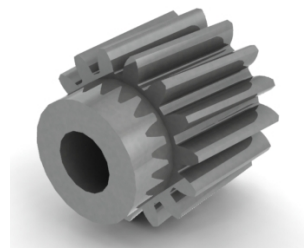
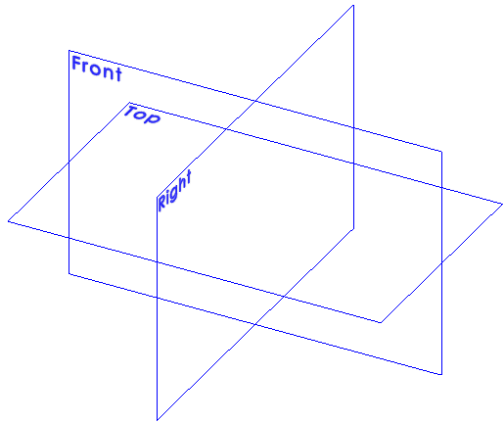


Helical Gear



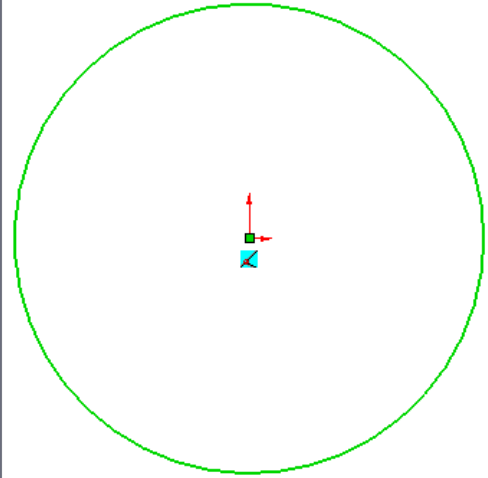
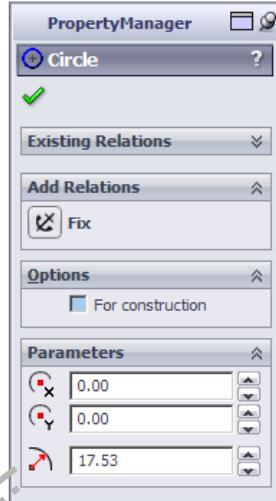
1. Make Extruded

1. Click **Extruded Boss/Base**  on Feature toolbar.



2. choose plan **Front**.

3. Click **Circle**  on Sketch toolbar.

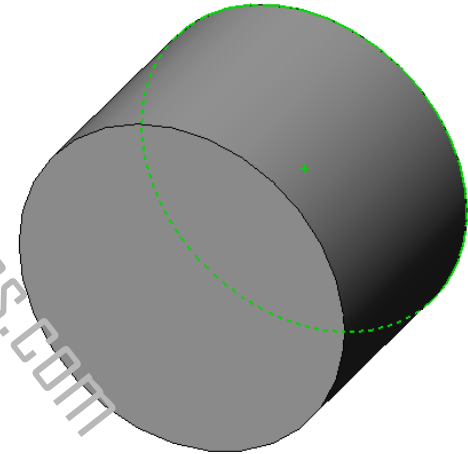
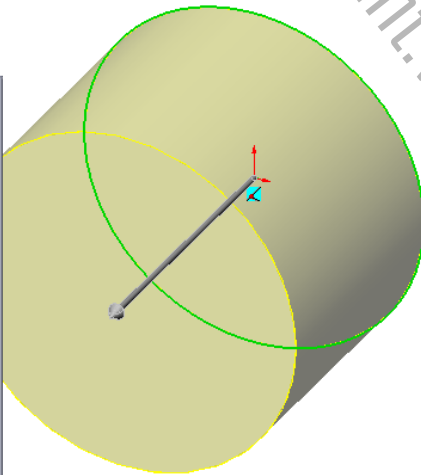
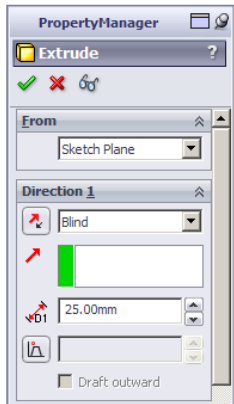


4. Exit Sketch



- chose **Blind** on **End Condition**.
- Set **Depth** to **25**

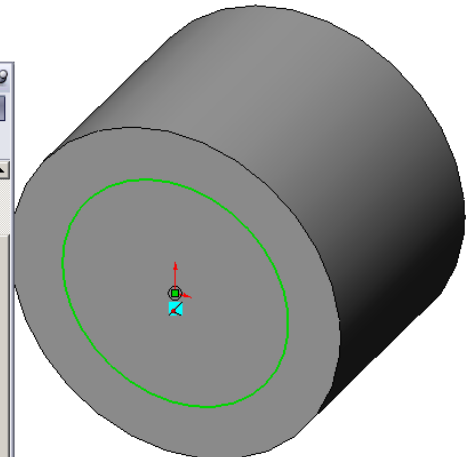
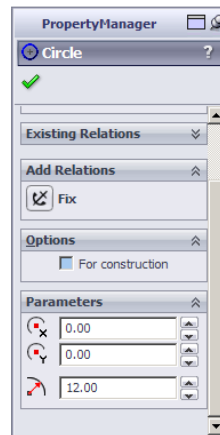
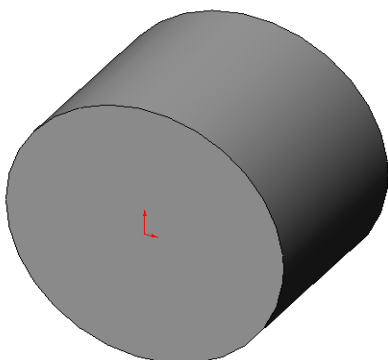
5. Click **OK** 



6. Click **Extruded Boss/Base** 
7. Click **Surface of part**

9. make Circle in **Origin**  point insert R 12 mm

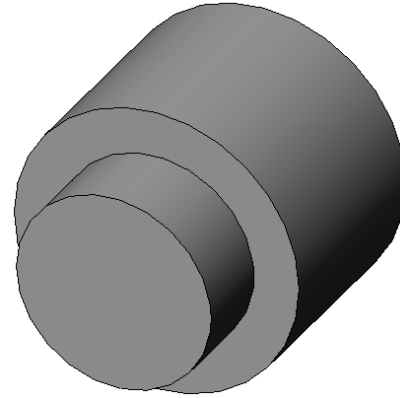
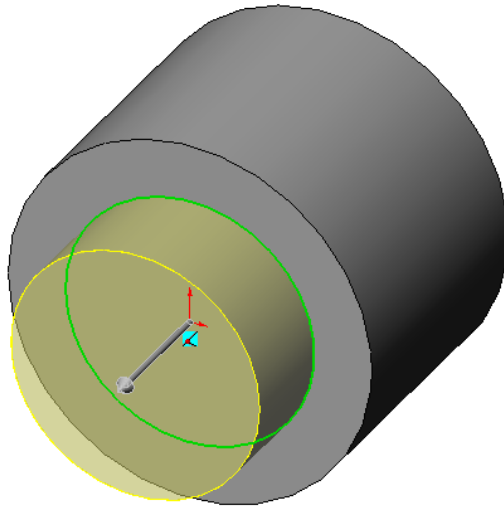
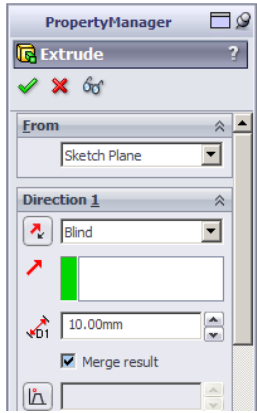
8. Click **Circle**  on toolbar



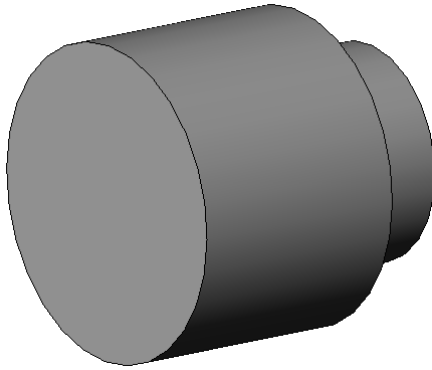
10. Klik **Exit Sketch**  on Sketch

- In Property manager set
- Direction 1 set to **Blind**
 - D1 set to **10**


11. Klik **OK** 

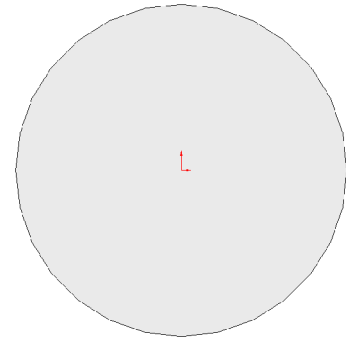
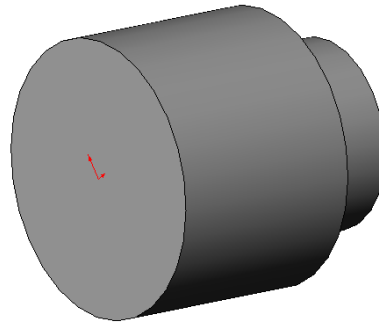




12. Click **Rotate View**  to see another surface of part

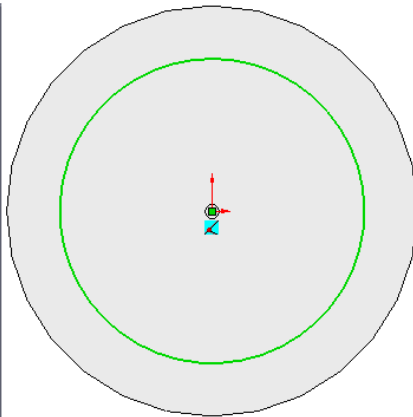
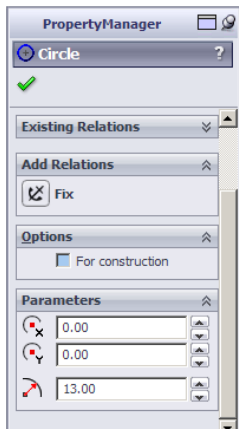


13. Click **Extruded Cut**  on Feature toolbar.

14. Chose surface of part and chose **Normal To** 

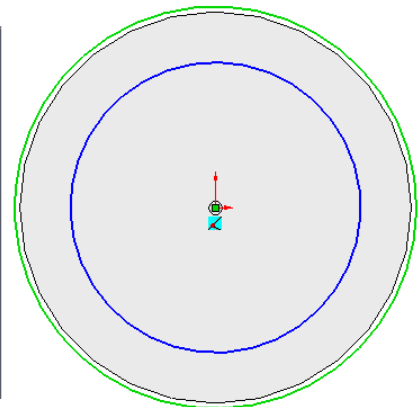
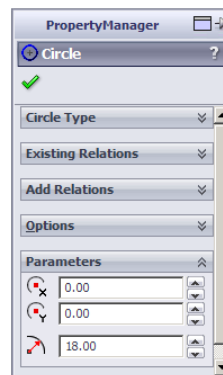


15. Click **Circle**  on Sketch toolbar. • Set **Radius**  to **13**.



16. Repeat step 15




Set **Radius**  to **18**

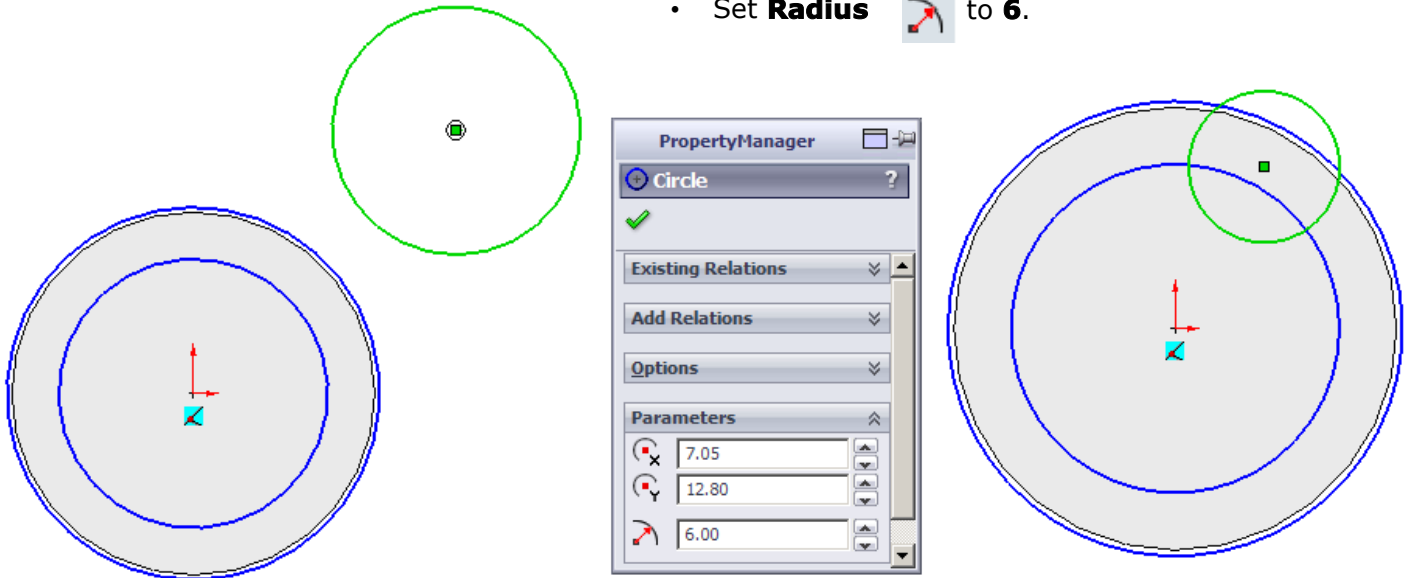


16. Klik **Circle**  on Sketch toolbar.

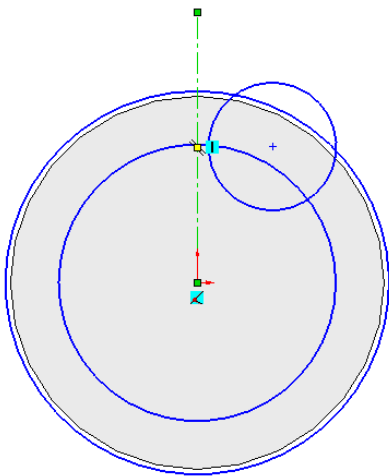
NB:

For make this circle up to you cause to put it Later We will be adjust it follow on that, don't fearing about that

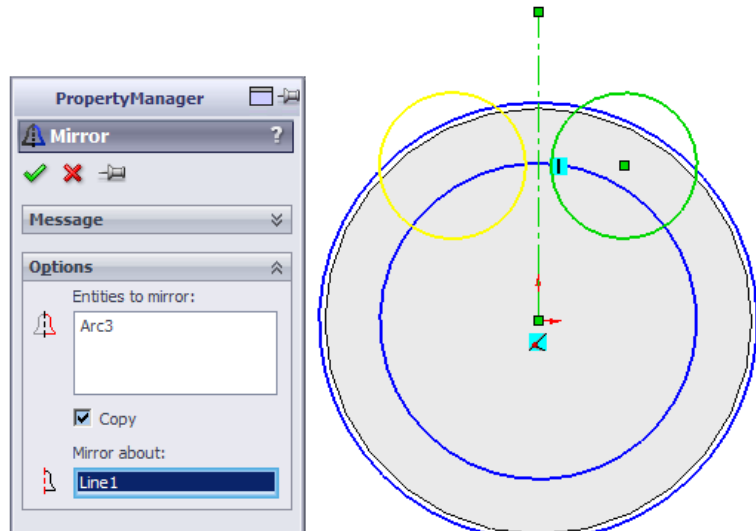
- Set **Center X Coordinate**  to **7.05**.
- Set **Center Y Coordinate**  to **12.8**.
- Set **Radius**  to **6**.



17. Klik **Centerline**  on Sketch toolbar. Make Line at Origin



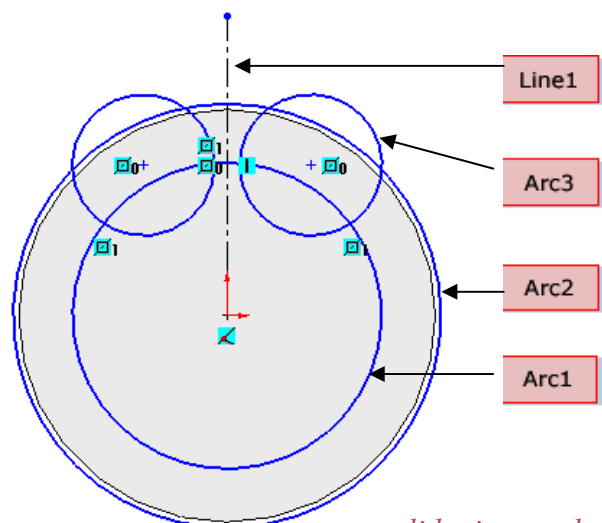
18. Klik **Cursor** on toolbar 



20. Click  OK

19. Klik **Cursor** on toolbar

1. Klik **Mirror Entities**  on Sketch toolbar.
2. **Arc3** pada **Entities to mirror:** 
3. Klik **Copy** . 
4. Pilih **Line1** pada **Mirror about:** . 
5. Click OK 



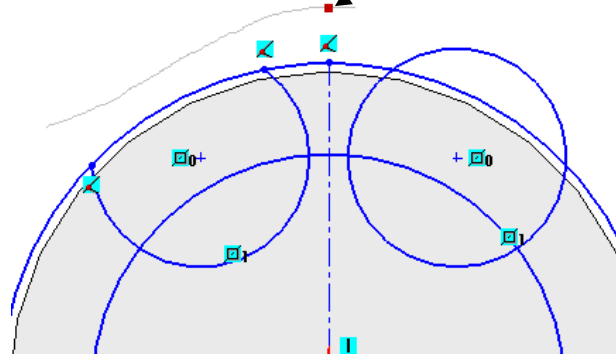
Remove line with Trim Entities

1. Click **Trim Entities** on Sketch toolbar.

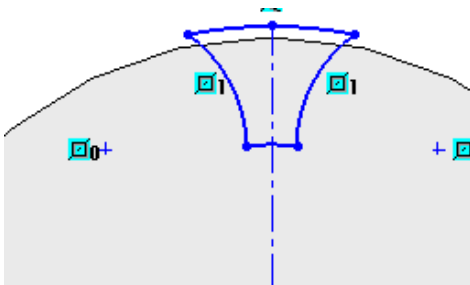
- In Property Manager Click **Power Trim** 

To trim entities, hold down and drag your cursor across the entities, or pick on an entity and then pick on a bounding entity or anywhere on the screen. To extend entities, hold down the shift key and drag your cursor across the entities.

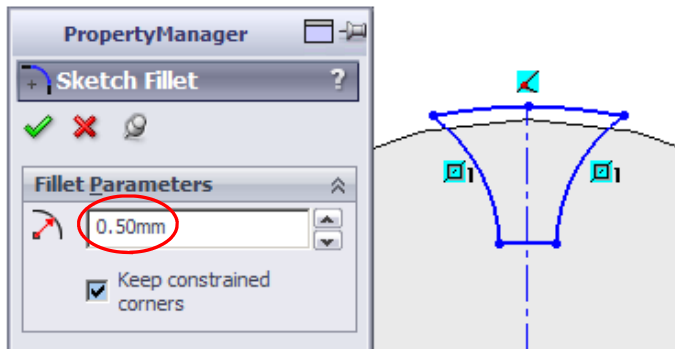
Path parameter to removed unused line, don't forget to hold Mouse to used Trim



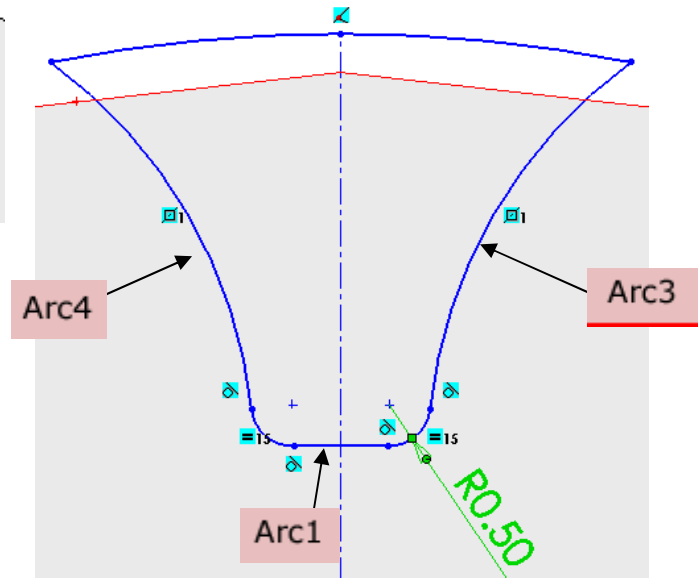
Make path like as bellow



2. Click **Sketch Fillet**  on Sketch toolbar.



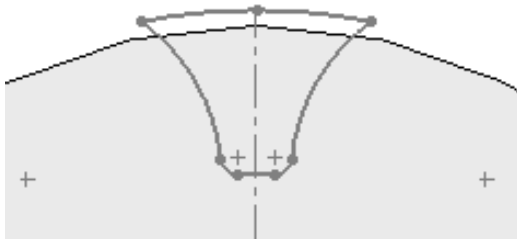
3. click **Arc1** and **Arc3** on sketch entity, than click **Arc1** and **Arc4**. Klik **OK**



On Property Manager, in fillet **Parameters**:

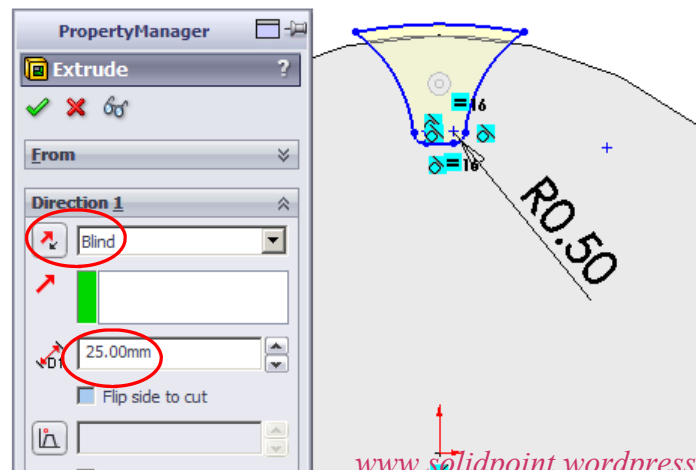
- Set **Radius** ke **0.5**.

4. Click **Exit Sketch**

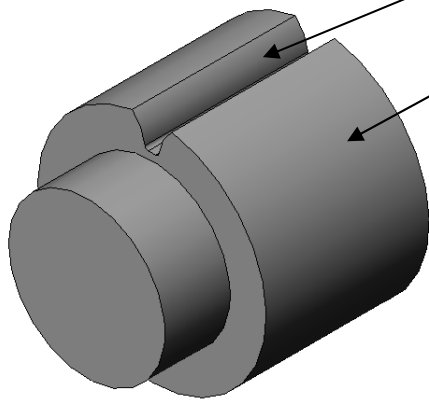


5. Click **Extrude Cut**

Insert in Direction 1 BLIND
Set Depth 25



6. View with ISOMETRIC



Cut-Extude1

Extude1

Copy with circle Cut

1. Click AXIS



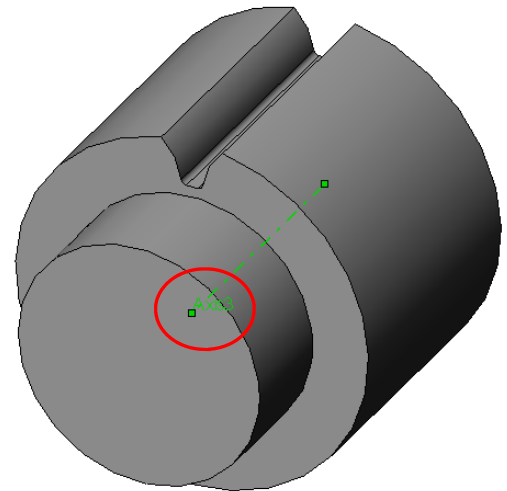
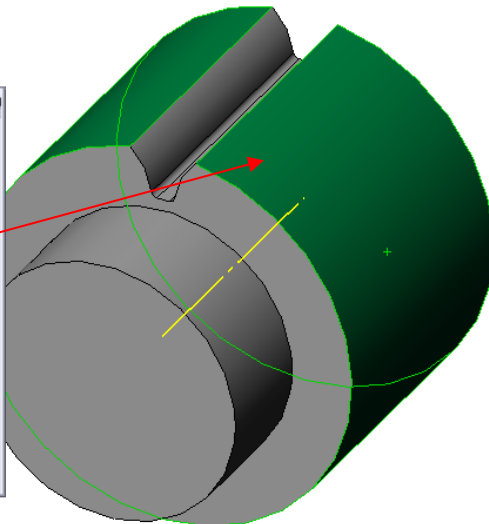
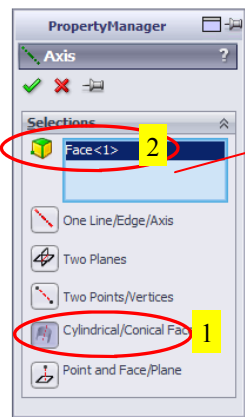
• Click Cylindrical/Conical Face .



• select **Extrude1** pada **Reference Entities**



Then Klik **OK** . This will be make Axis



2. Click **Circular Pattern**



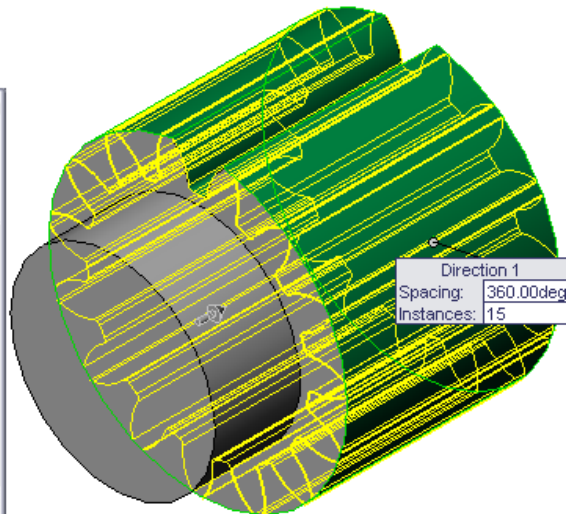
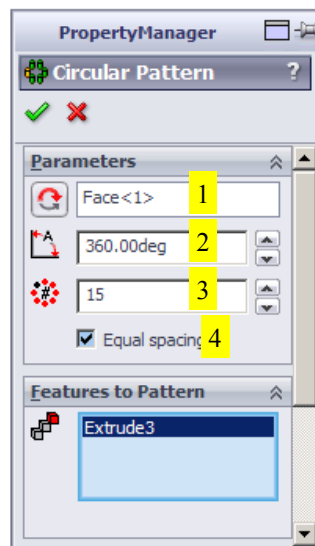
In Property manager chose

1. Click **Axis 1** in **pattern Axis**
2. set **Angle** to **360**.
3. Set Number of **Instances** to **15**.
4. Click **Equal spacing**

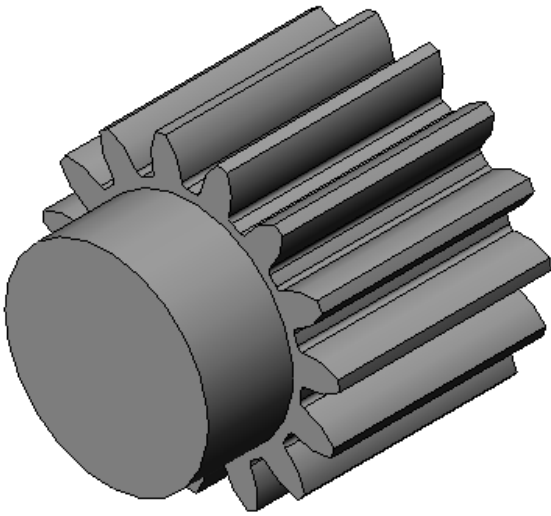
Blellow **Features to Pattern**:



- select **Cut-Extrude1** pada **Features to Pattern**



Then Click **OK** 



4. create **Helical** with command **Flex Twisting**

- Select **Insert, Feature, Flex** 

In property manager select

1.select Solid Body<1> in **Bodies for Flex**

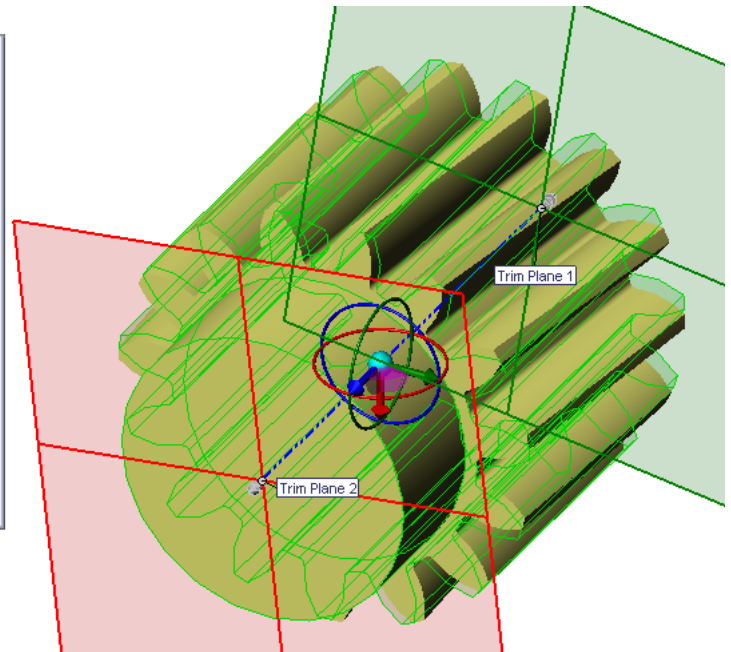
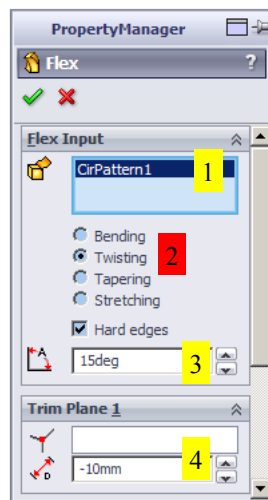
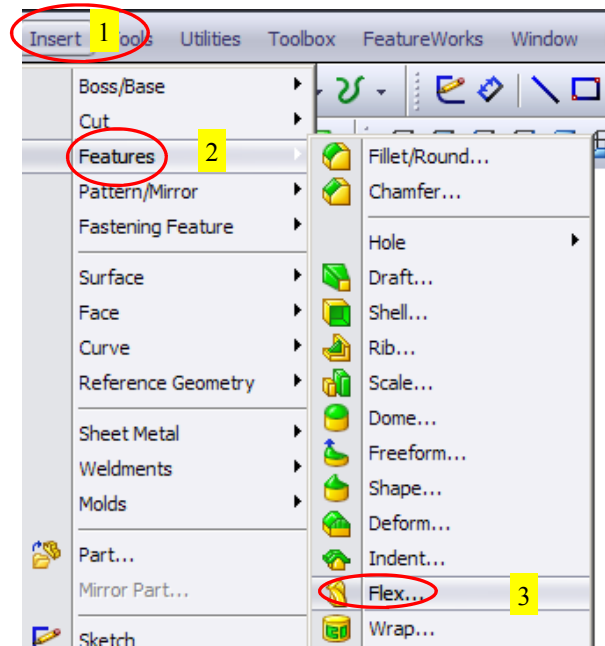
2. Click **Twisting** (don't forget this very important)

3. Set Angel  to 15

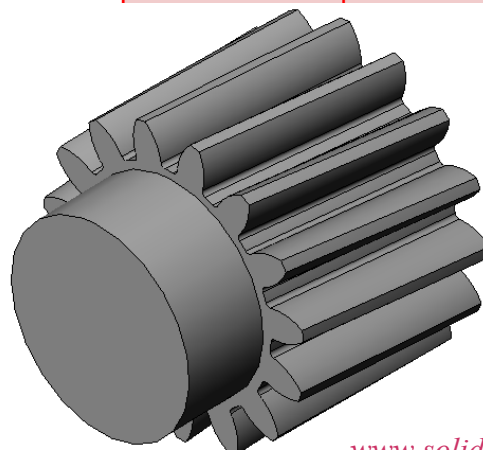
Bellow Trim Plane 1

4.Set Distance  to **-10**.

This point a part of which does not twisting. You can minus (-) point or plus (+) point depend direction of part



Then Click **OK** 



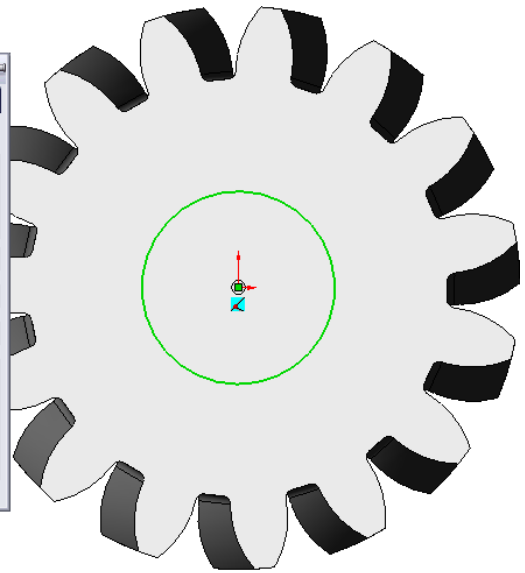
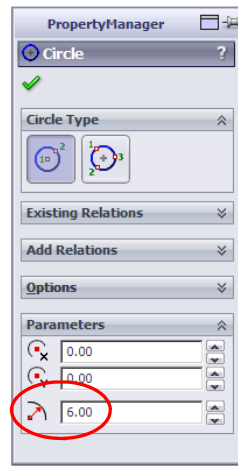
5. Make Axle hole

Extruded Cut  on Feature toolbar.

1. Select **Back** in standard View
2. Select **surface** of part
3. Select **Circle** in toolbar
4. Make hole sketch in **Origin Point**

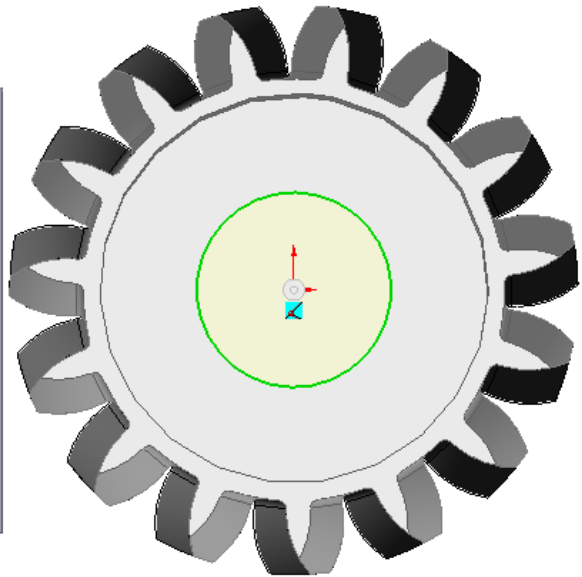
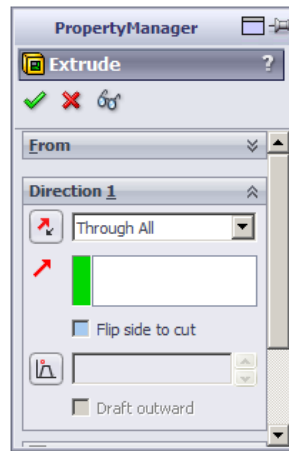
- In property manager set radius in 6 

5. Exit Sketch 

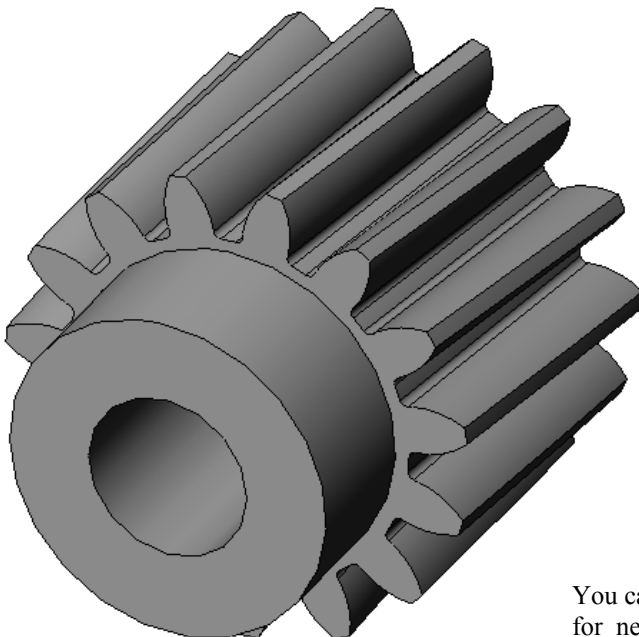


In property Manager under **Direction 1**

- Select **Through All** in **End Condition** 



Then Click **OK** 



Well done you have finished

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