

## Fatigue Study

It is observed that repeated loading and unloading weakens objects over time even when the induced stresses are considerably less than the allowable stress limits. This phenomenon is known as fatigue. Each cycle of stress fluctuation weakens the object to some extent. After a number of cycles, the object becomes so weak that it fails. Fatigue is the prime cause of the failure of many objects, especially those made of metals.

For this lecture a Fatigue study will be performed for the geometry shown in figure (1) as a torque force is applied to it.

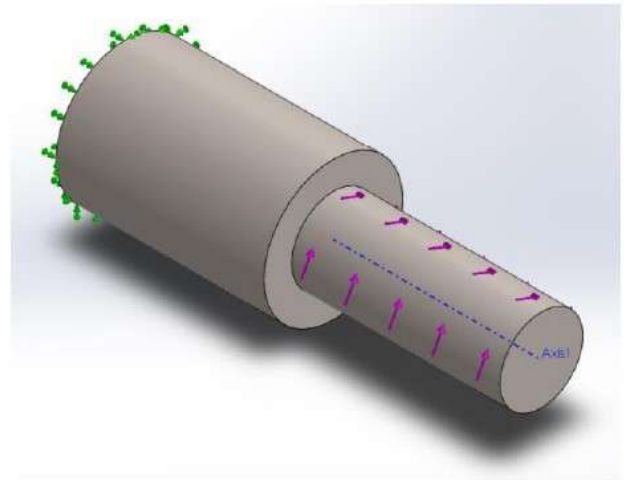


Figure (1)

- ⇒ On the right plane sketch 100 mm circle
- ⇒ Extrude to 150 mm, figure (2)

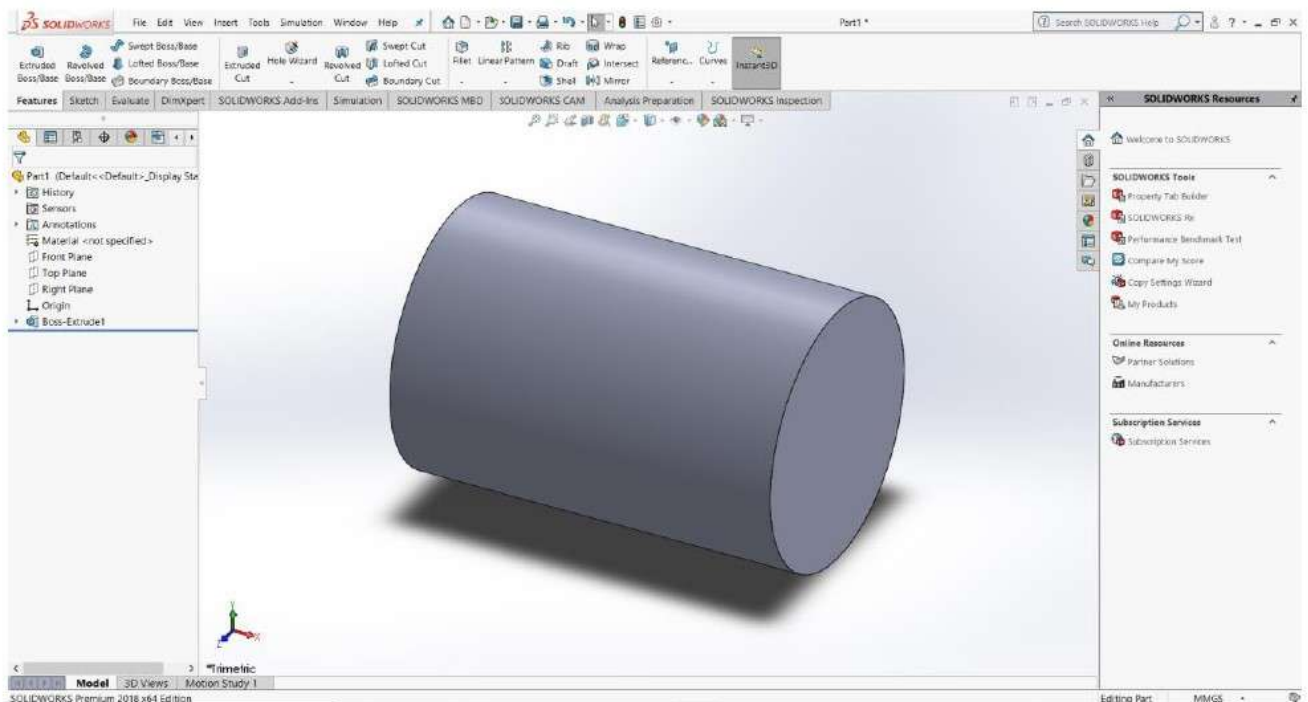


Figure (2)

- ⇒ Sketch 60 mm circle on the face of the cylinder you just created
- ⇒ Extrude to 150 mm, figure (3)

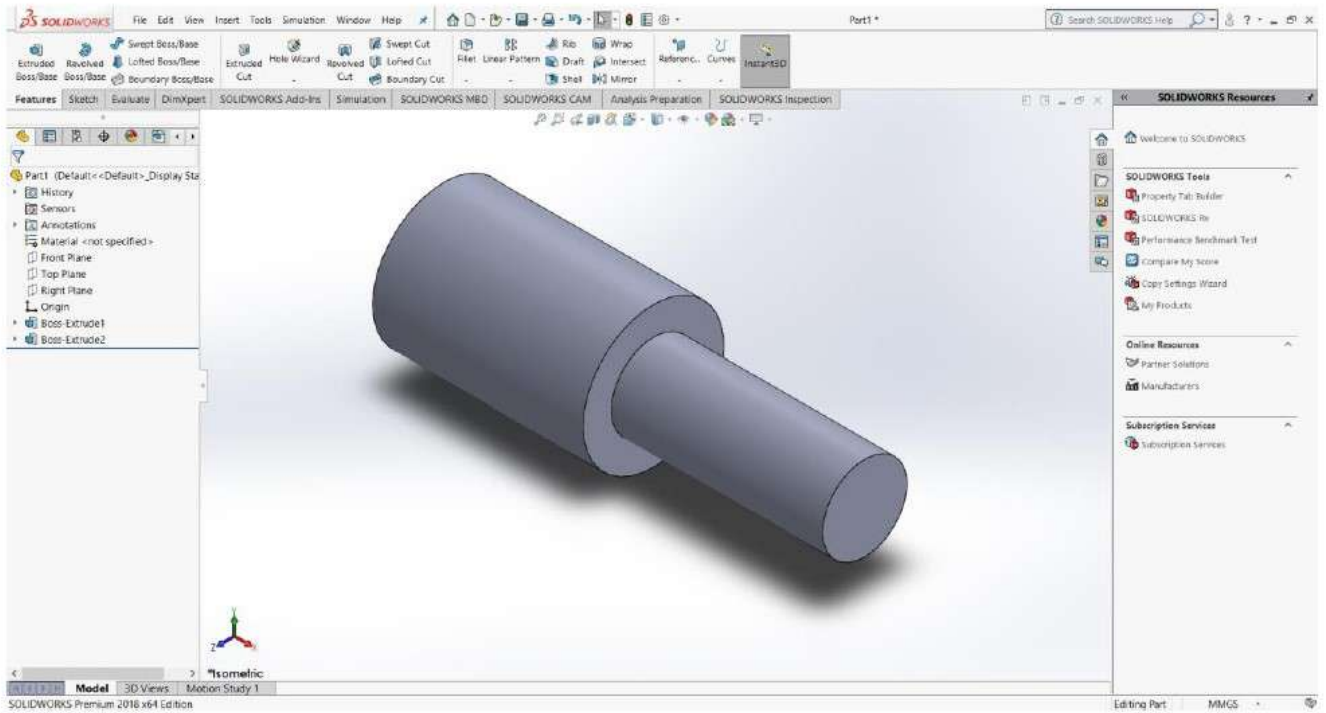


Figure (3)

- ⇒ From features toolbar, reference geometry select Axis, figure (4)

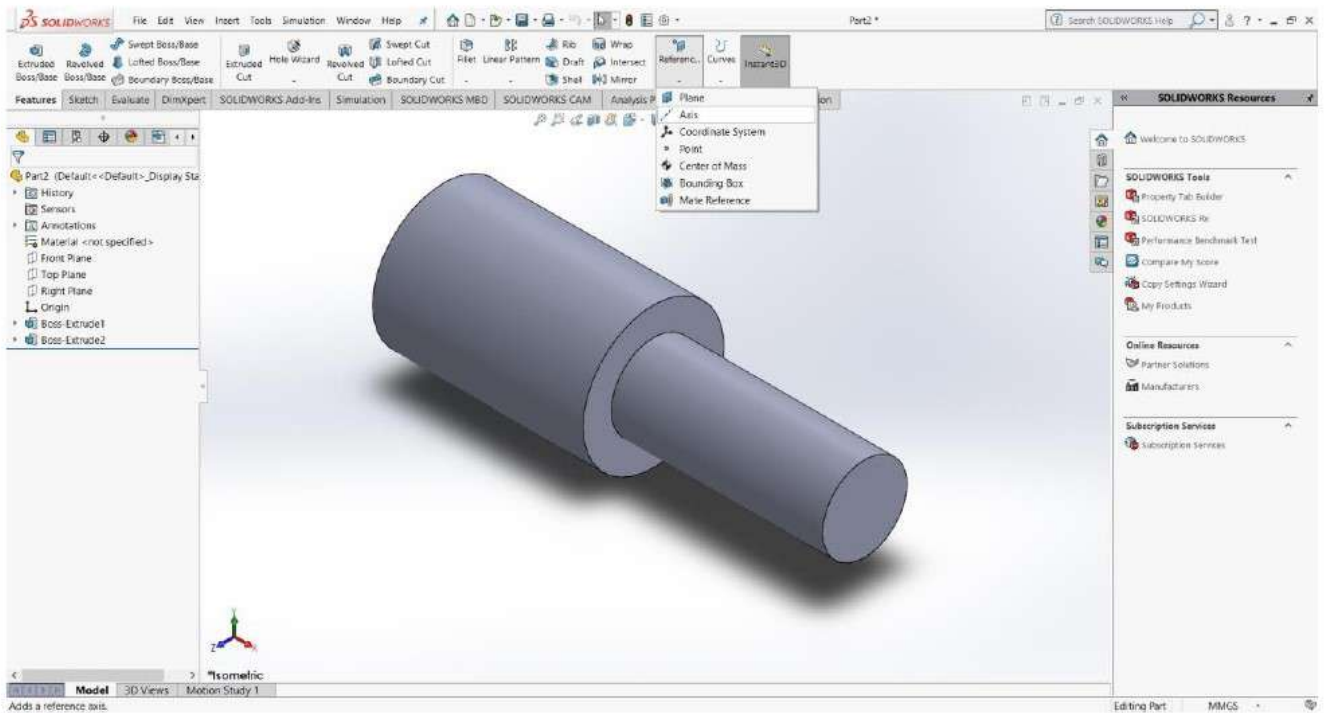


Figure (4)

- ⇒ Click cylindrical and select one of the cylindrical surface faces, figure (5)
- ⇒ Click OK

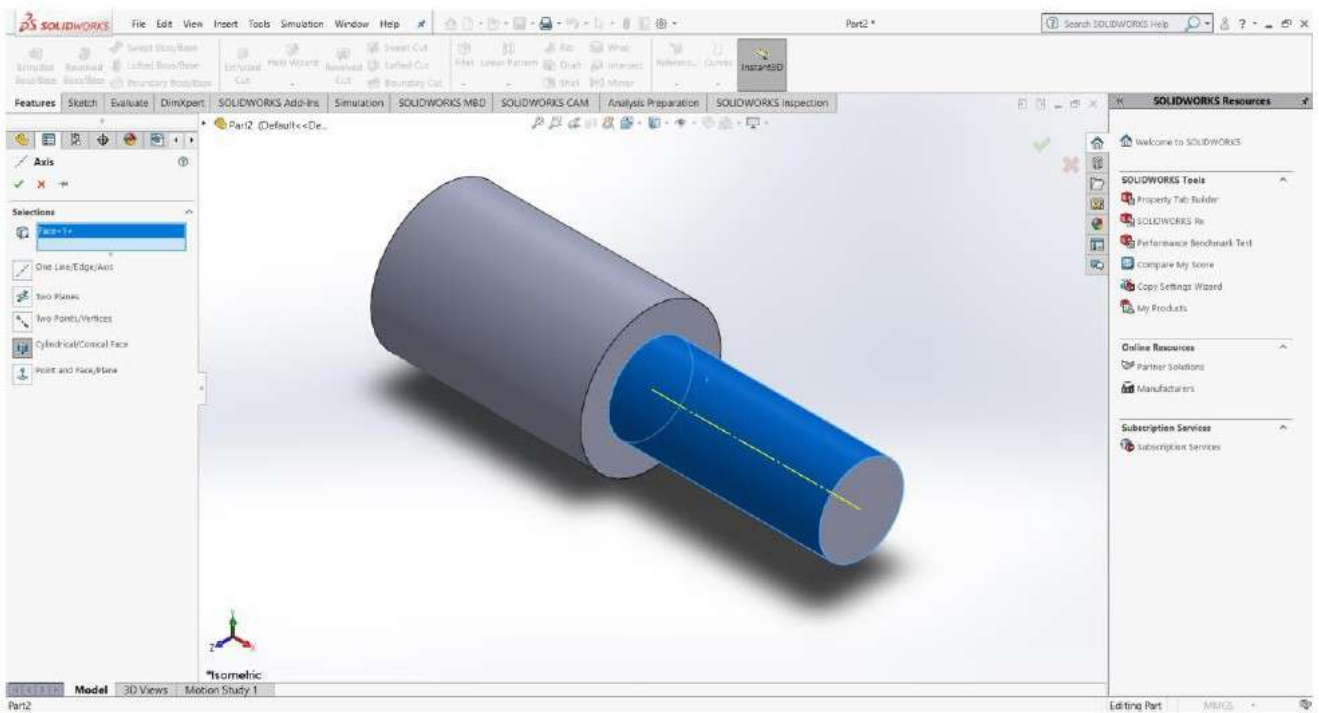


Figure (5)

- ⇒ Open New Static study
- ⇒ Call it Study 1, figure (6)

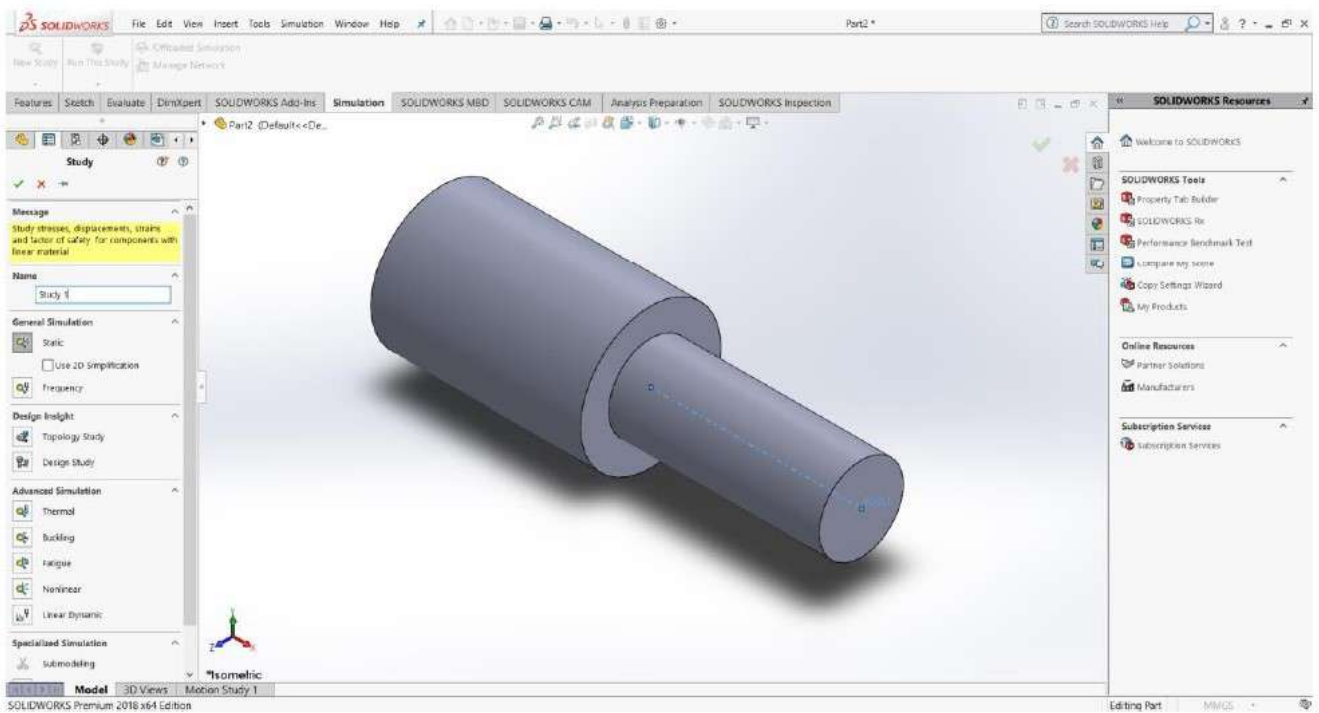


Figure (6)

- ⇒ Set the material as Alloy Steel
- ⇒ Fix the base of the geometry, figure (7)

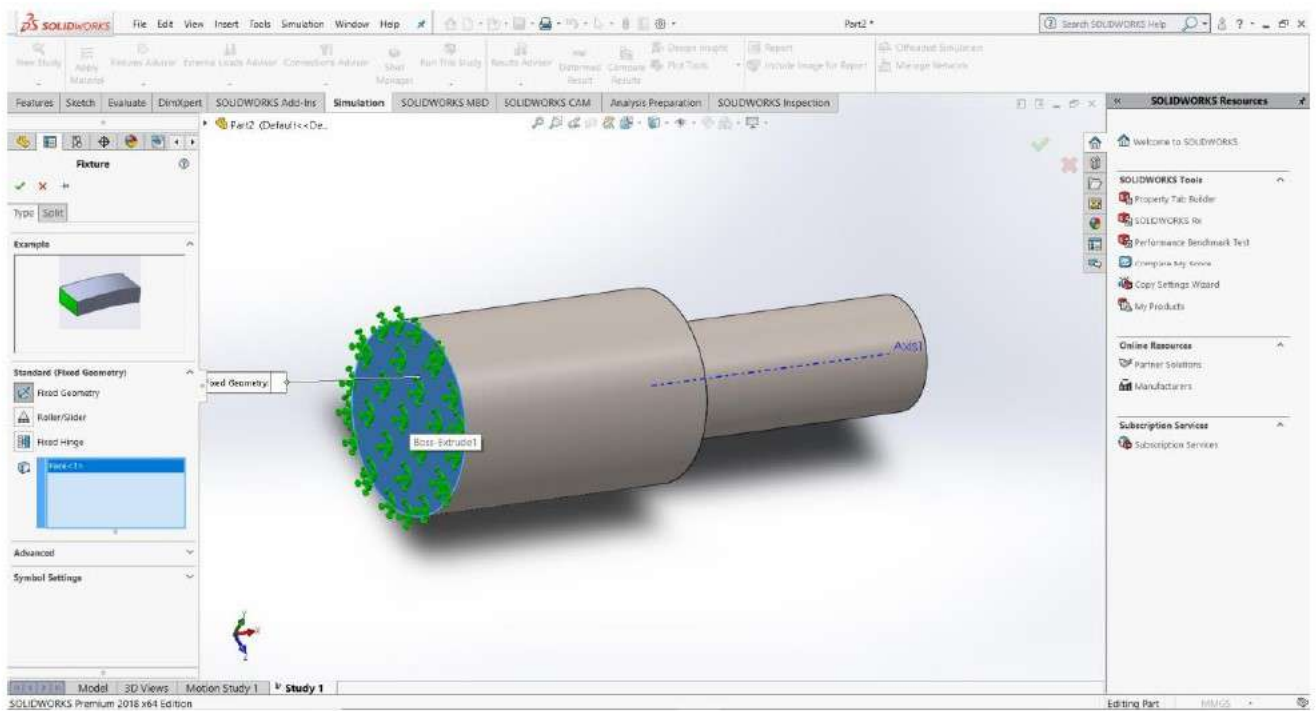


Figure (7)

- ⇒ Right click on external loads and select Torque
- ⇒ Select the face of 60 mm diameter cylinder
- ⇒ Select the axis for direction, figure (8)

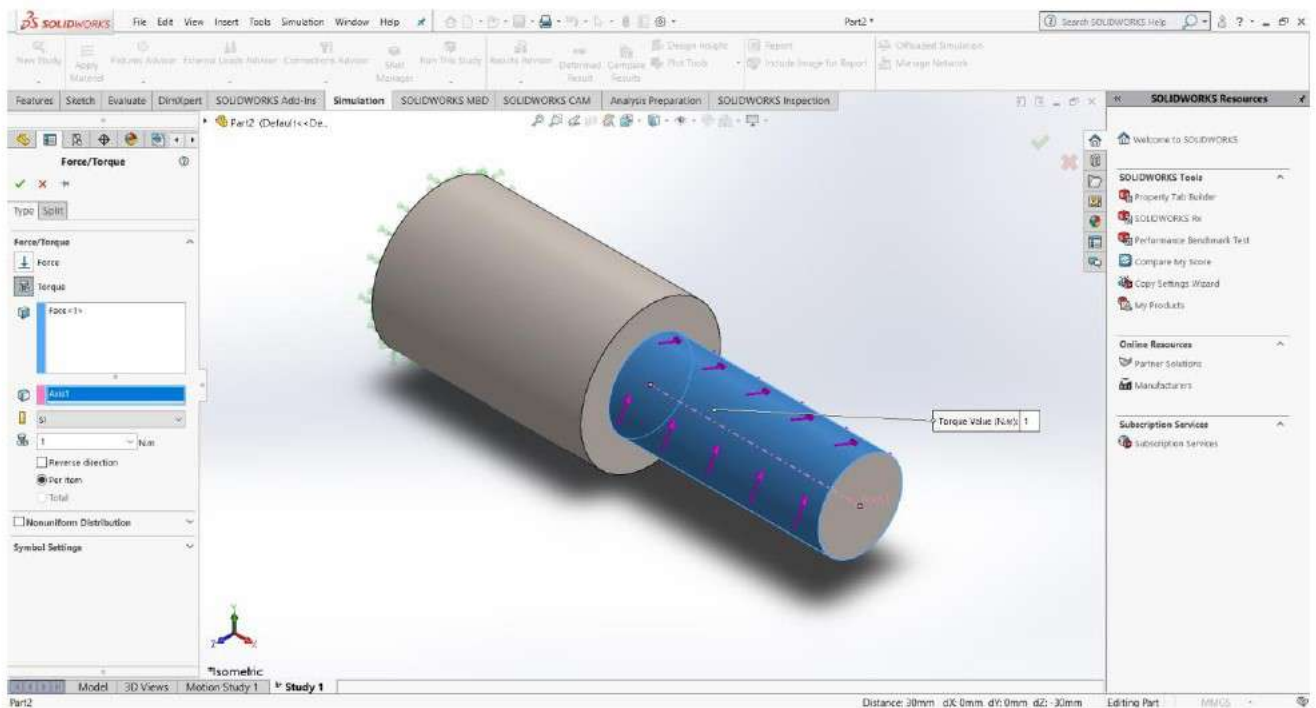


Figure (8)

- ⇒ Set the value of torque as 3000 N.m
- ⇒ Click OK

Mesh and Run the study

The stress result are as figure (9)

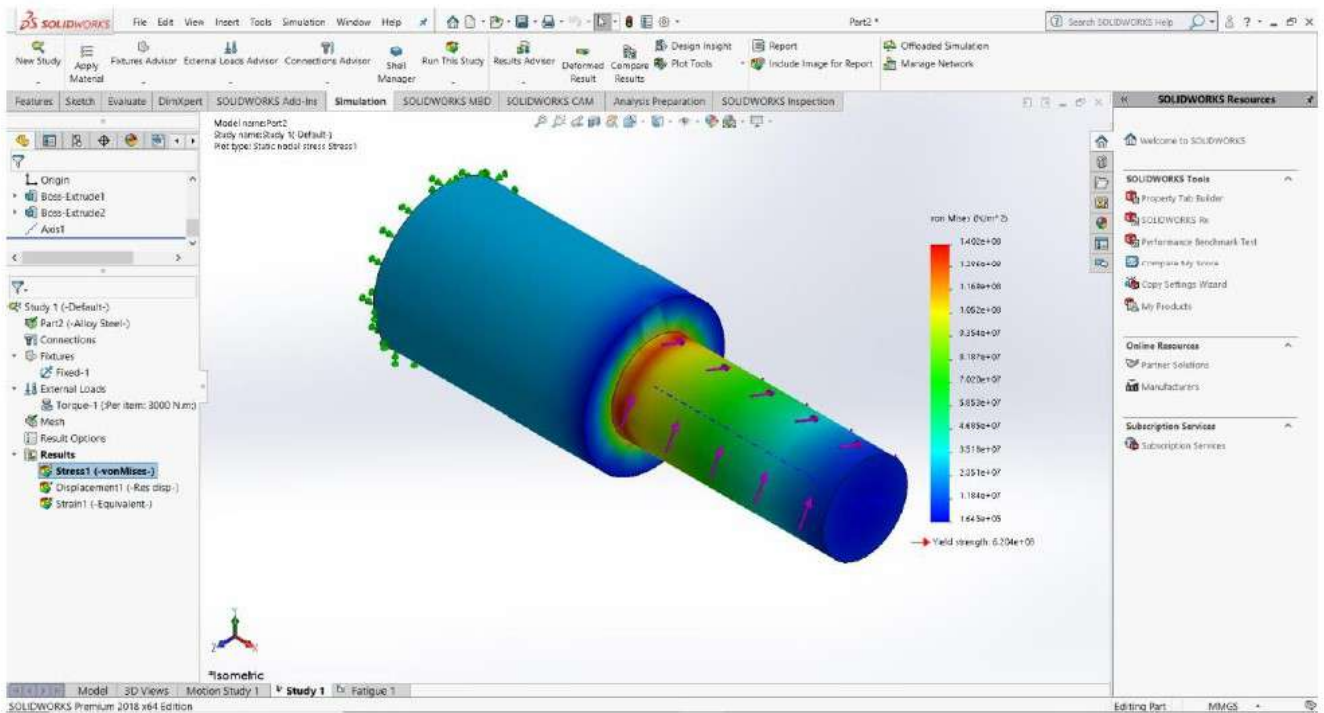


Figure (9)

- ⇒ From simulation toolbar select New study
- ⇒ Select Fatigue, figure (10)

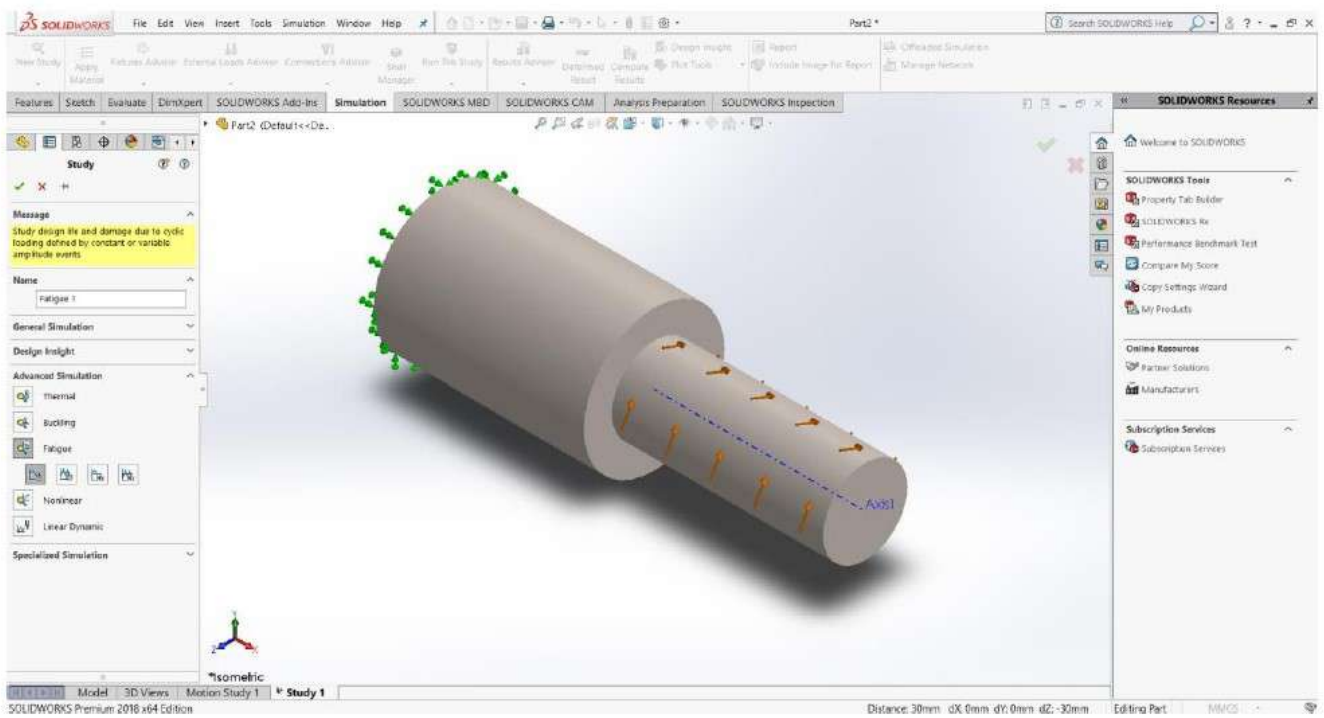


Figure (10)

- ⇒ Right click on loading and select Add Event, figure (11)

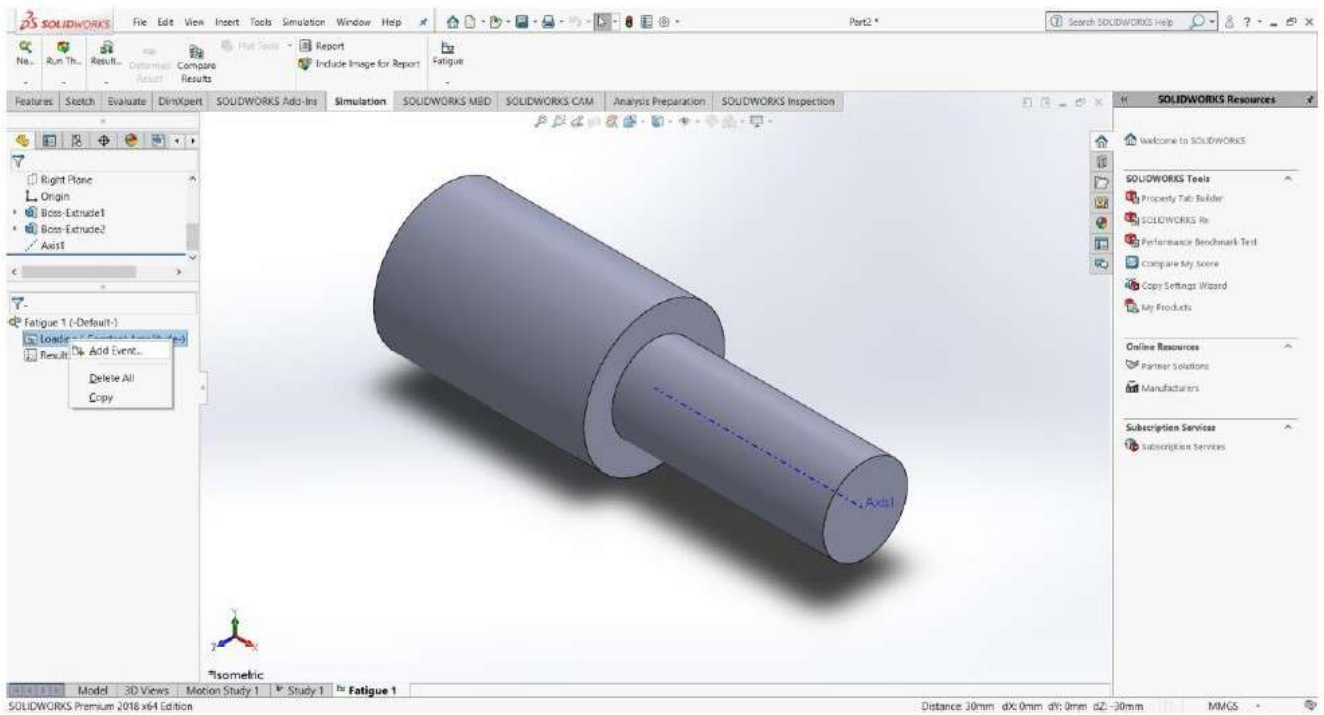


Figure (11)

- ⇒ Set the cycles as 10000000 cycle
- ⇒ Select fully reversed
- ⇒ The previous study is set by default
- ⇒ Click Ok, figure (12)

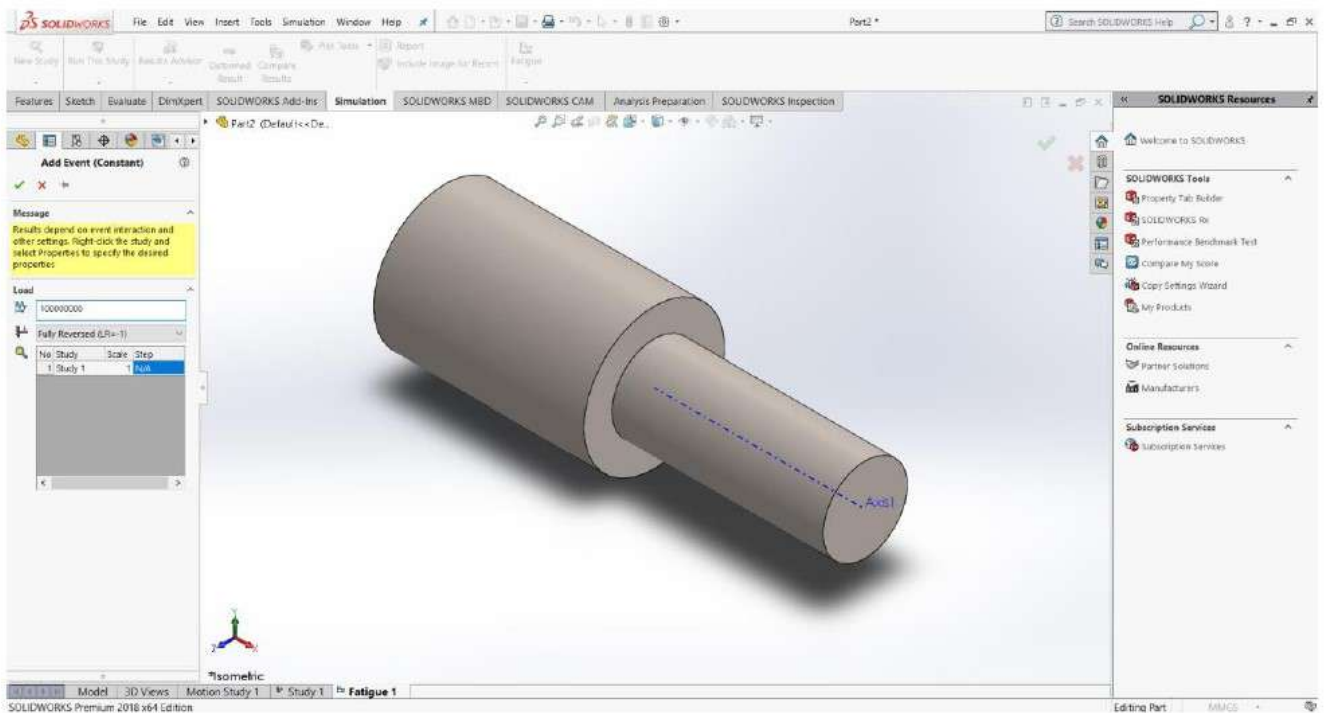


Figure (12)

- ⇒ Right click on the part name and select Apply/Edit Fatigue data, figure (13)

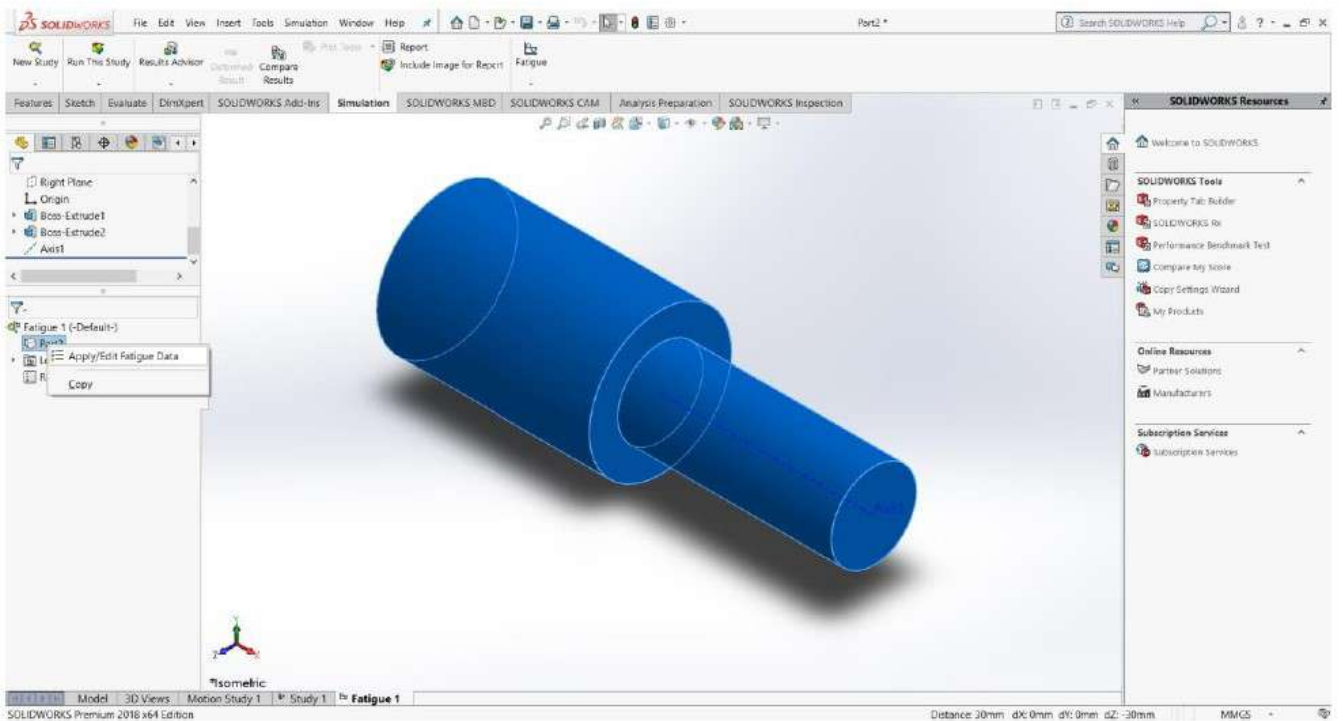


Figure (13)

- ⇒ Click on Derive from material Elastic Modulus
- ⇒ Click on Based on ASME Carbon Steel curves
- ⇒ Click Apply, figure (14)

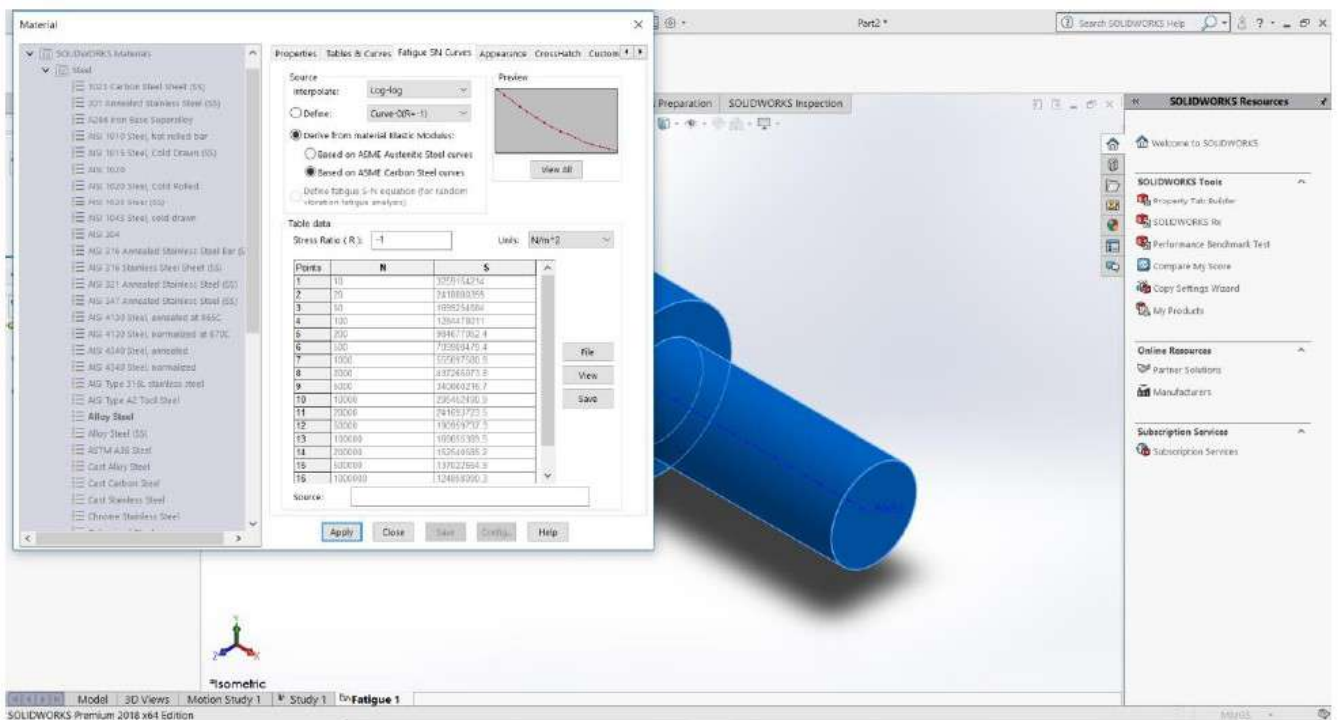


Figure (14)

Run the study

The damage contour is as figure (15)

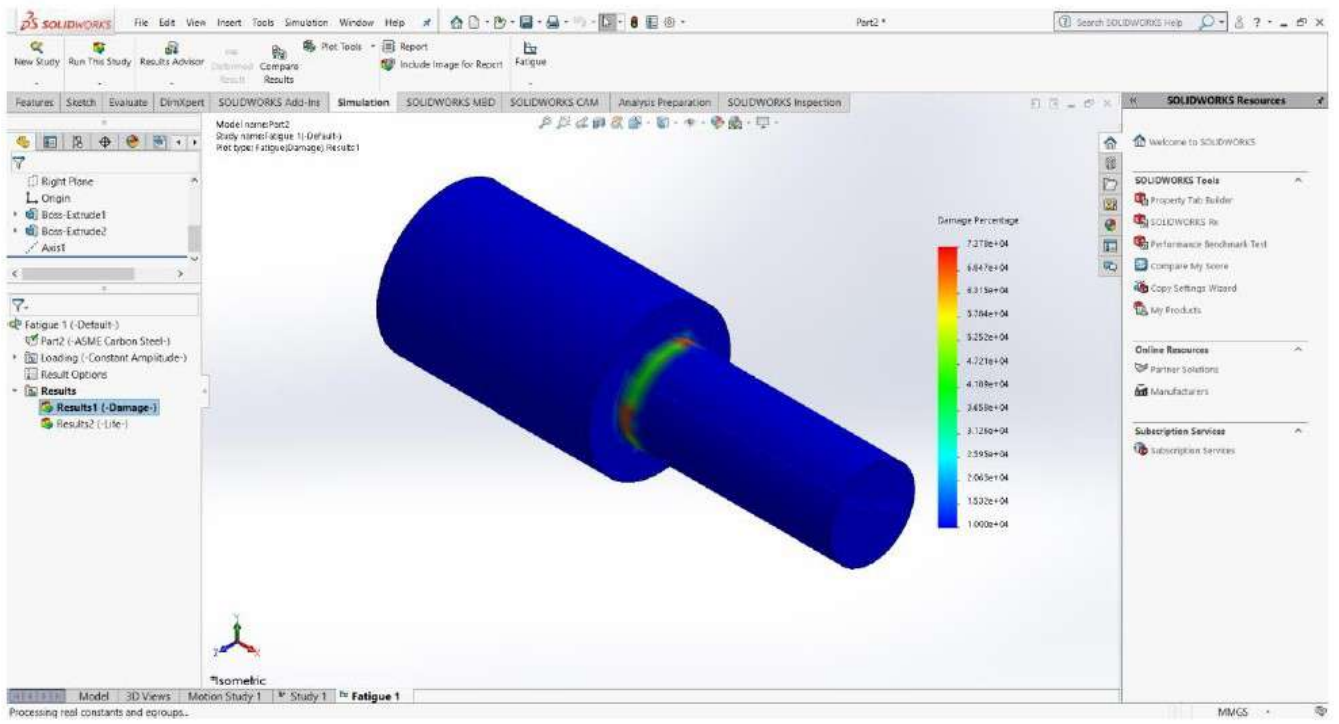


Figure (15)