

Gear and Shaft Analysis

The gears that were assembled in the previous lecture will be analyzed along with shafts added to the model, figure (1)

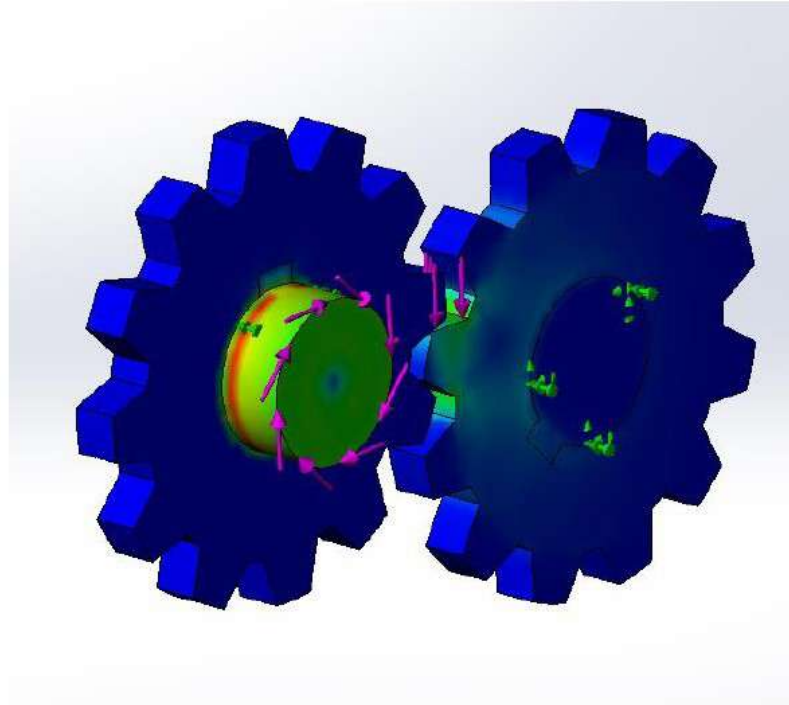


Figure (1)

- ⇒ Start by creating the shafts
- ⇒ On the front plane sketch figure (2)

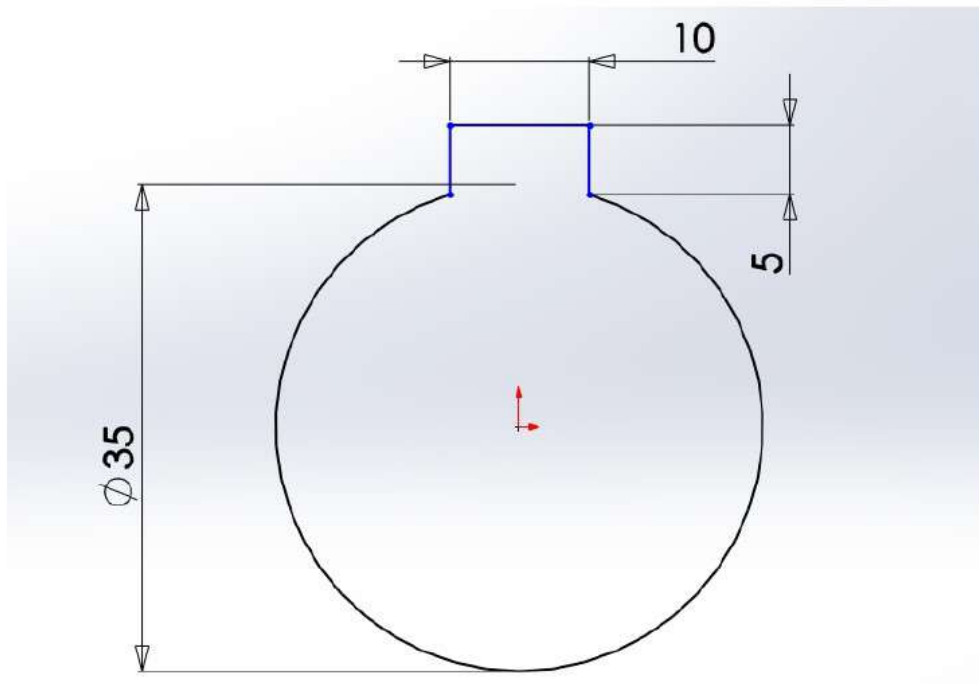


Figure (2)

- ⇒ Extrude to 10 mm
- ⇒ Save as shaft one
- ⇒ Open another part
- ⇒ Make the same previous sketch on the front plane
- ⇒ Extrude to 20 mm
- ⇒ On the back surface sketch 35 mm circle, figure (3)

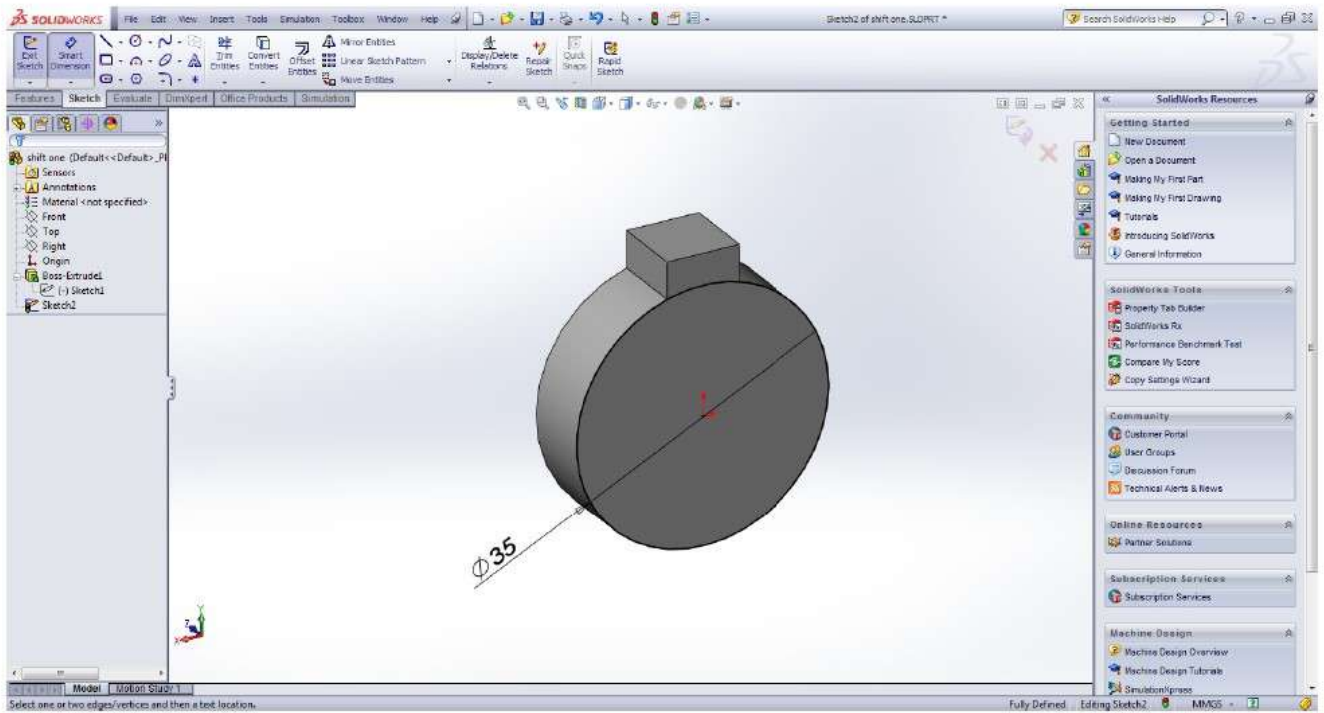


Figure (3)

- ⇒ Extrude to 15 mm
- ⇒ Save as shaft two, figure (4)

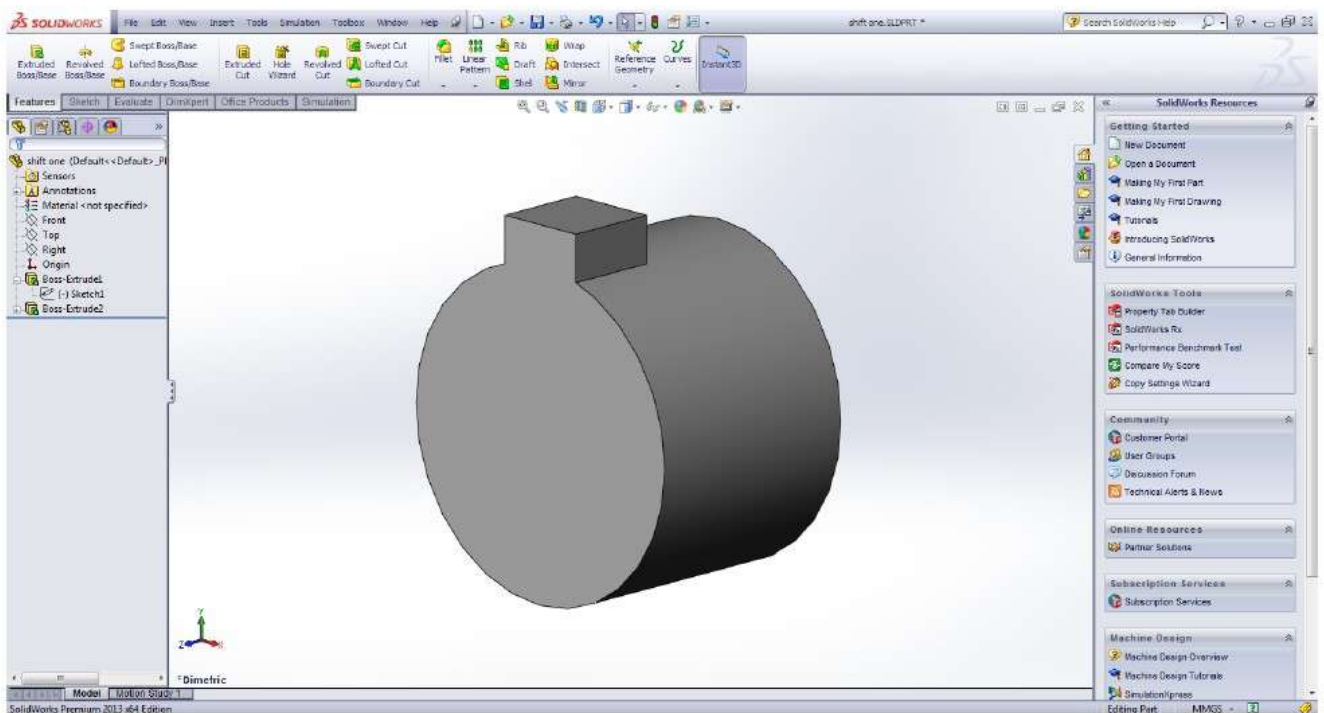


Figure (4)

OPEN the assembled gears from the previous lecture

To make the rest of the assembly make the two gears fix for NOW

- ⇒ From assembly toolbar select insert components
- ⇒ Browse for shaft one and shift two
- ⇒ Select Mate from assembly toolbar
- ⇒ Select face of shaft one and the face of one of the gears
- ⇒ Click on coincident, as figure (5)

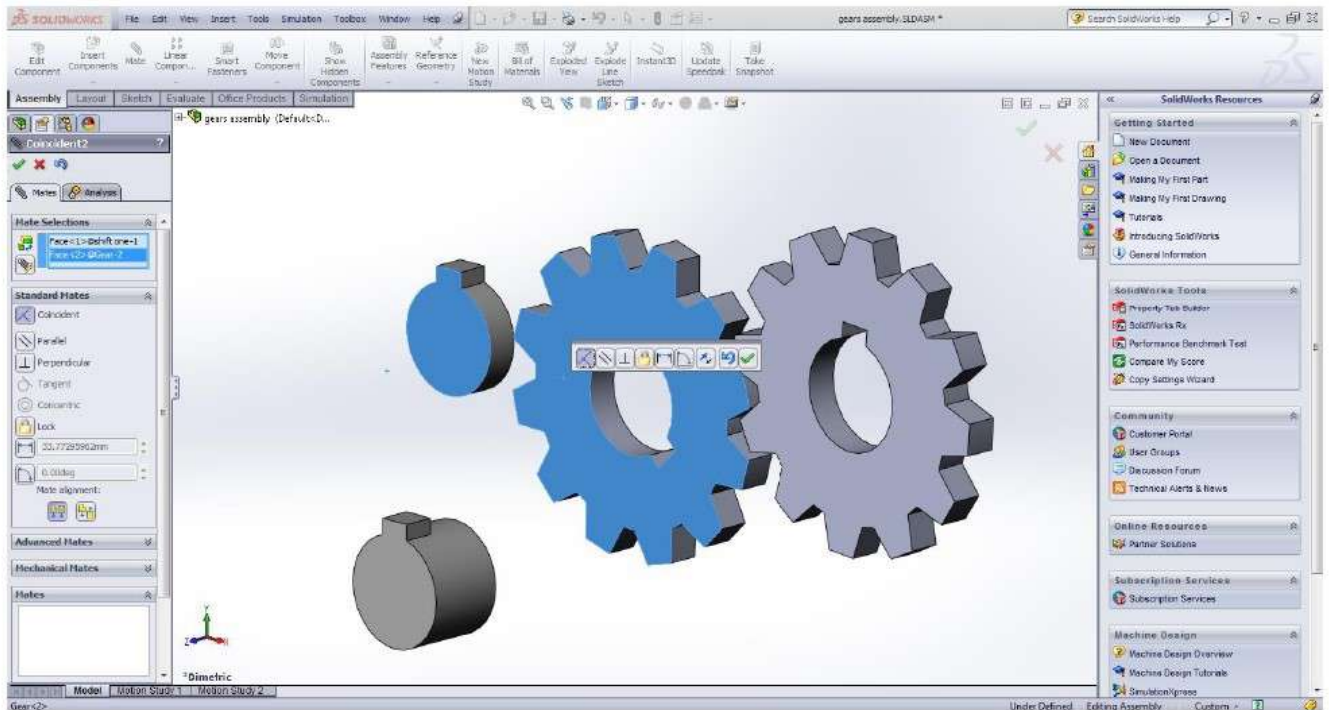


Figure (5)

- ⇒ Click OK
- ⇒ Click on the surfaces illustrated in figure (6)
- ⇒ Make coincident mate
- ⇒ Click on Anti-Aligned if needed, as figure (7)

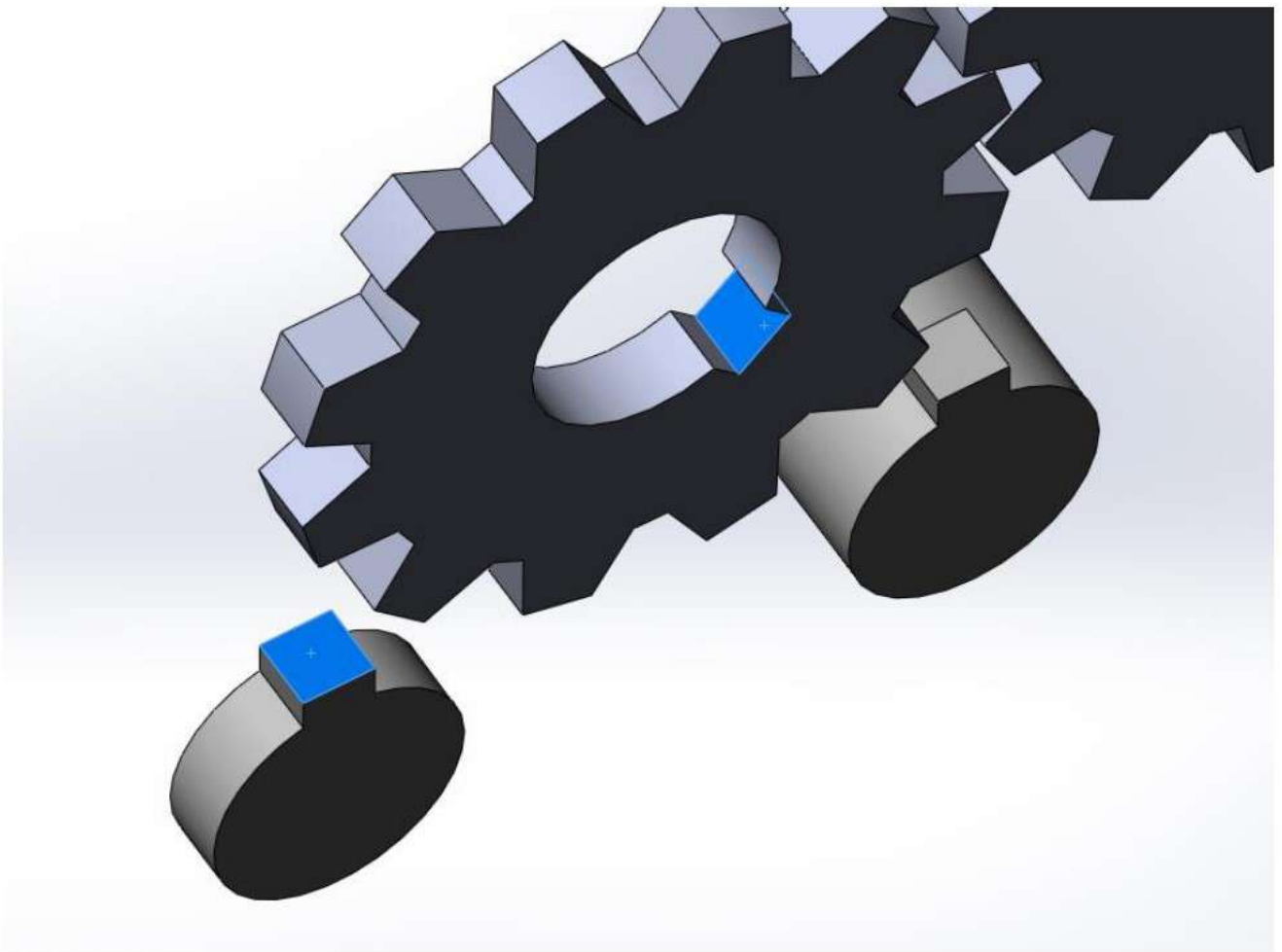


Figure (6)

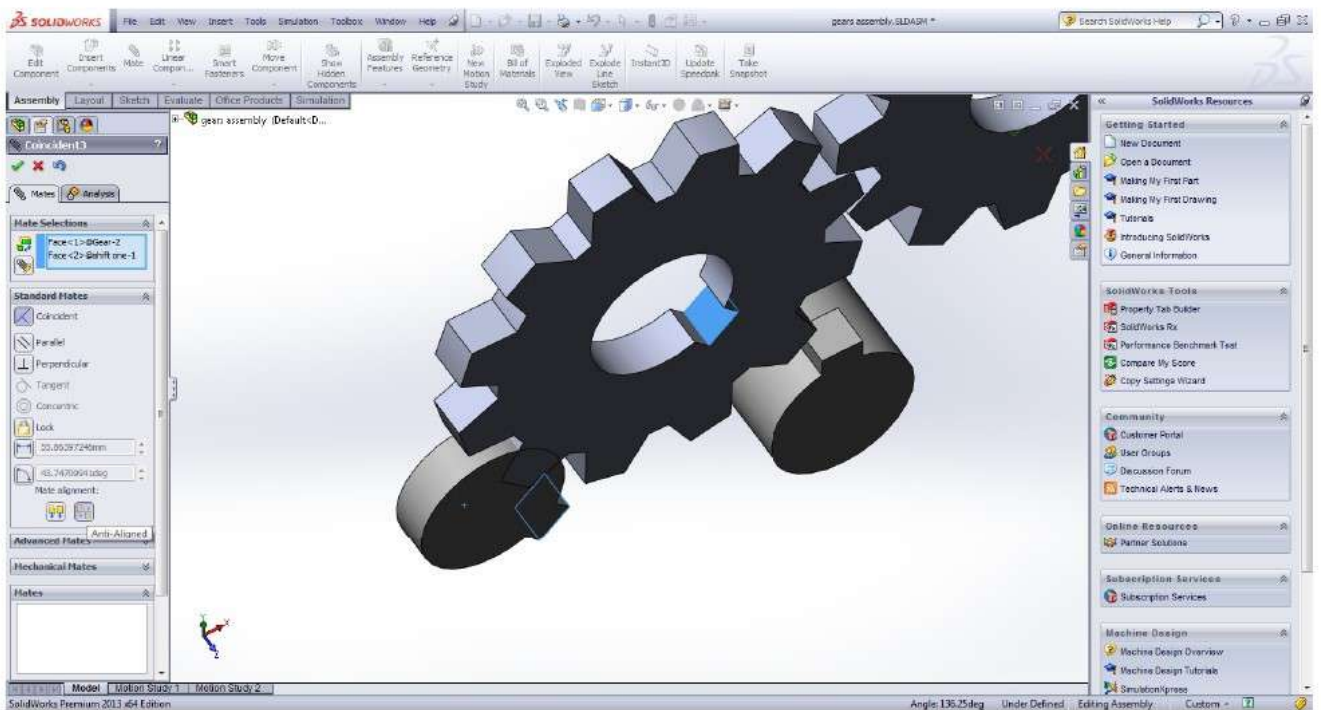


Figure (7)

- ⇒ Click OK
- ⇒ Select the circular edge of the shaft and the gear
- ⇒ Click on concentric mate

⇒ Click OK to figure (8)

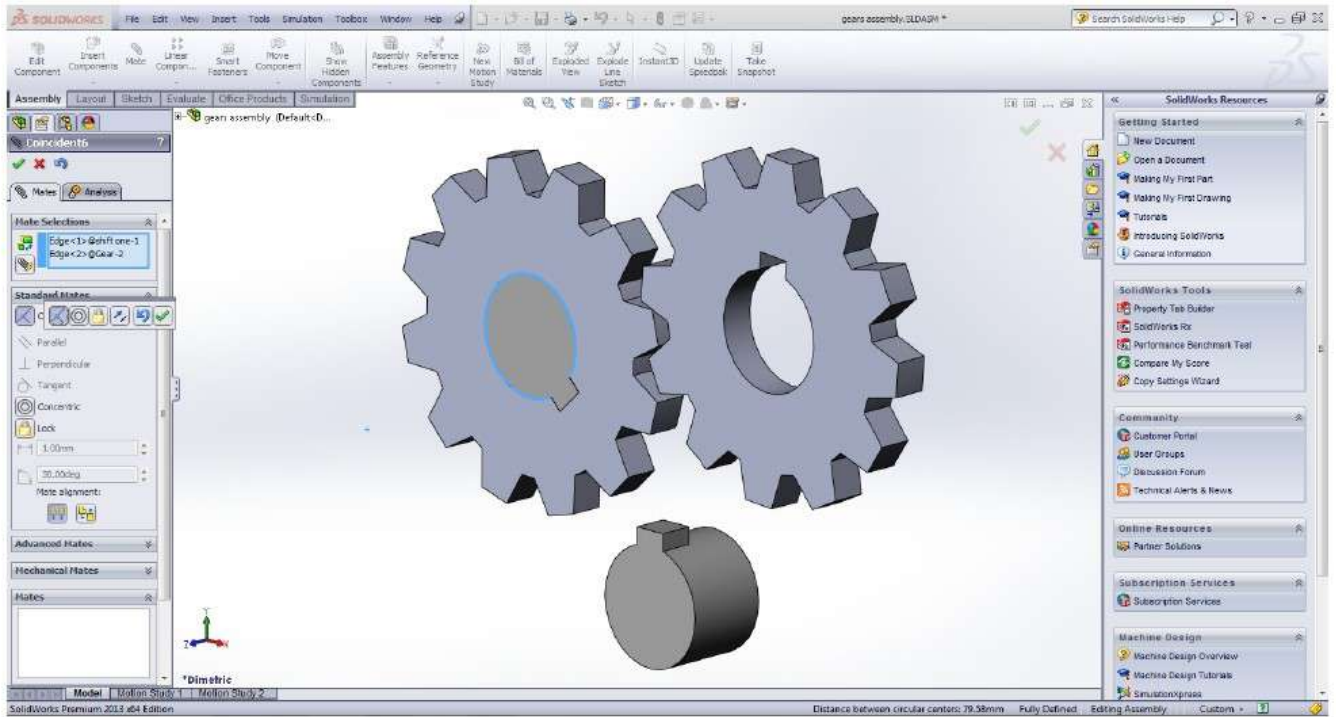


Figure (8)

⇒ Assembly shaft two the same way as shaft one

⇒ Remove the fix from the two gears, right click on the part and select float, figure (9)

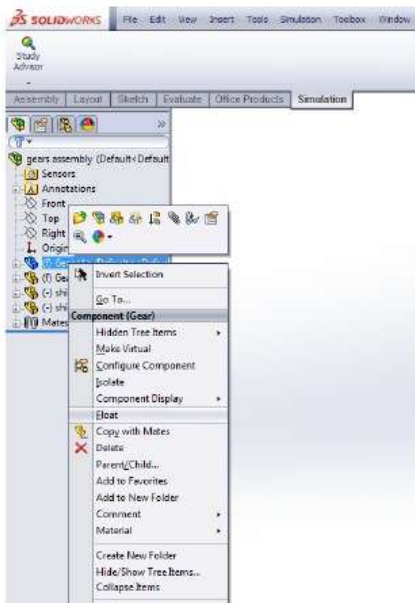


Figure (9)

From simulation toolbar, select new study

⇒ Open static study

⇒ Set the material as Alloy steel

⇒ Right click on shaft one and select Hide components, figure (10)

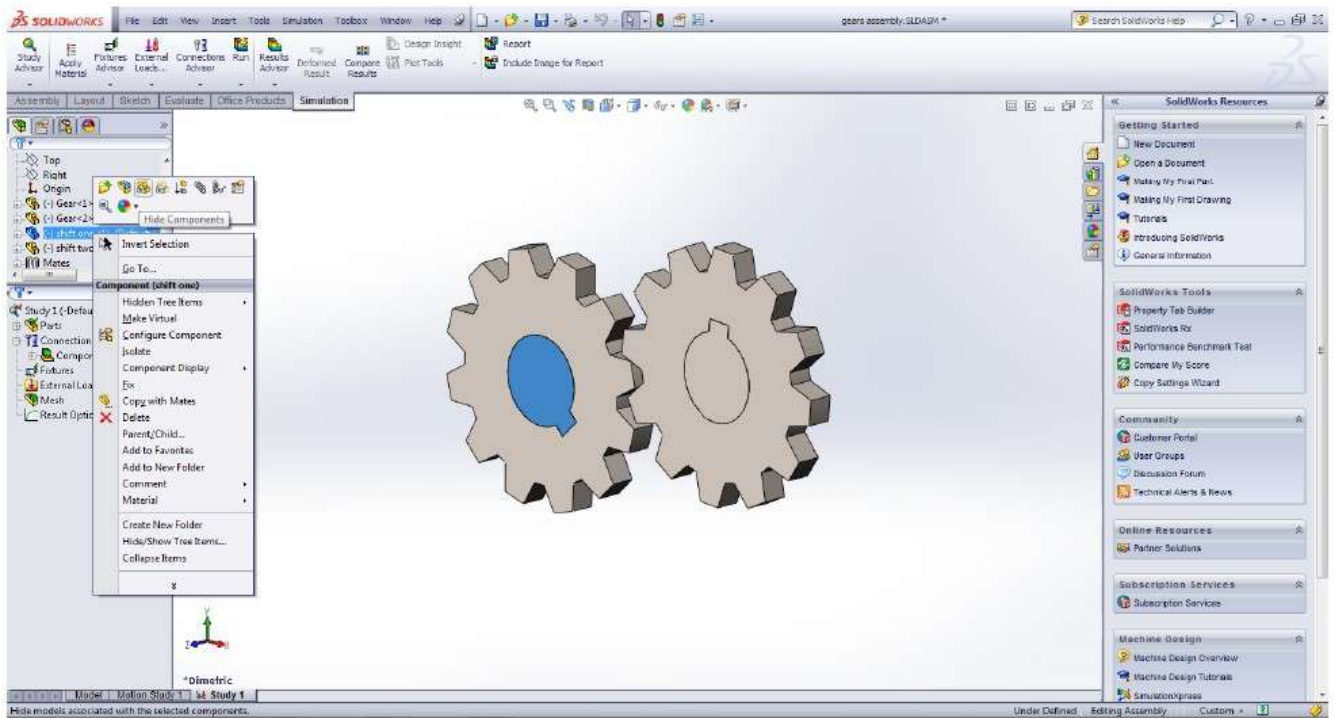


Figure (10)

- ⇒ Hide shaft two as well
- ⇒ Right click on fixtures and select fixed geometry
- ⇒ Select the cylindrical surfaces of the two gears
- ⇒ Click OK to figure (11)

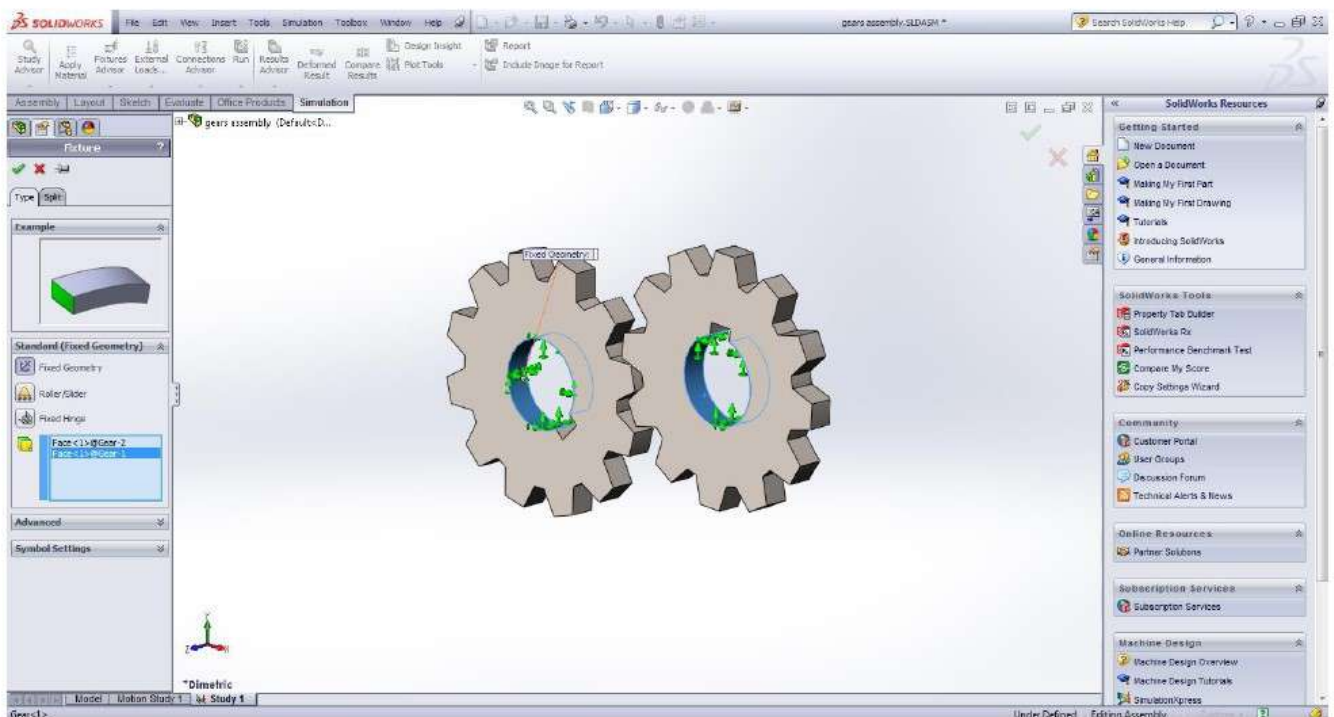


Figure (11)

- ⇒ Right click on the shaft and select show components, figure (12)

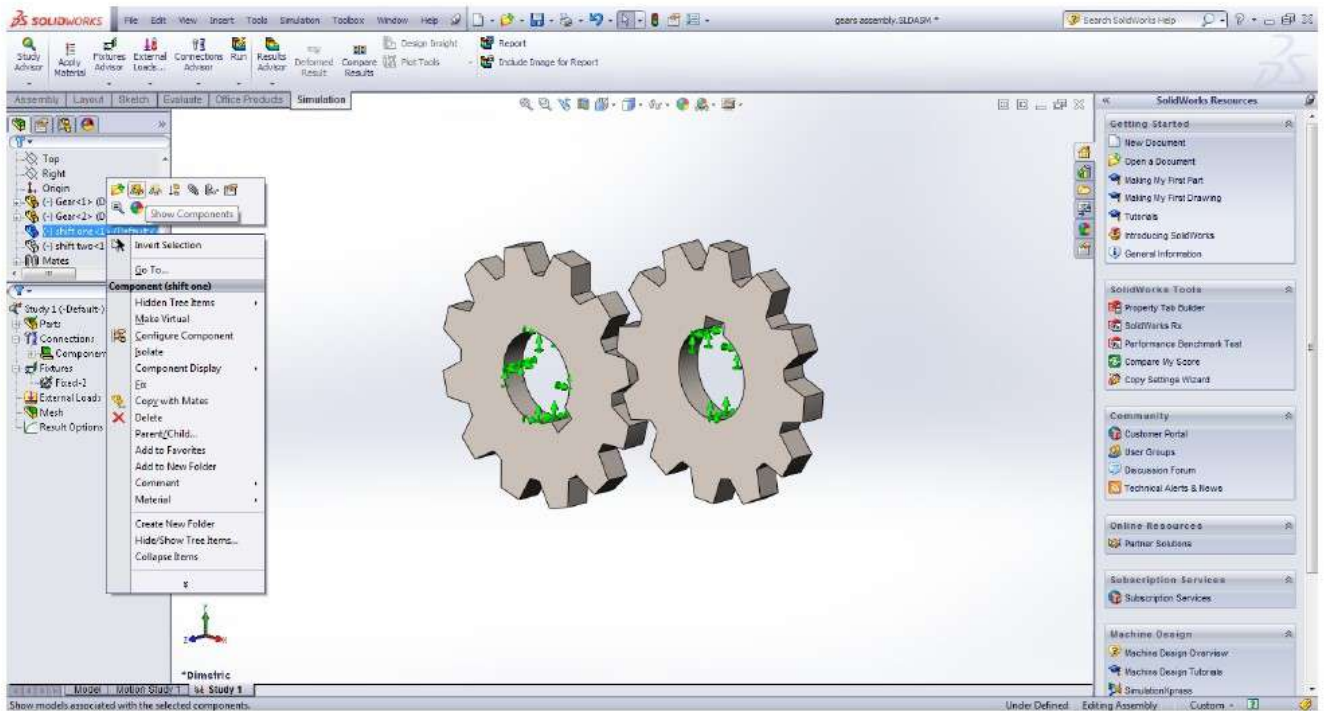


Figure (12)

- ⇒ Show the other shaft as well
- ⇒ Right click external loads and select Torque
- ⇒ Select the circular face as face for torque
- ⇒ Select the cylindrical surface for direction, figure (13)

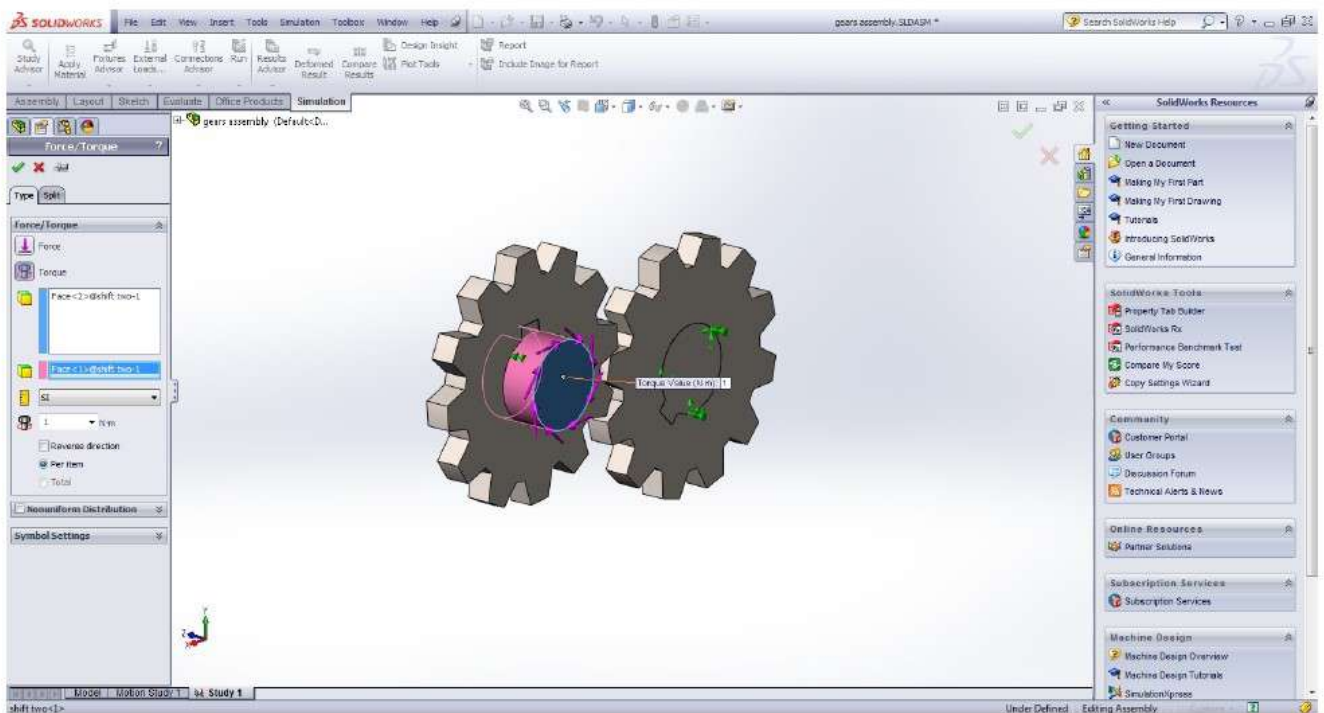


Figure (13)

- ⇒ Set the torque value as 100 Nm
- ⇒ Right click on mesh and select mesh and run
- ⇒ Right click on the stress results and select Deformed result, figure (14)

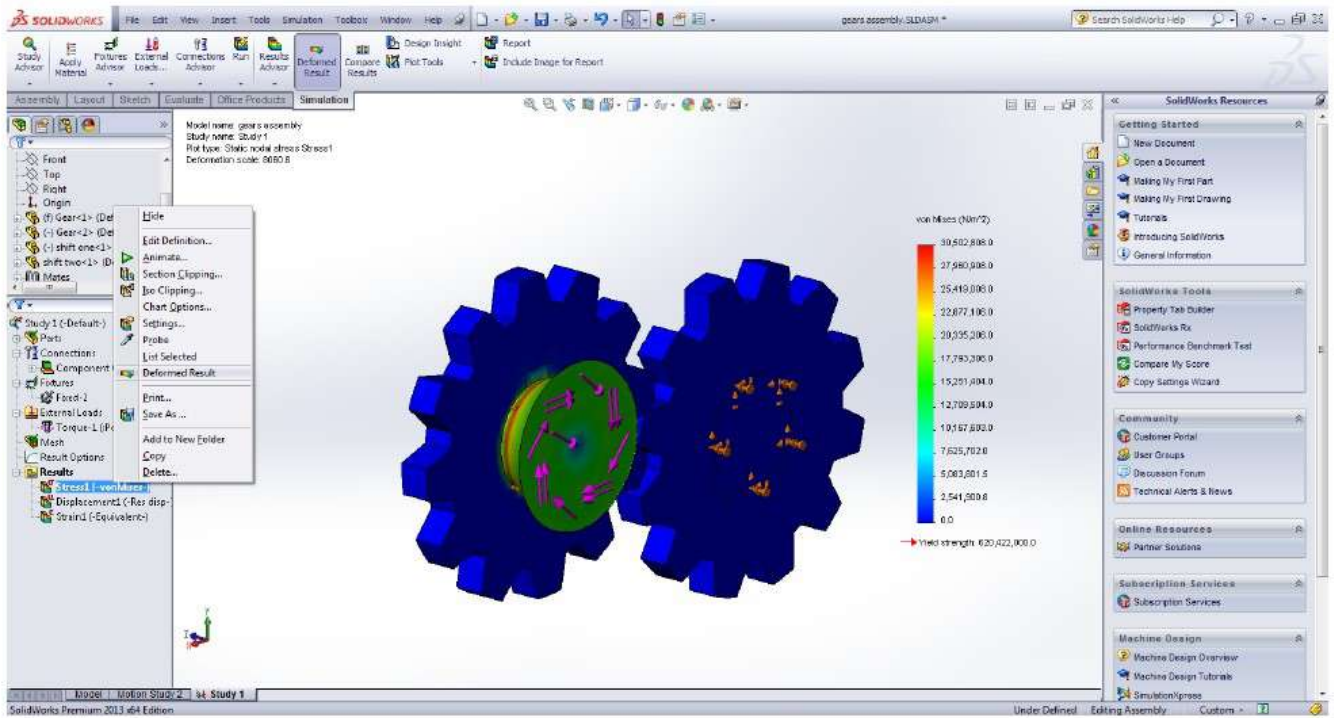


Figure (14)

⇒ The results should be as figure (15)

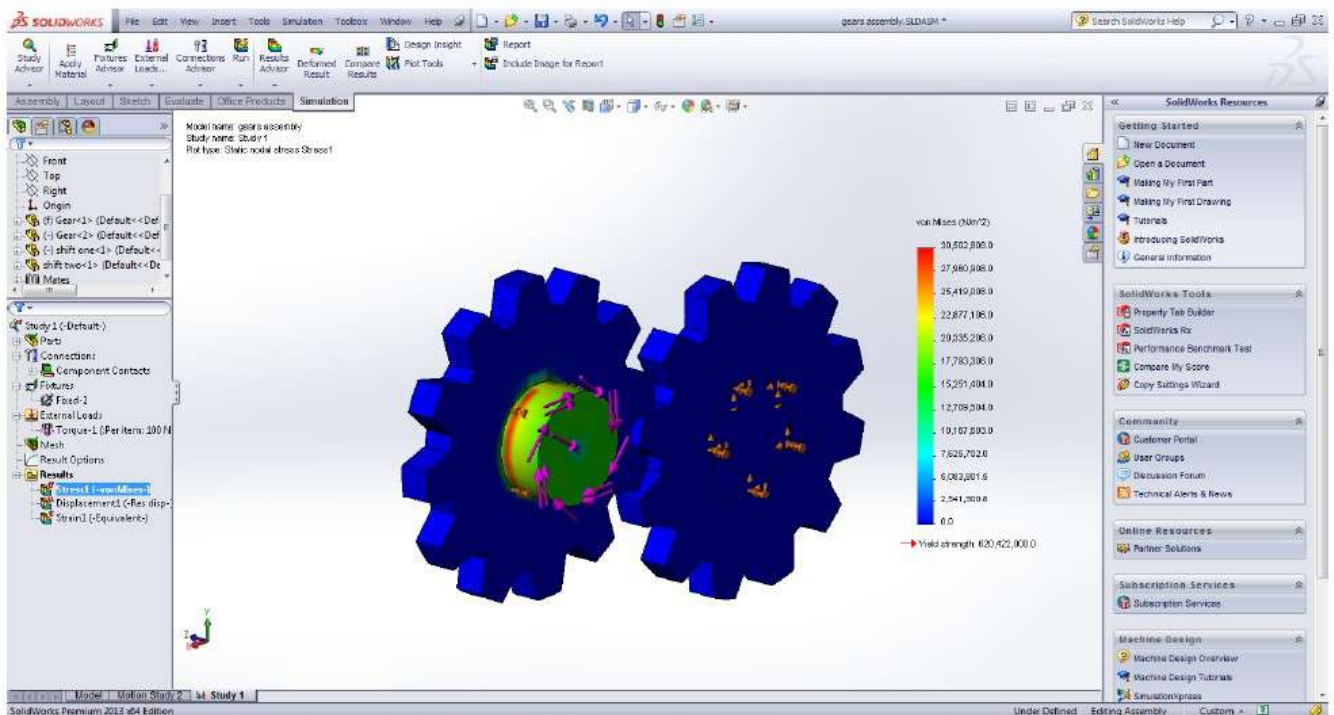


Figure (15)

- ⇒ To add another load to the model
- ⇒ Right click on external loads and select force
- ⇒ Select the gear teeth surface as shown in figure (16)

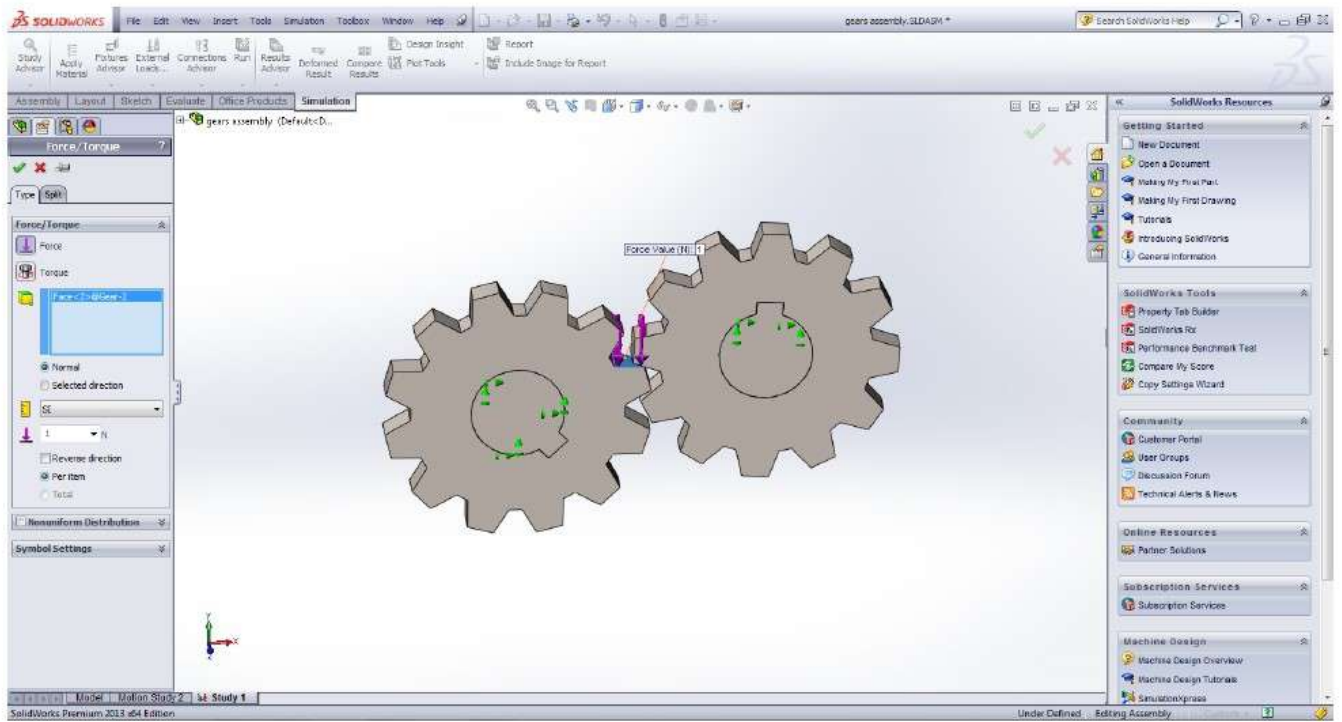


Figure (16)

- ⇒ Set the force value as 1000 N
- ⇒ Click OK
- ⇒ Right click on mesh and select mesh and run
- ⇒ The stress results should be as figure (17),(18)

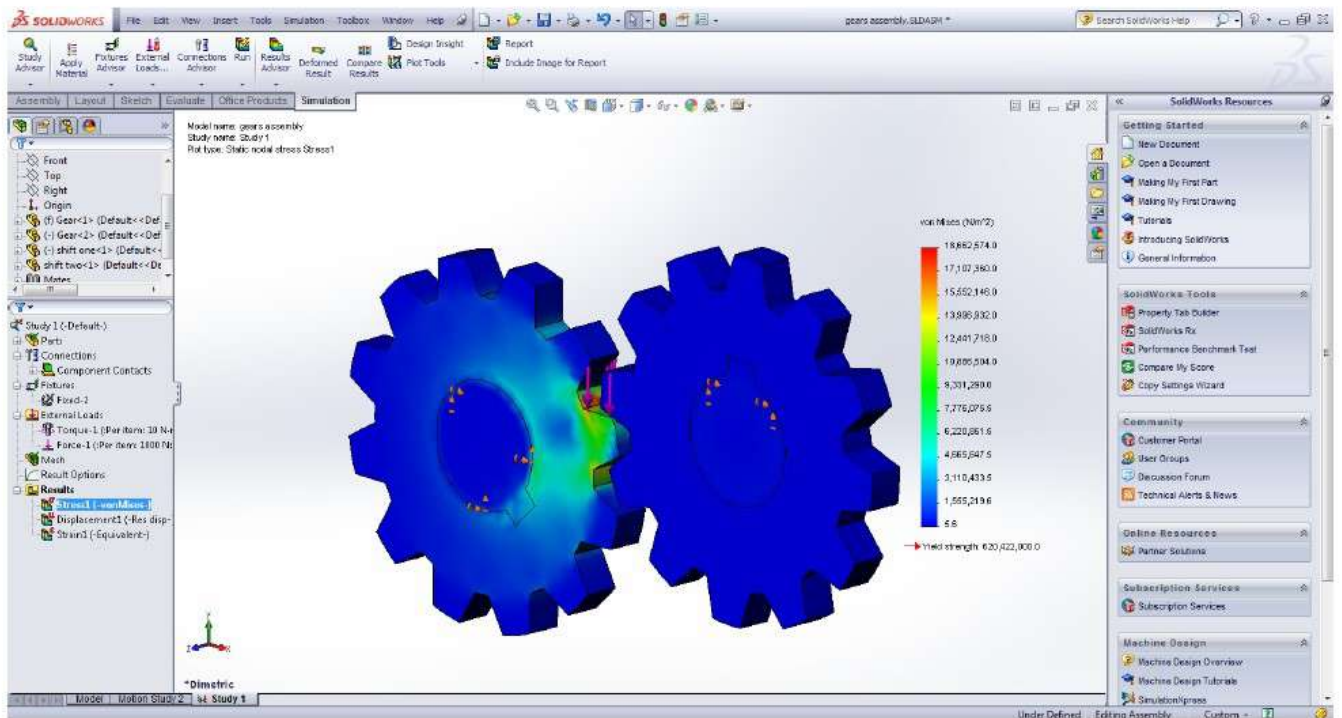


Figure (17)

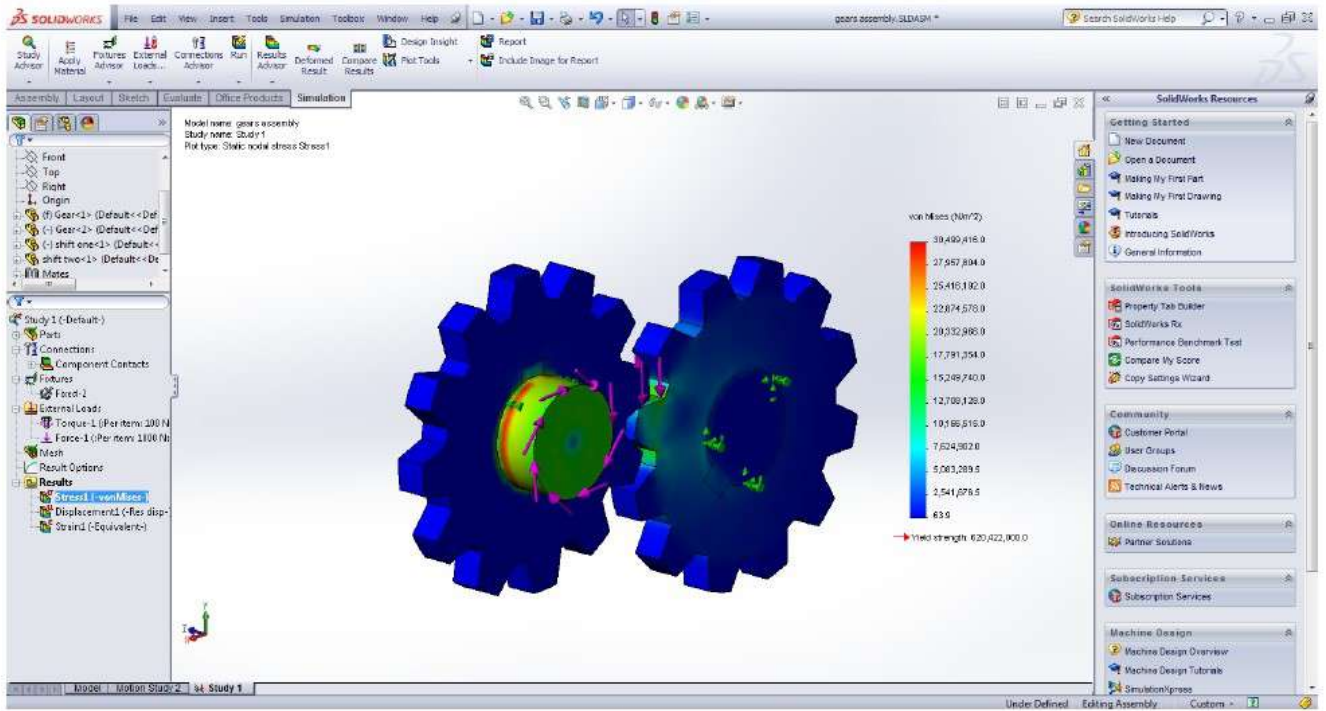


Figure (18)

H.W

On you on Use reasonable dimensions

