

VM 250 Computational Lab Sessions

Lab #5

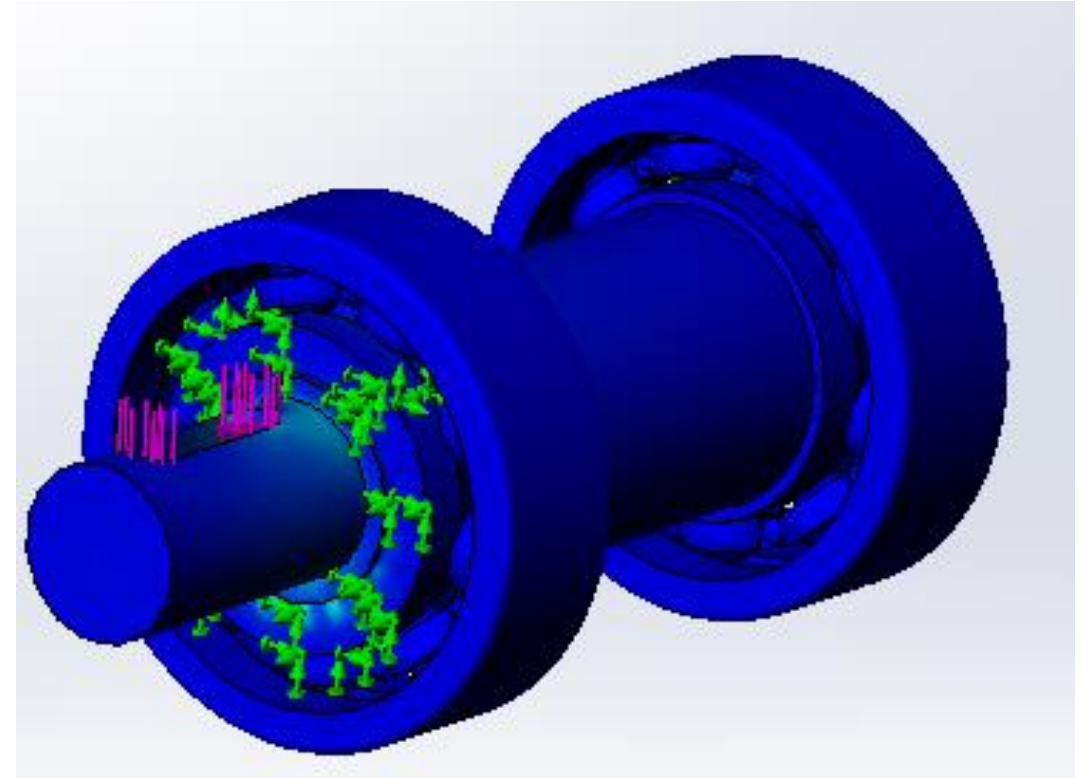
Finite Element Analysis with SolidWorks

Prepared by TA Group



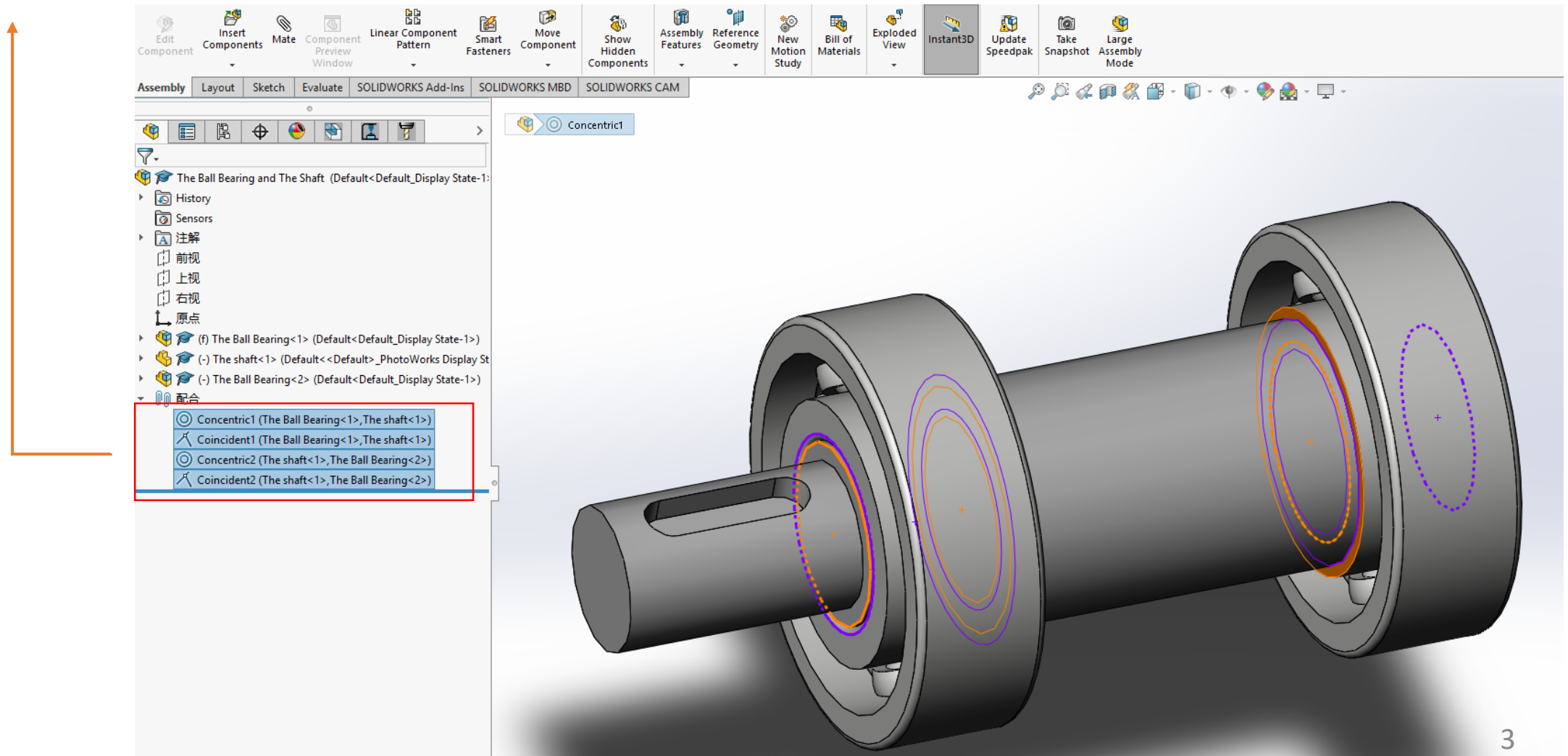
Outline

- Assign materials
- Fix the inner ring of the bearing
- Apply loads
- Local contact/ global contact
- Mesh
- Output of results



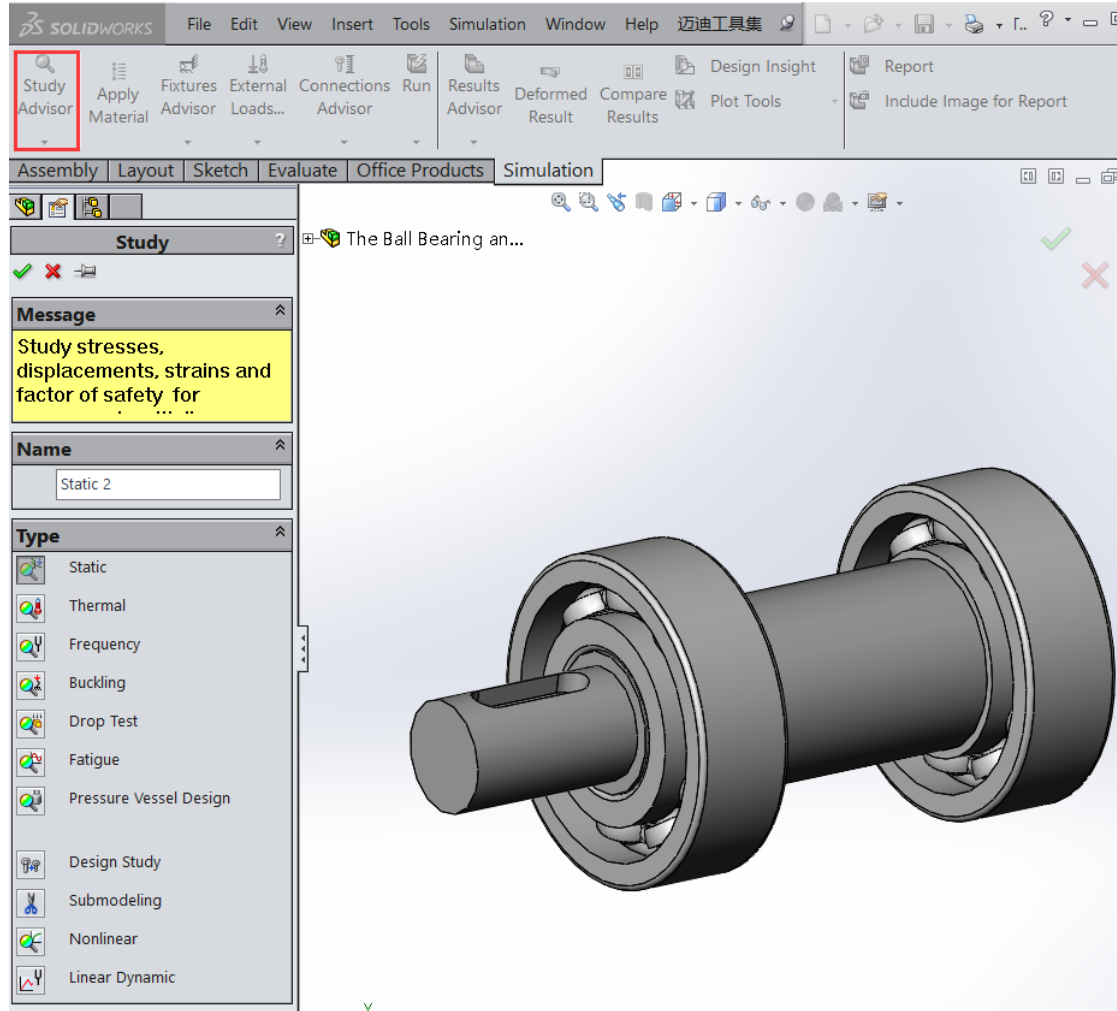
Create an Assembly

- Import all the previous parts including 1 shaft and 2 bearings, mating them.

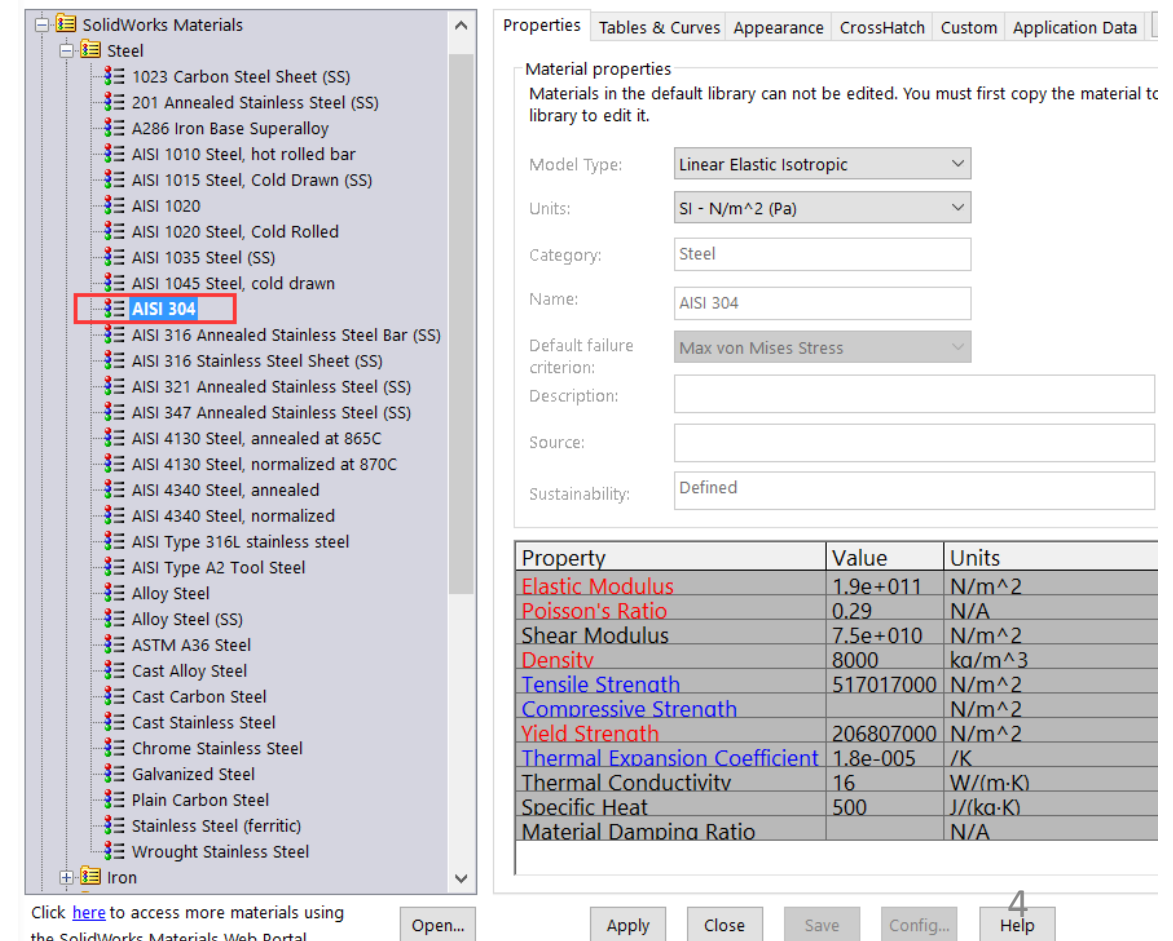
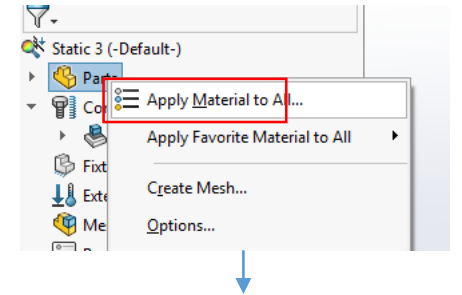


Assign materials to all parts

- Build a new static study in the simulation module



- Assign material to all →

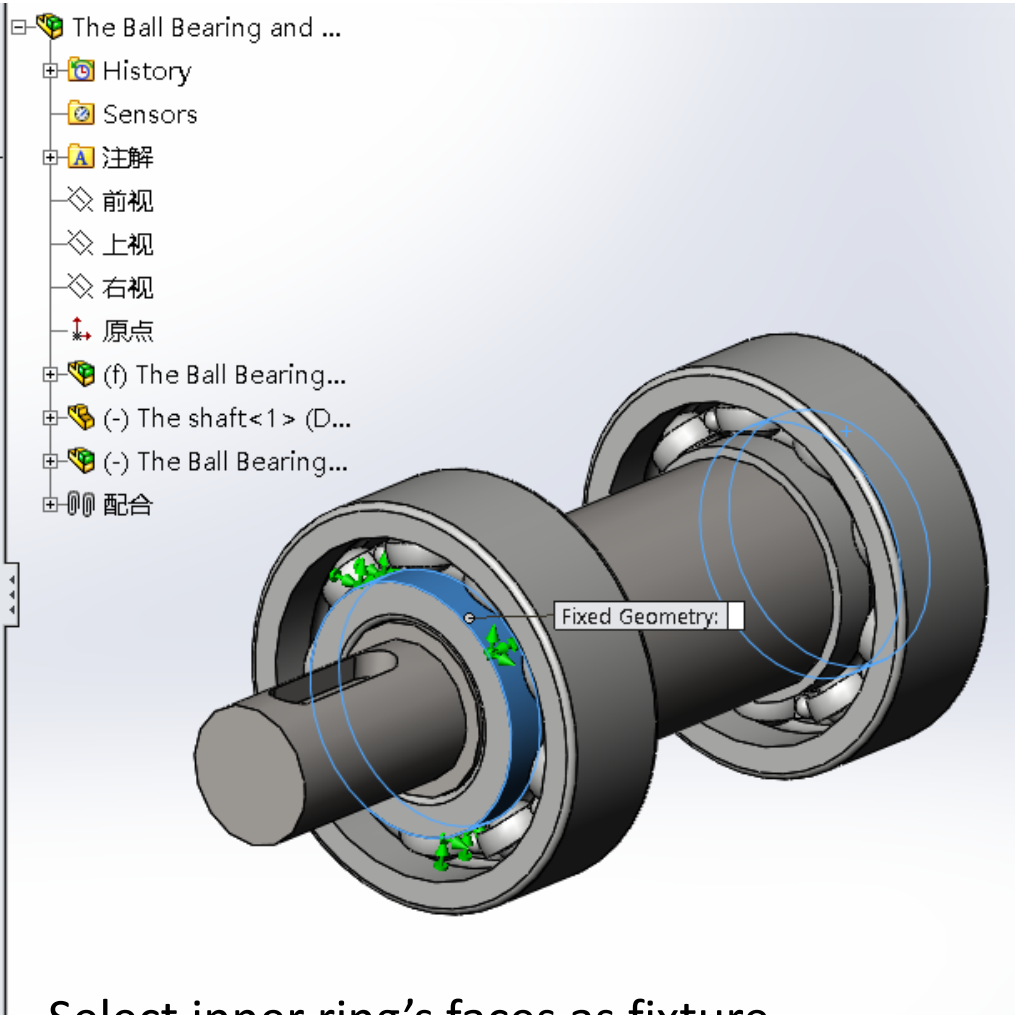
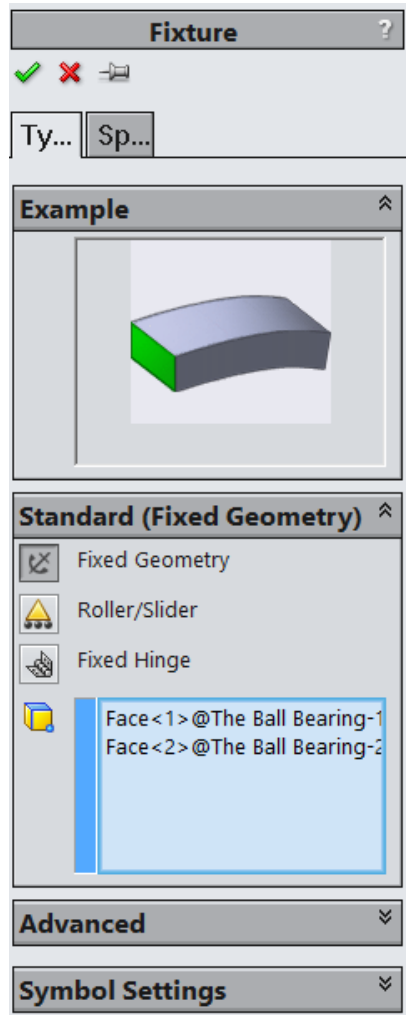
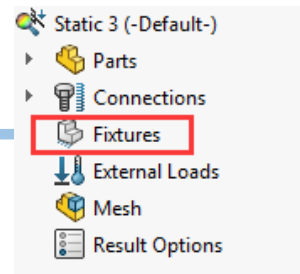


The image shows the SolidWorks Materials library and the Material properties dialog. The Materials library lists various materials, with 'AISI 304' highlighted. The Material properties dialog shows the selected material's properties, including Model Type, Units, Category, Name, Default failure criterion, Description, Source, and Sustainability. A table of material properties is also displayed.

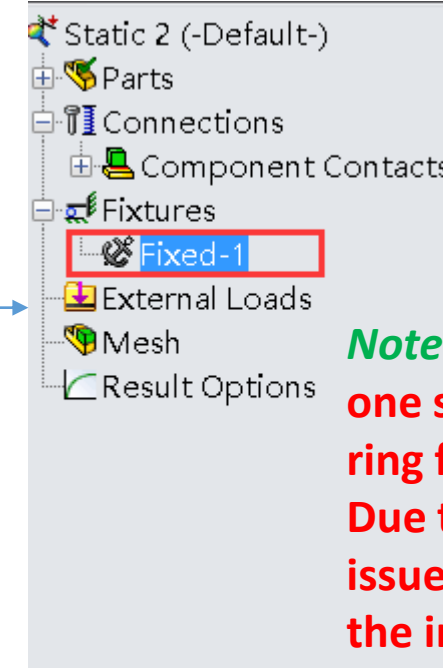
Property	Value	Units
Elastic Modulus	1.9e+011	N/m^2
Poisson's Ratio	0.29	N/A
Shear Modulus	7.5e+010	N/m^2
Density	8000	kg/m^3
Tensile Strength	517017000	N/m^2
Compressive Strength		N/m^2
Yield Strength	206807000	N/m^2
Thermal Expansion Coefficient	1.8e-005	/K
Thermal Conductivity	16	W/(m·K)
Specific Heat	500	J/(kg·K)
Material Damping Ratio		N/A

Fixture

- Fix the inner ring of the bearing



Select inner ring's faces as fixture

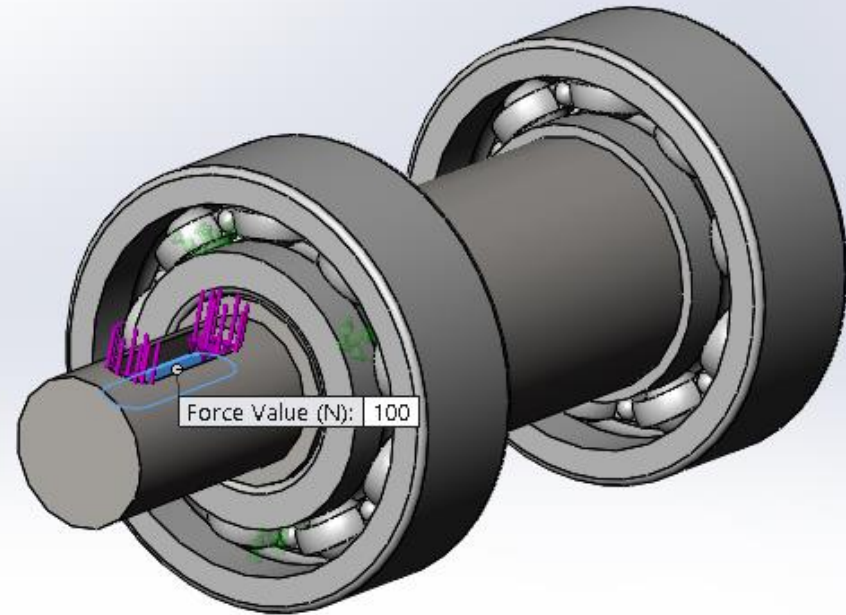
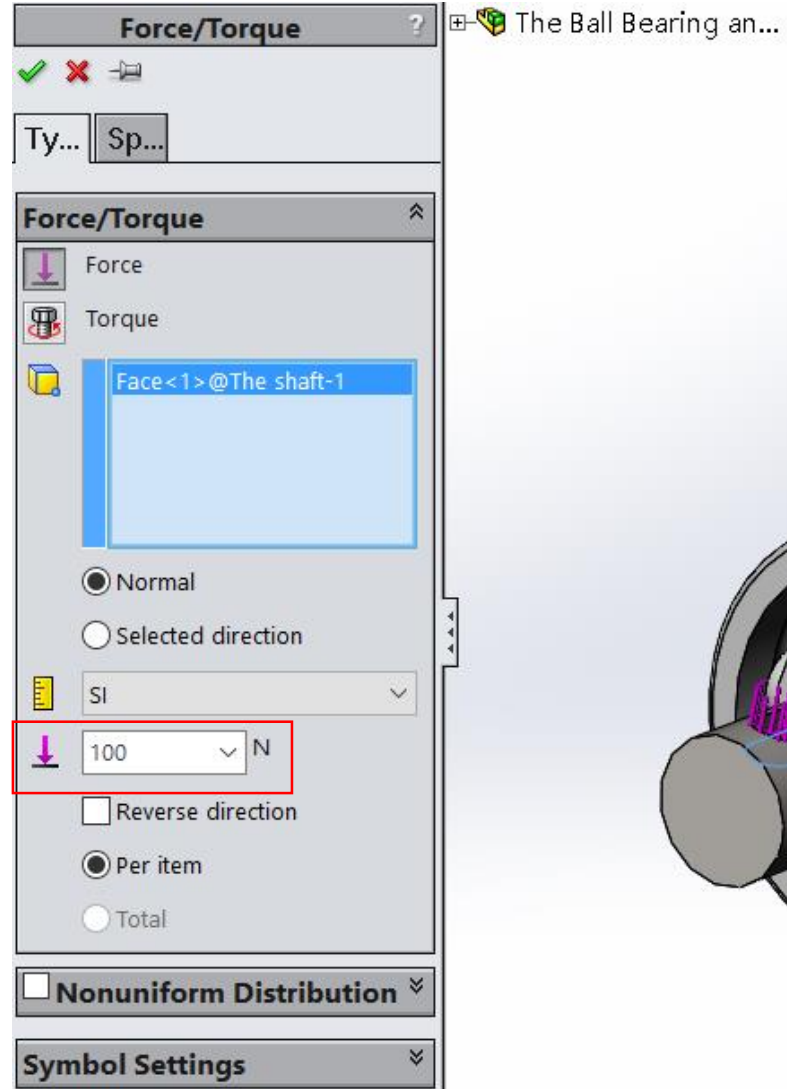
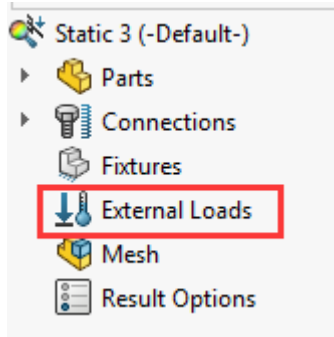


Note: For real applications, one should fix the outer ring for the analysis. Due to a computational issue on SolidWorks, we fix the inner ring in this activity.

- Hide the green arrow by right clicking the fixed geometry.

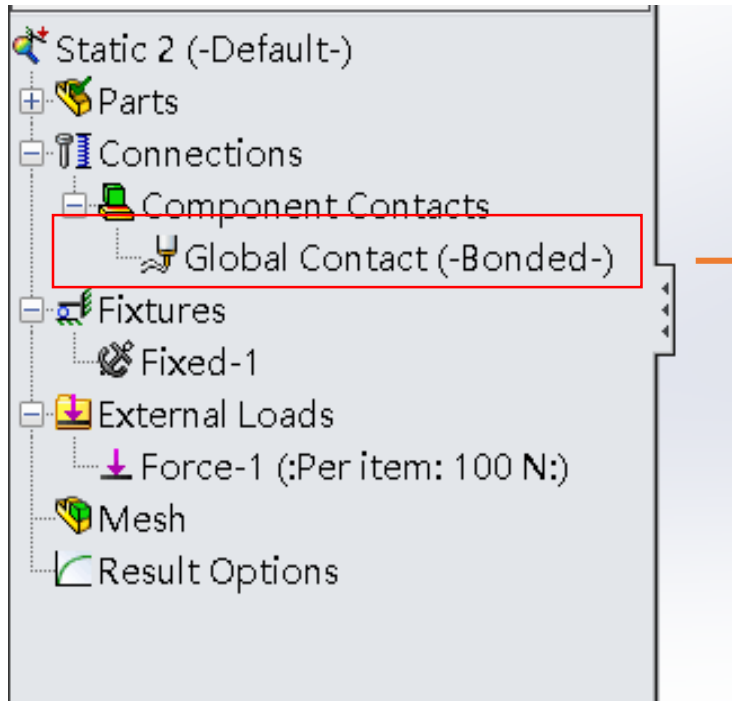
Apply loads

- Apply loads on the face of the key.



Local contact/ Global contact

- Assign a contact option on the surface sets.



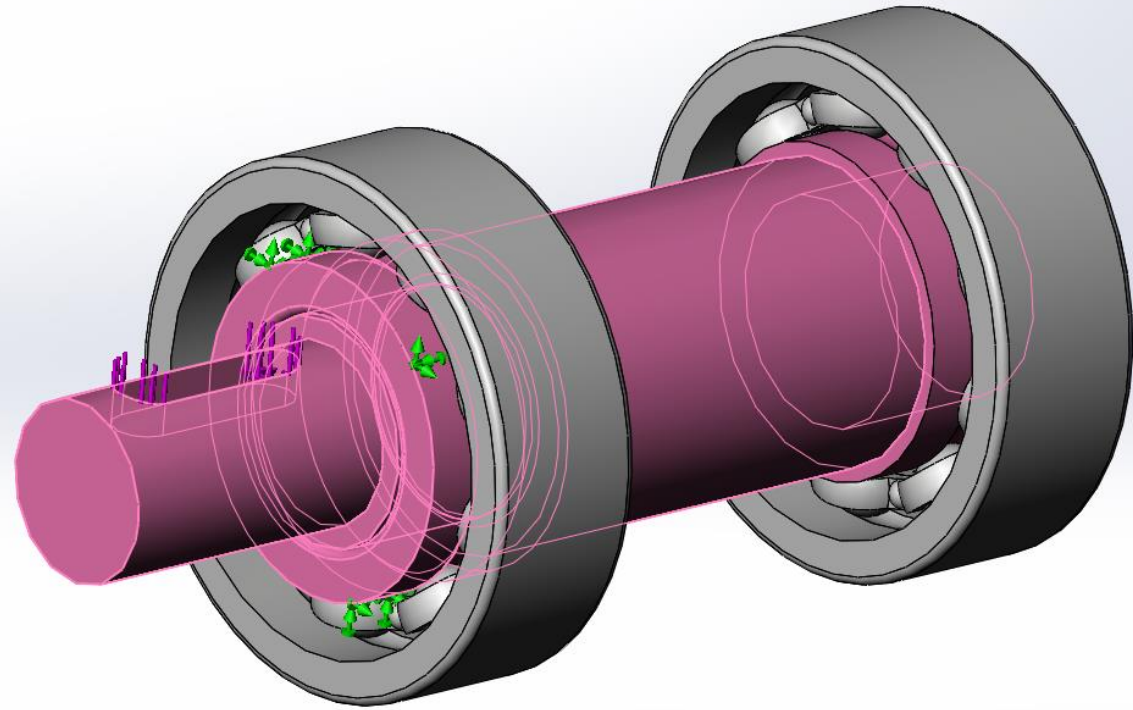
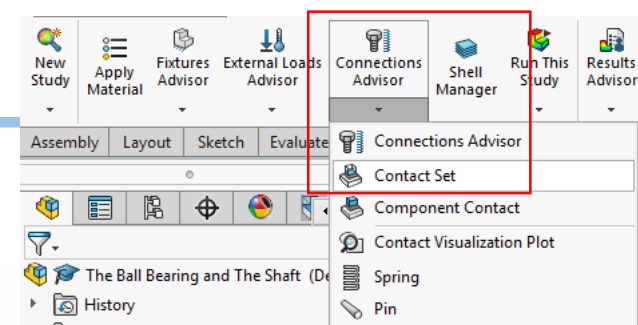
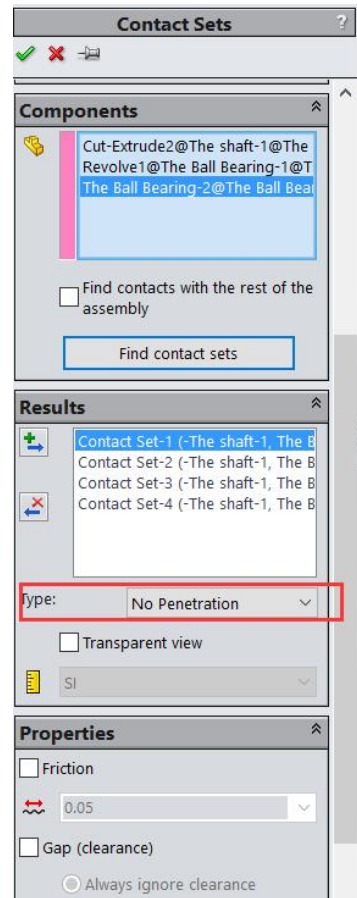
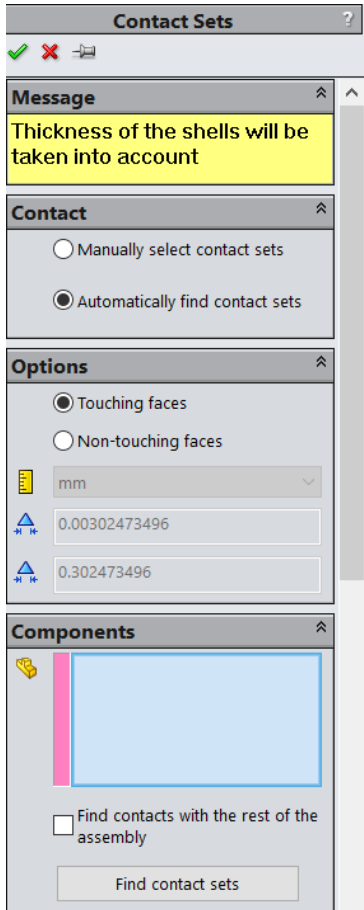
SolidWorks creates a **global contact condition**. The default option is a bond (tie), which means parts that are touching any other parts will be treated as bonded unless you choose any **local contact sets**.

Note: Local contact sets:

Allow the contacting surface **to be separable** if the loads become extremely high.

Local contact/ Global contact

- Manually/ Automatically choose the contact sets.

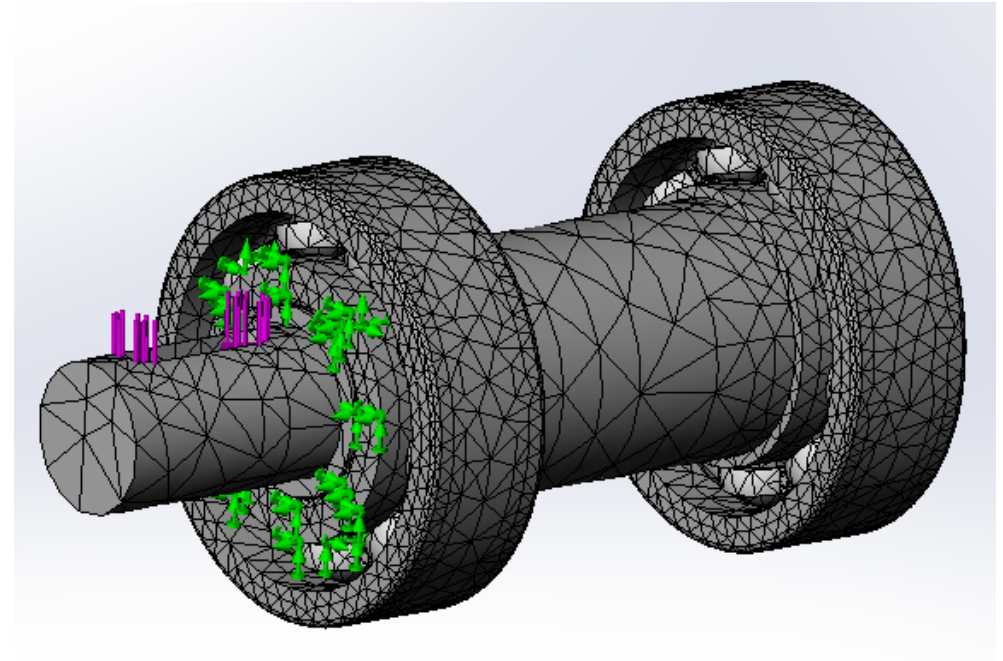
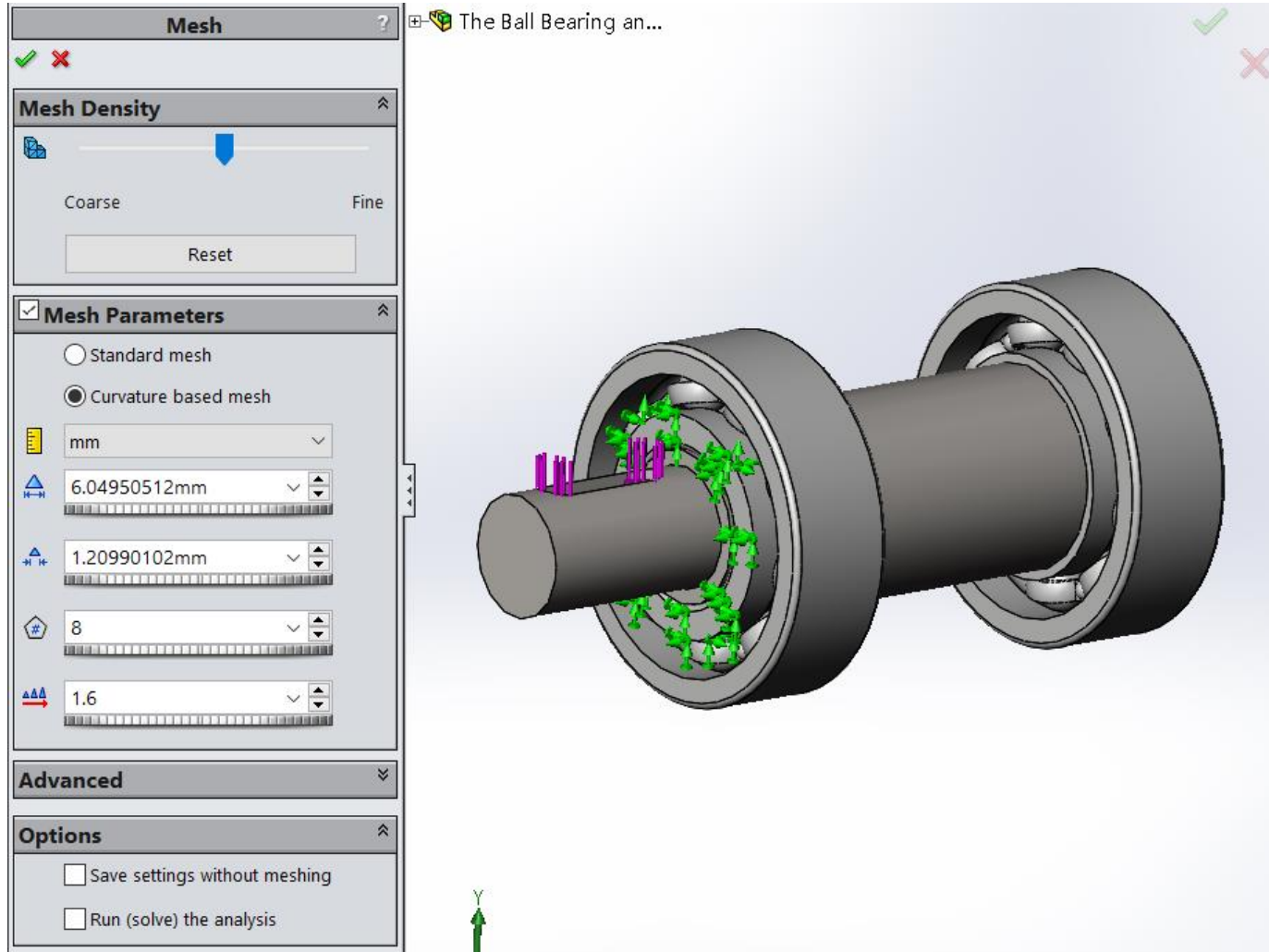
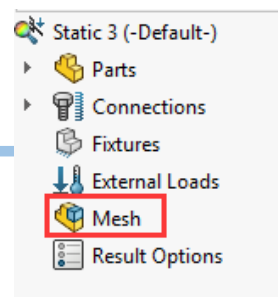


Automatically

- No penetration** means SolidWorks will monitor the contacting surfaces which may be separated from each other depending on the level of loads.

Mesh

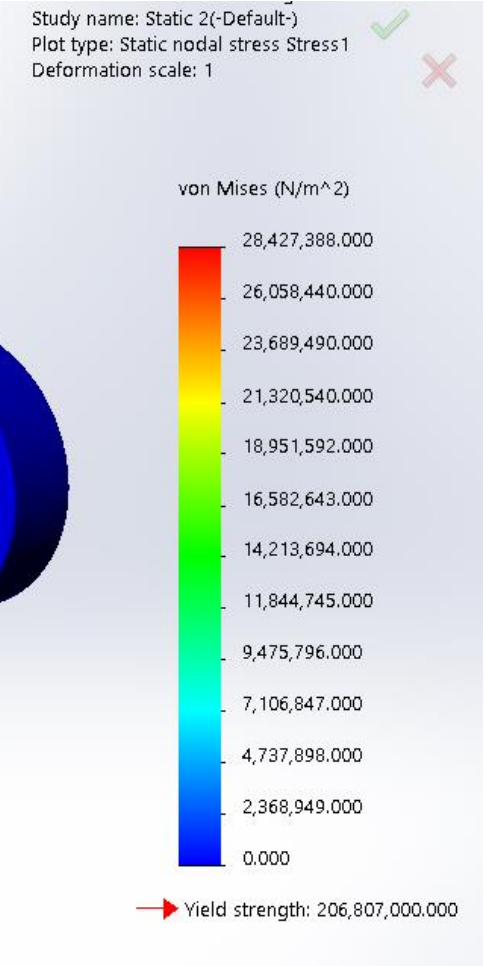
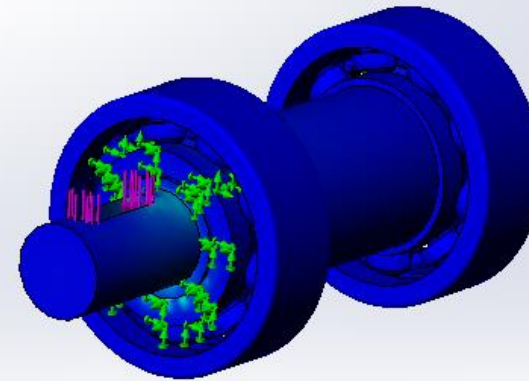
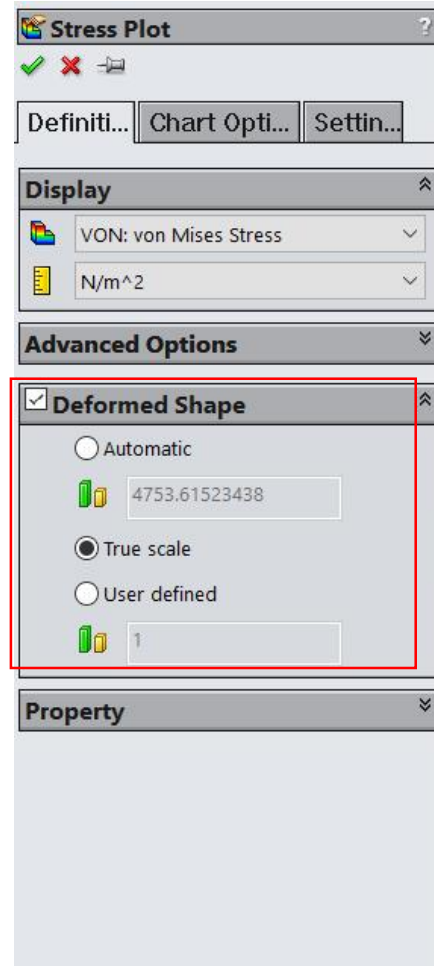
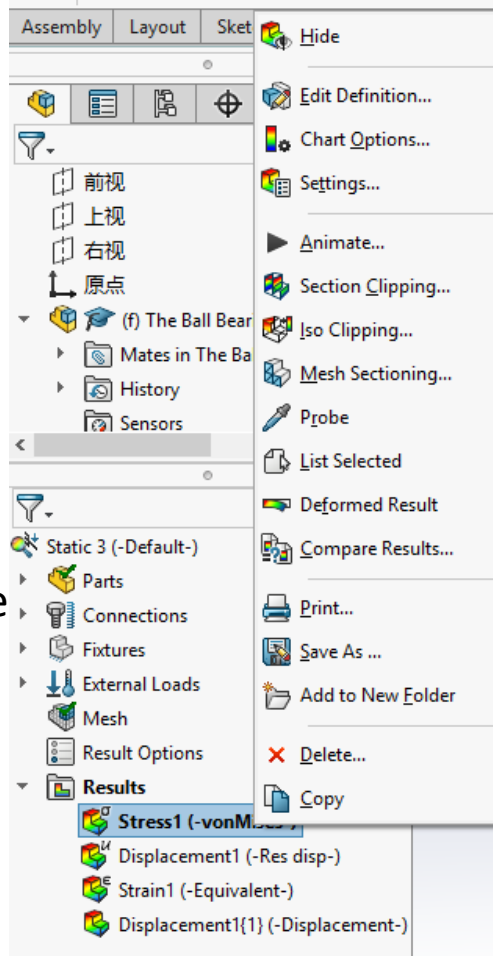
- Curvature based mesh



Run

- Run this study →
- Exaggerate the deformation or use true scale setting

1. Right-click on the stress and edit definition.
2. Choose the scale value for exaggerating the results.

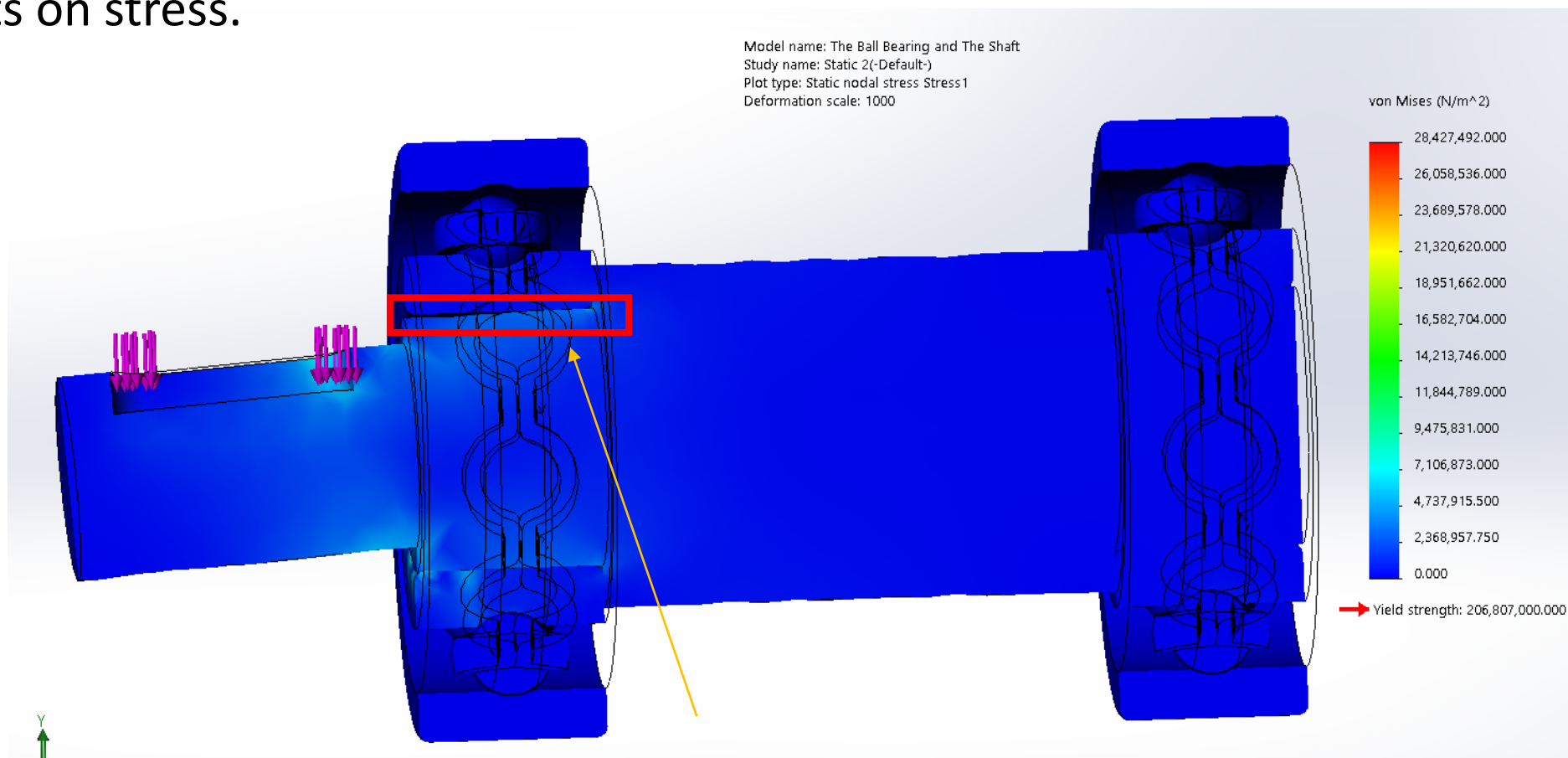
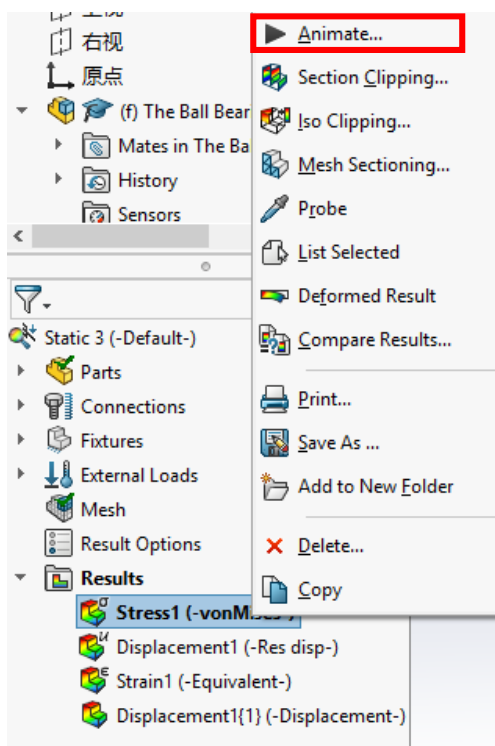


- The run time of the setting with no penetration contact will be longer than the one without it

Results on stress

The procedures to generate results

- 1. Create a cross section view.
- 2. Show results on stress.
- 3. Animation

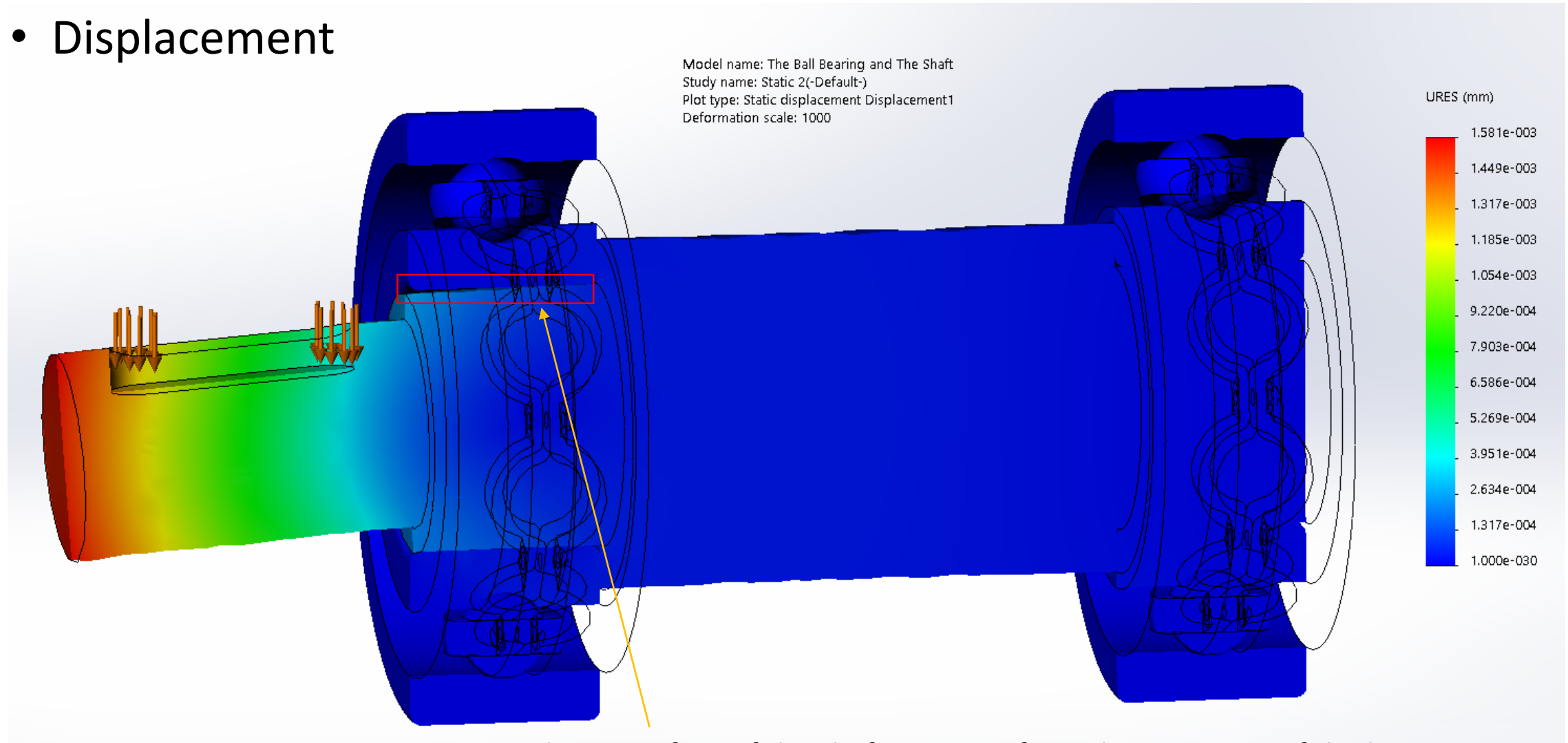


Visualize with an animation

By setting **no penetration contact**, the outer face of the shaft separates from the inner ring of the bearing (shown on the rectangle in red)

Results on displacement

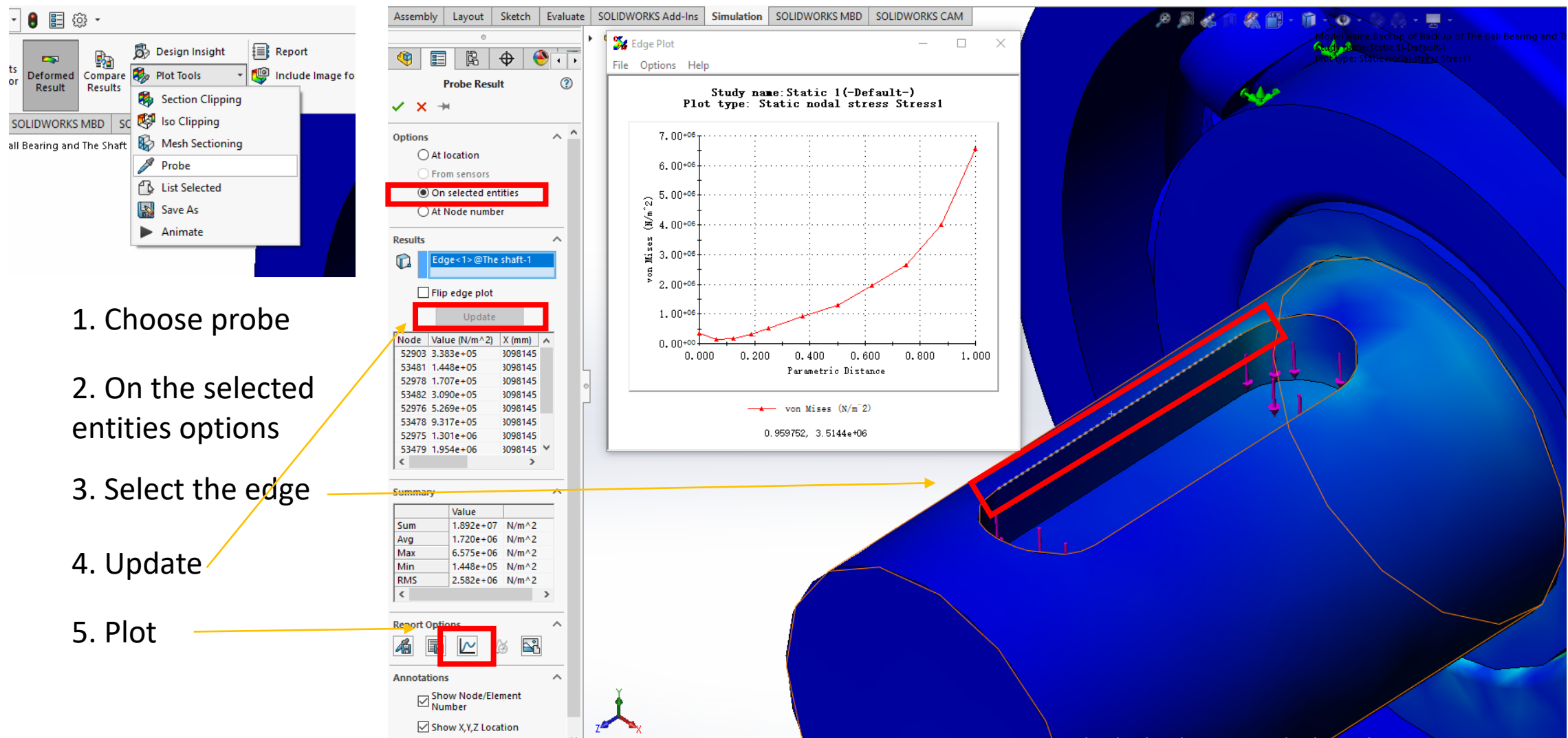
- Displacement



By setting no penetration contact, the outer face of the shaft separate from the inner ring of the bearing as shown on the rectangle in red.

Plot of stress on an edge

- Stress



Lab assignment

3D Sketch

- Draw this part with SolidWorks.

