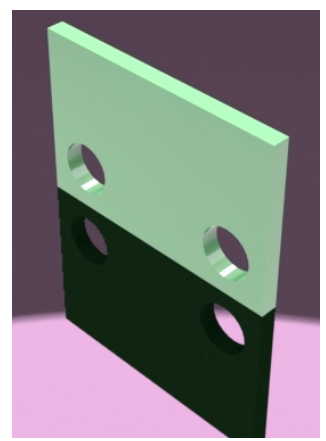
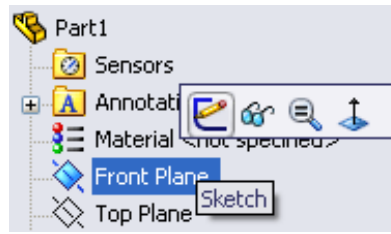


Create mirror

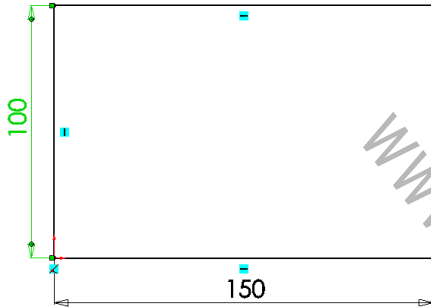
1. Click New.  Click Part,

Click **Front Plane** and click on **Sketch**



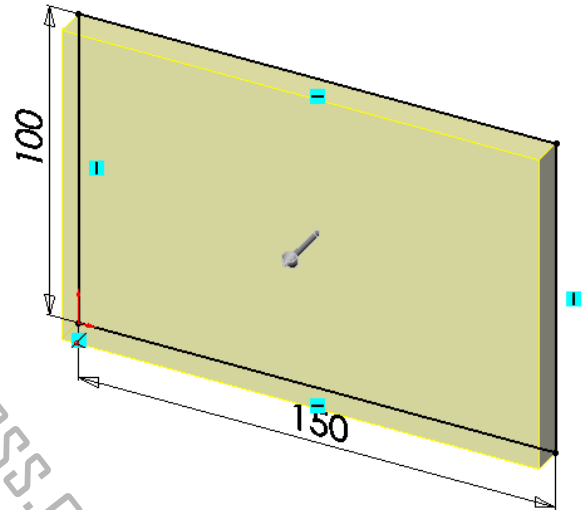
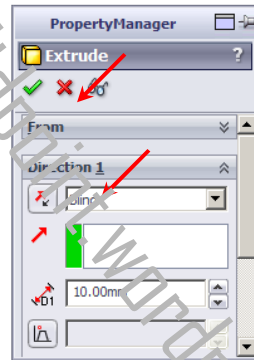
3. Click Rectangle, sketch rec-  tangular. Click Smart Dimension, 


dimension rectangular 100 x 150 mm

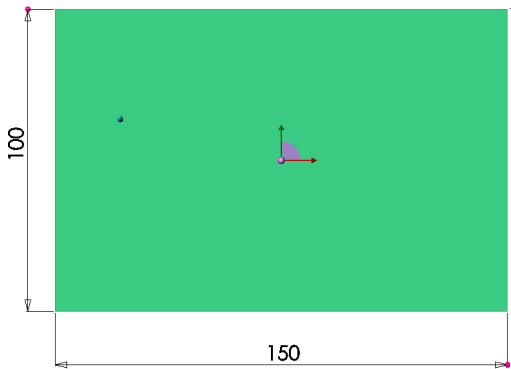


4. Click **Feature**>**Extruded Boss/Base**, 

- End condition **Blind**
- **D1** 10mm
- Click OK

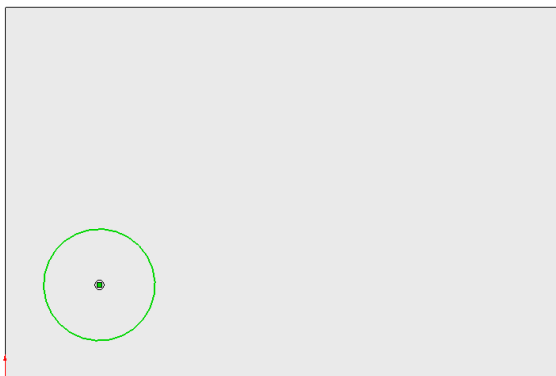


5. Click on front face and select Normal to. 

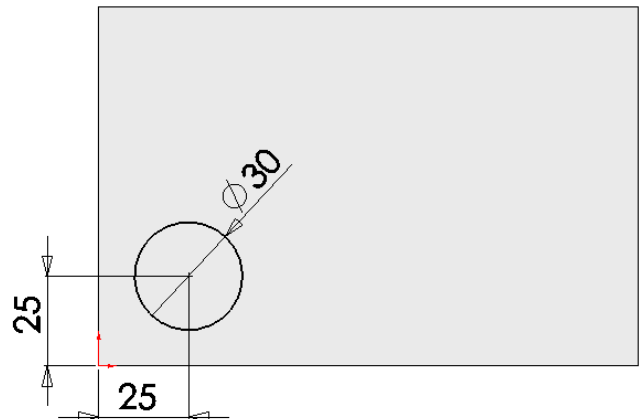


6. Click **front face** and Insert Sketch.

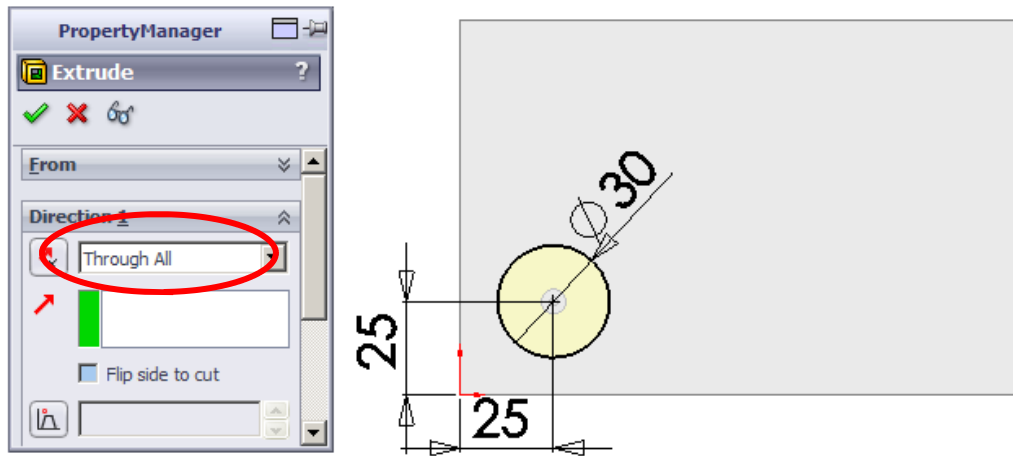
7. Click Circle,  sketch circle at one edge.



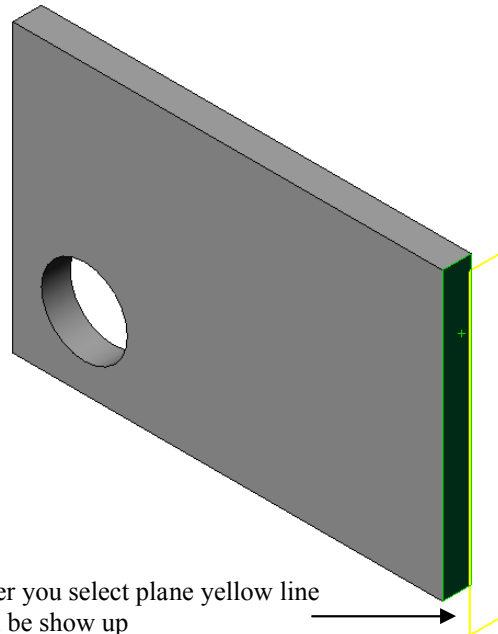
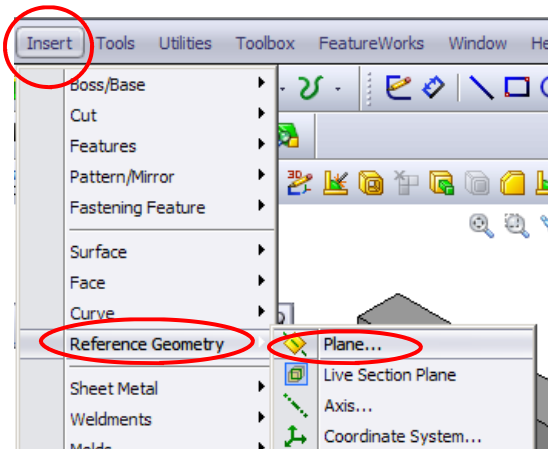
8. Click Smart Dimension, dimension circle as below sketch.



9. Click Features>Extruded Cut,  select **through All**



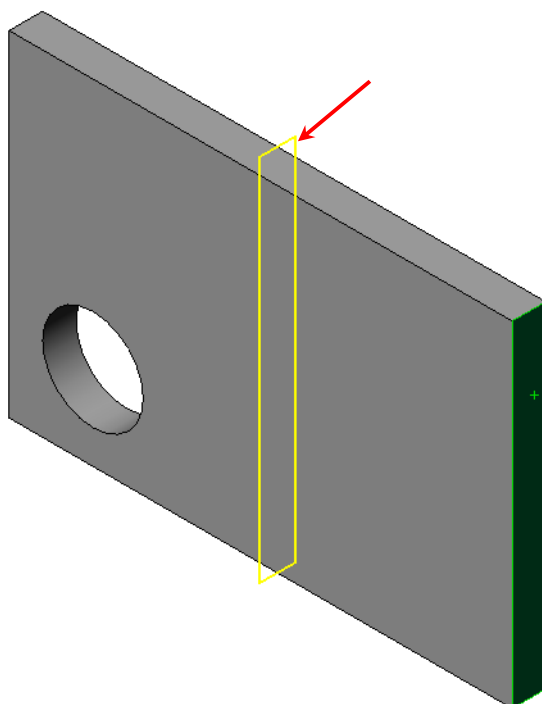
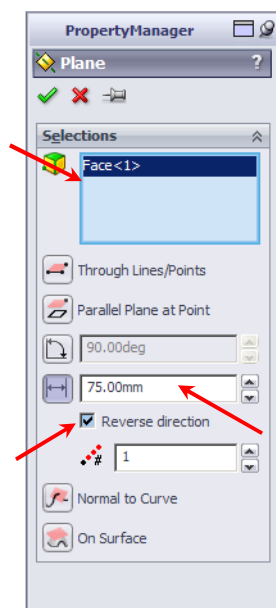
11. To add mirror plane, click Reference Geometry>Plane.



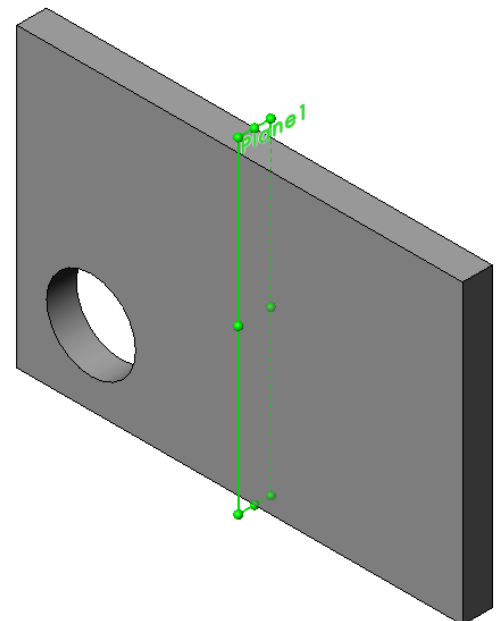
In proptimanager insert


- In section chose Plane 1
- In offset distance 75 mm
- Click Reverse direction

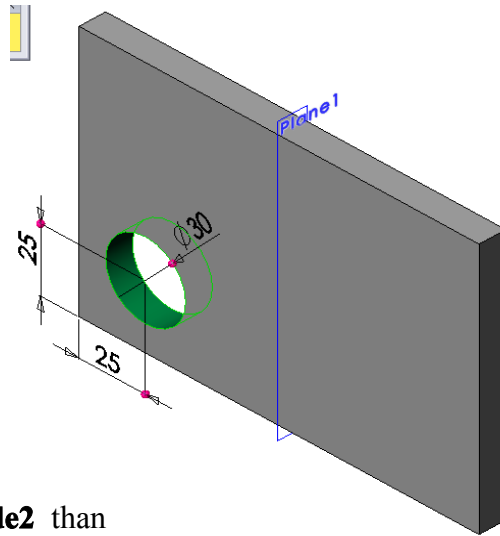
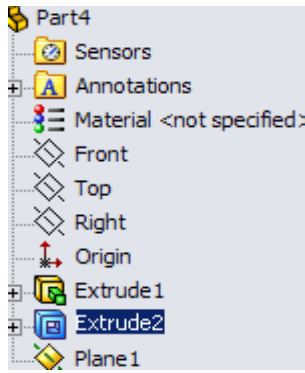
After you select plane yellow line will be show up



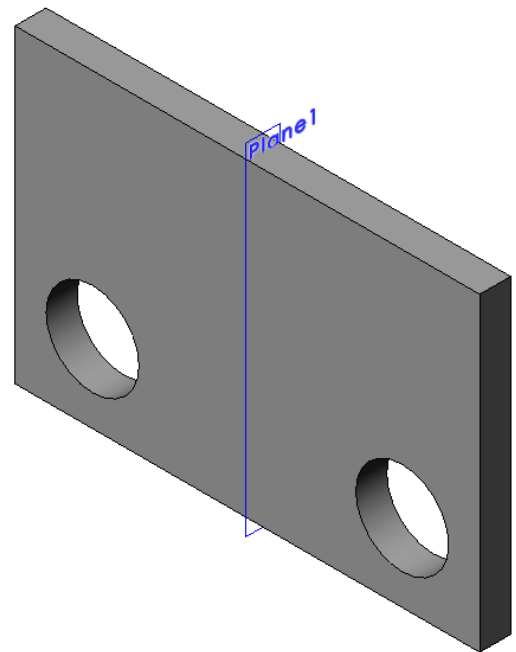
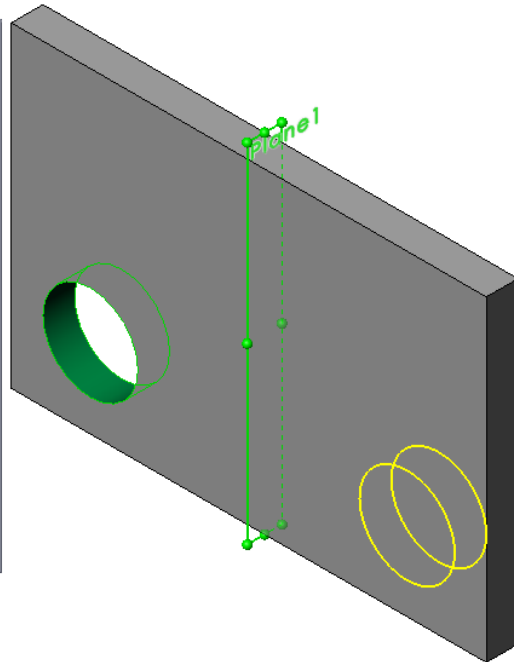
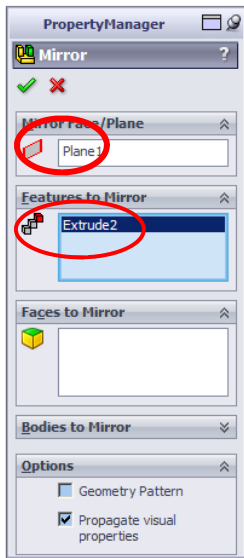
12. Click OK.



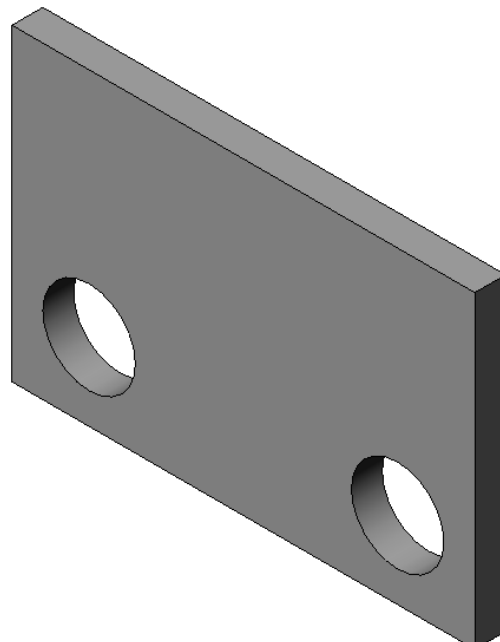
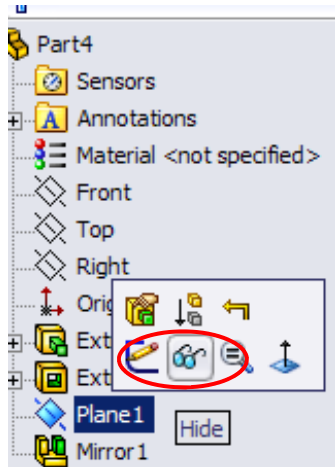
13. Click on Extrude2, click Mirror,  or,



select **Plane 1** and **Extrude2** than



14. To hide mirror plane, click **Plane 1**, select **hide**.



You can send your design to my blog, that will be in good stead for newcomer in solidwork design. Thanks awfully