch on the **“Feature Manager Design Tree”** then on the Features toolbar click on **“Revolved Boss/Boss”**

9.) On the Revolved properties manager on the left side, specify the correct **“axis of revolution”** and **“angle to revolve”** then OK

10.) Draw the circle on front-most surface, but clicking on the **“circle”** and select the front-most surface as a plane to draw, using smart dimension to modify the radius of the circle. Click **“Exit sketch”**

11.) Select the latest sketch, go to Features toolbar and select **“Extrude Cut”**. Change to **“Through All”** mode, Click on check mark

12.) Draw the small circle on the remaining the big surface by drawing the construction line from the center of the object first. **Make sure you select the correct plane to draw**. Then draw the circle and give the correction dimension using Smart Dimension.

13.) Delete the construction line and Exit Sketch.

14.) Select the sketch you just draw and on the Features toolbar click on **“Extrude cut”** and select Through All mode. Then OK

15.) Doing the **circular array** of the small holes by making the selection on the latest Extrude Cut and click the drop-down arrow below **“Linear Patter”** and select **“Circular Pattern”**.

16.) On the left hand, the properties manager, specify the angle to pattern = 360 degree and check mark at **“Equal Spacing”** and number of the objects=6 (including the original one), **importantly select the axis to pattern using any circular surface from your 3D design** that share the same center as the feature that you want to pattern. Then click **check mark (=Done on each drawing step)**.

17.) Do the Fillet, by clicking on the **“Fillet”** on the feature tab and select the edge that you want to do fillet. Don’t forget to specify the correct **fillet radius** and don’t forget to turn on **Full “Preview”**.