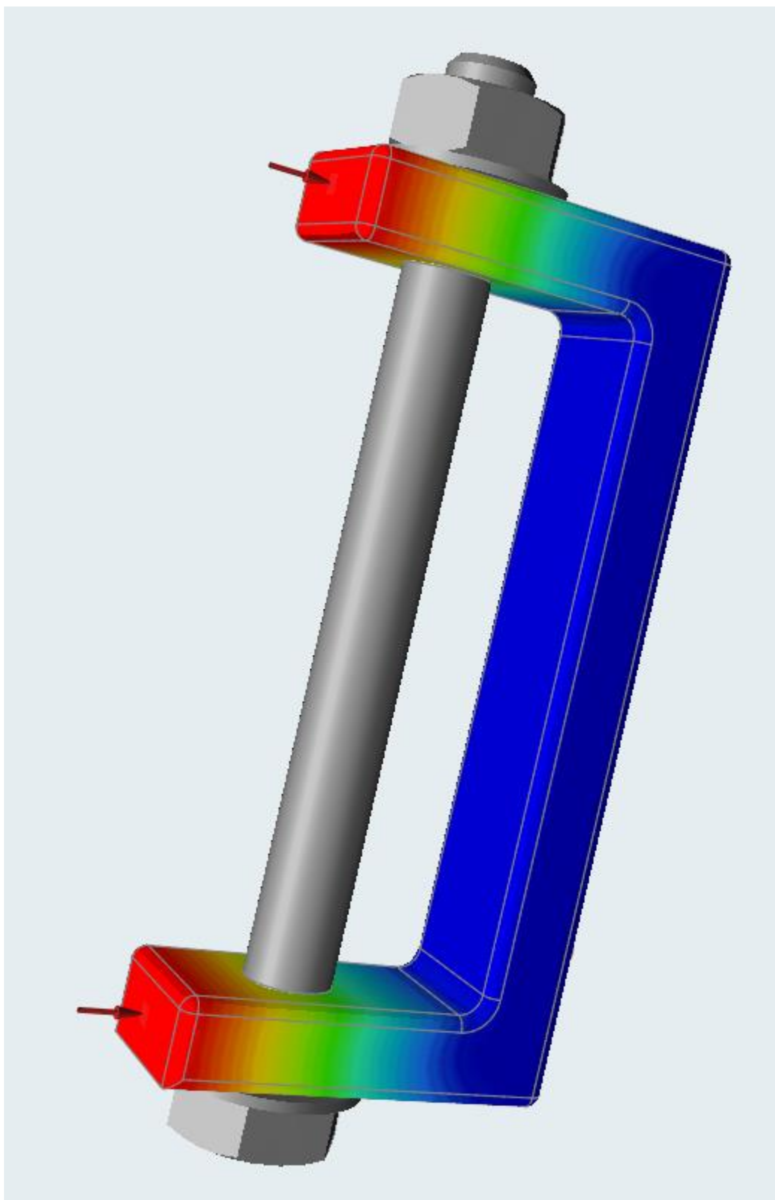
Tutorial: Analysis of a Bolt with Pretension

(Made with Altair Inspire)

(Updated by Nimisha Srivastava)



Tutorial: Analysis of a Bolt with Pretension

1. Opening the model:

- First, ensure that the MKS (m kg N s) unit system is activated by entering the **File > Preferences > Units** to ‘m Kg N s’.

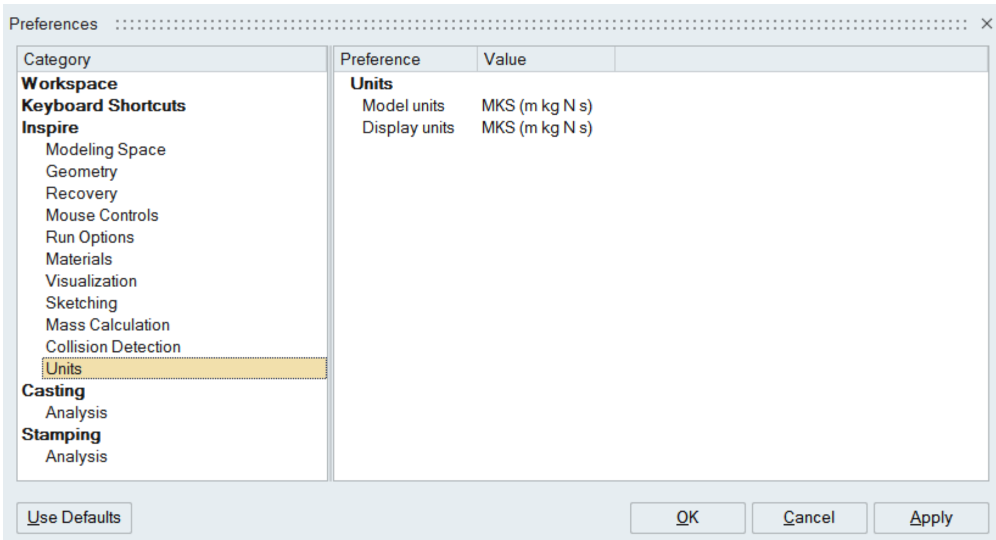


Figure 1 Setting the unit System

- Alternatively, use the dialogue at the bottom right hand corner

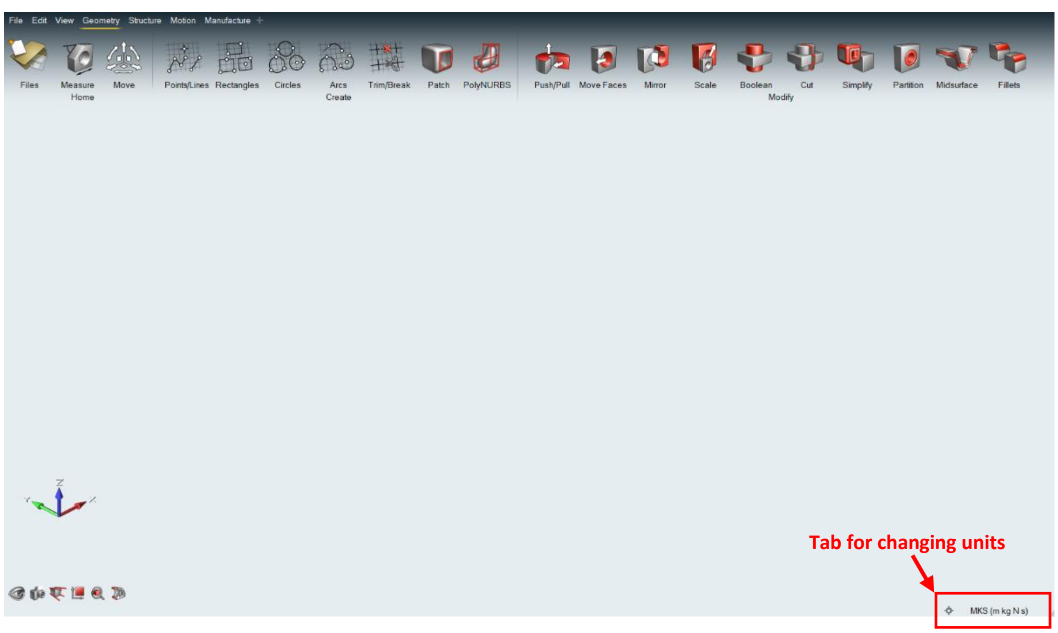


Figure 2 Tab for Changing Units

- Download the **.stmod** file provided with the tutorial and then open topography_11_18.stmod it using the **File> Open** option.

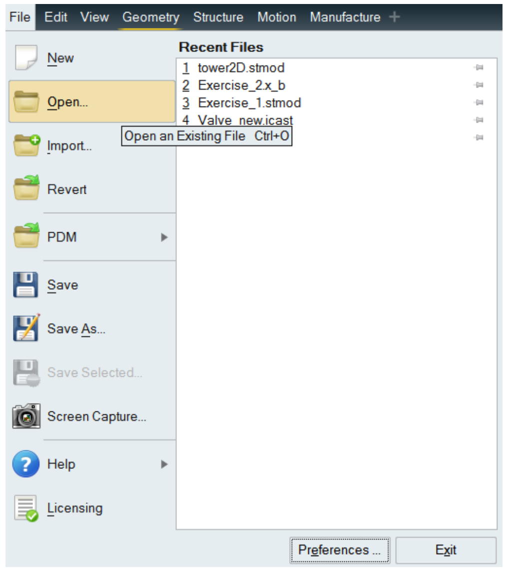


Figure 3 Opening the .stmod file

- Once the file is opened, the interface should look something like this:

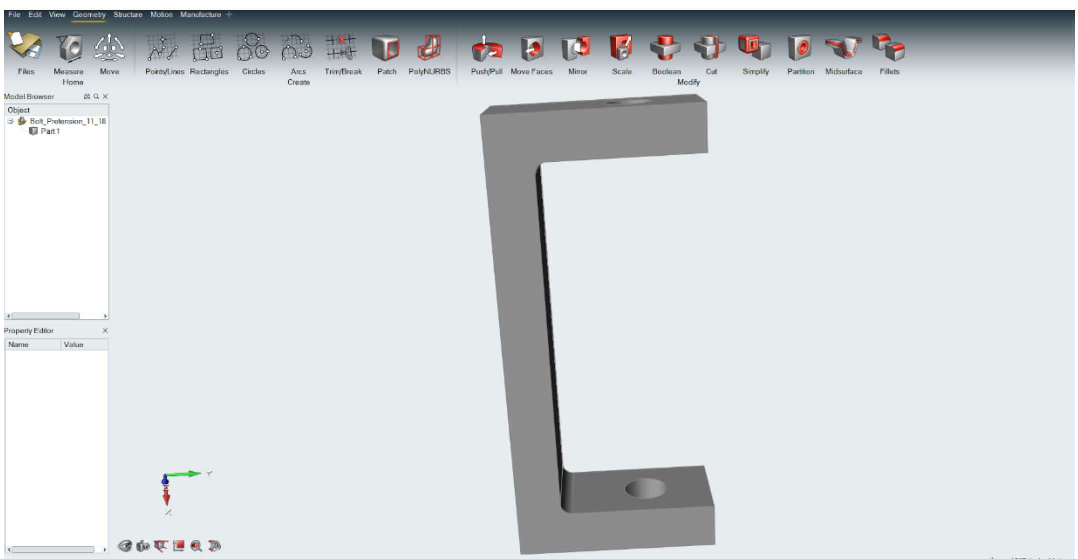


Figure 4 Interface after opening the file

2. Assigning the materials:

- This model need to be assigned as Steel (AISI 304) for this problem. Right click on the parts and select the **Materials** option. Select Steel (AISI 304) as the material.

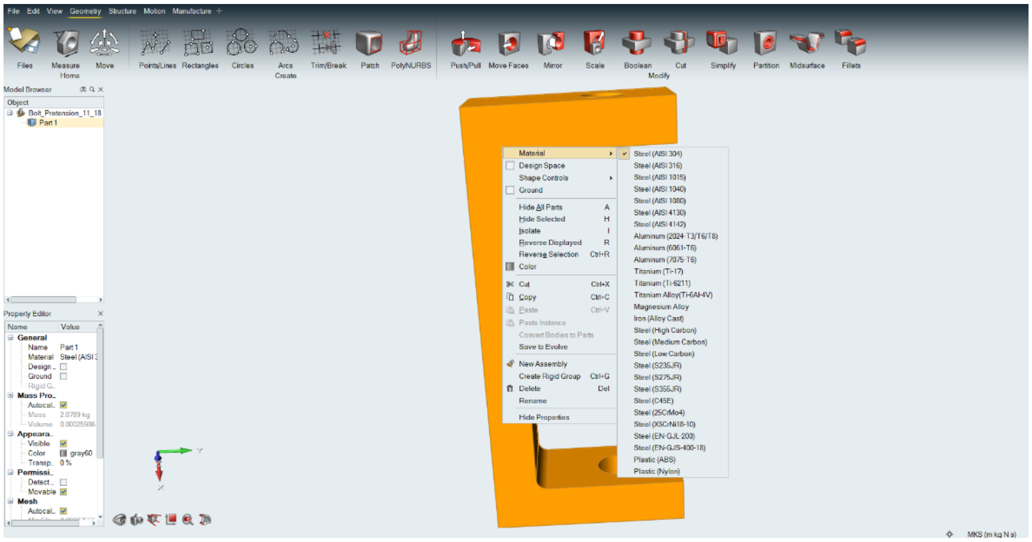


Figure 5 Assigning the material

- The same can be achieved by right-clicking on the Bolt_Pretension_11_12 icon in the model browser and assigning the material.
- The material assigned can be checked in **the Structure > Materials > Parts** tab.

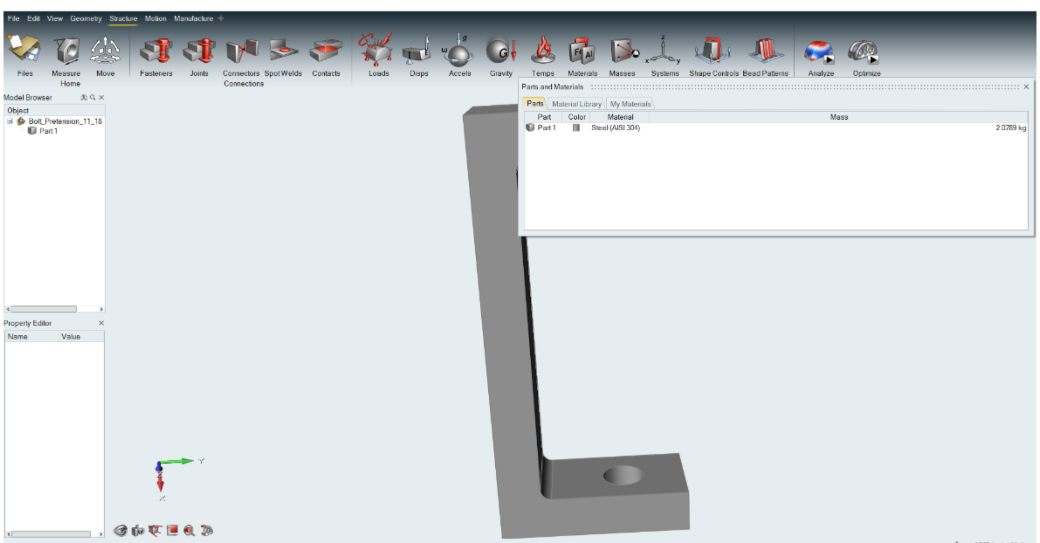
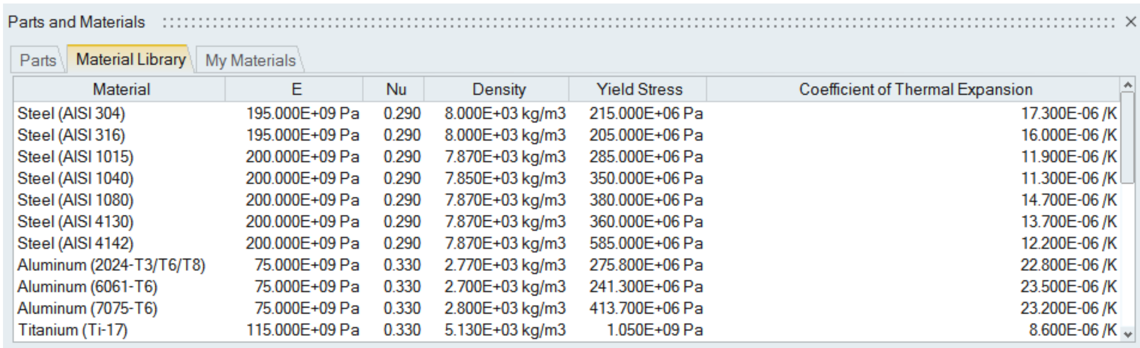


Figure 6 Checking the material

- The material properties can be reviewed by clicking on the **Structure> Materials > Material Library** tab.



Material	E	Nu	Density	Yield Stress	Coefficient of Thermal Expansion
Steel (AISI 304)	195.000E+09 Pa	0.290	8.000E+03 kg/m3	215.000E+06 Pa	17.300E-06 /K
Steel (AISI 316)	195.000E+09 Pa	0.290	8.000E+03 kg/m3	205.000E+06 Pa	16.000E-06 /K
Steel (AISI 1015)	200.000E+09 Pa	0.290	7.870E+03 kg/m3	285.000E+06 Pa	11.900E-06 /K
Steel (AISI 1040)	200.000E+09 Pa	0.290	7.850E+03 kg/m3	350.000E+06 Pa	11.300E-06 /K
Steel (AISI 1080)	200.000E+09 Pa	0.290	7.870E+03 kg/m3	380.000E+06 Pa	14.700E-06 /K
Steel (AISI 4130)	200.000E+09 Pa	0.290	7.870E+03 kg/m3	360.000E+06 Pa	13.700E-06 /K
Steel (AISI 4142)	200.000E+09 Pa	0.290	7.870E+03 kg/m3	585.000E+06 Pa	12.200E-06 /K
Aluminum (2024-T3/T6/T8)	75.000E+09 Pa	0.330	2.770E+03 kg/m3	275.800E+06 Pa	22.800E-06 /K
Aluminum (6061-T6)	75.000E+09 Pa	0.330	2.700E+03 kg/m3	241.300E+06 Pa	23.500E-06 /K
Aluminum (7075-T6)	75.000E+09 Pa	0.330	2.800E+03 kg/m3	413.700E+06 Pa	23.200E-06 /K
Titanium (Ti-17)	115.000E+09 Pa	0.330	5.130E+03 kg/m3	1.050E+09 Pa	8.600E-06 /K

Figure 7 Reviewing Material Properties

3. Defining the Problem:

- First, we modify the model to be analyzed by creating fillets on all edges except the circular edges at the edges.
- Go to the **Geometry > Modify > Fillets**. This is a multi-sensitive icon which enables the user to create chamfers or fillets.

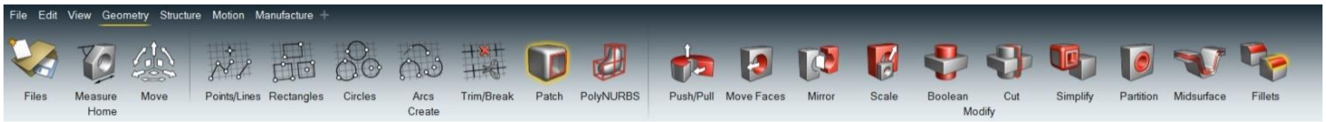


Figure 8 Fillet Icon on the Geometry ribbon

- Click and Drag to select all the edges of the model. Since the fillets are not required around the edges near the holes so press control and deselect the 4 edges.
- Press **Fillet all** to get the modified model.

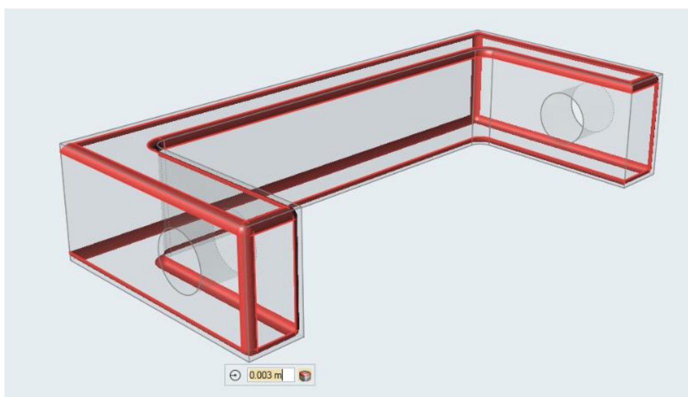


Figure 9 Edges to be filleted

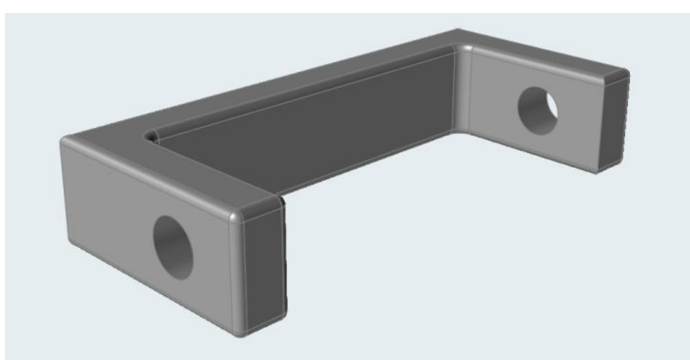


Figure 10 Modified model

- Next, we will create the fasteners at the holes.
- Go to **Structure > Fasteners**. Click on the two cylindrical holes and click on fasten all.

Figure 11 Fasteners Icon

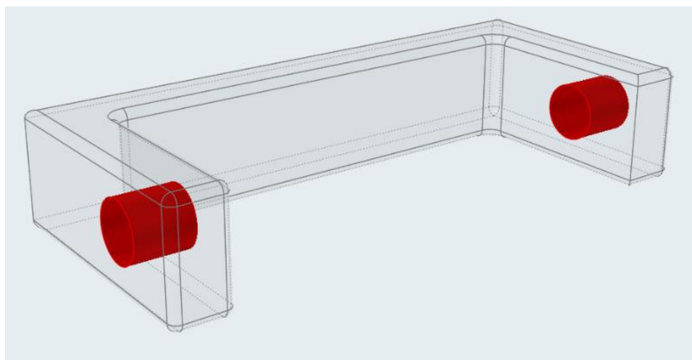


Figure 12 Cylindrical Holes to be Selected

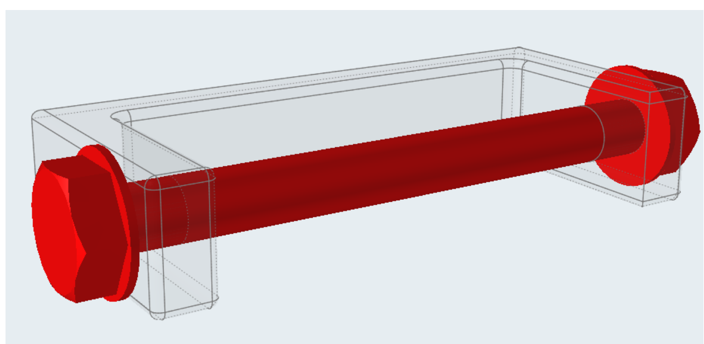


Figure 13 Fastened Holes

- Next, enable the pretension in the bolt by clicking on the fastener in the model browser and checking the box next to **Enable** under **Pretension** in the **Property Editor**.

  Enter 5000 N as the force.

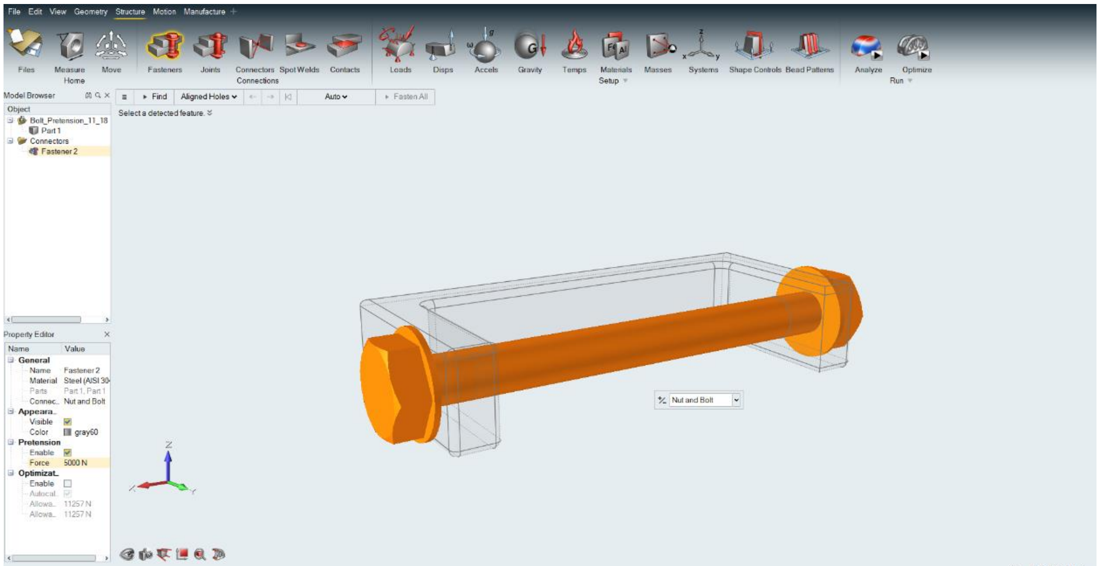


Figure 14 Enabling Pretension

- Go to Structure > Loads and click on the red cones at the bottom of the multi-sensitive icon to create supports.

Figure 15 Icon For creating Supports

Select the face shown in the following Figure to create the support to simulate a rigid attachment.

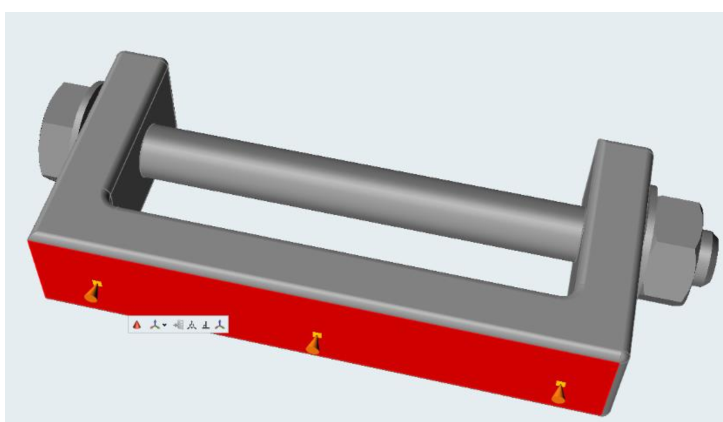


Figure 16 Supports Created

- The next step is to create loads. Go to **Structure > Loads** and click on the red arrow to create a directional load.

Figure 17 Icon for creating Loads

Click on the faces shown in the figure and enter the load magnitude as 1 N.

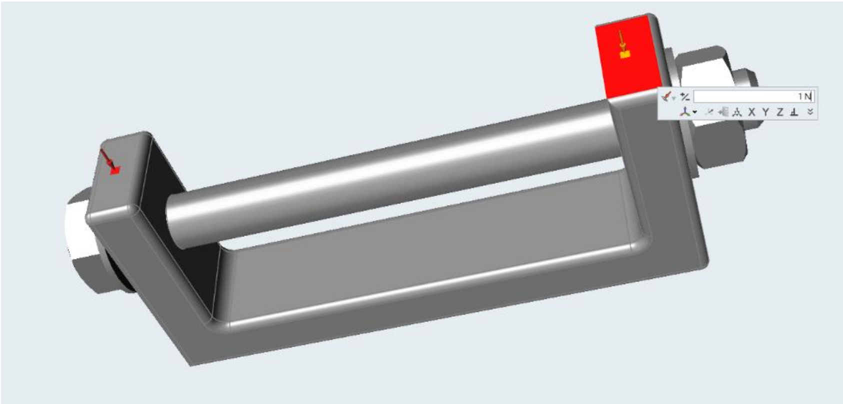


Figure 18 Imposed loads

4. Analysis

- Click on the play button on the analyze icon.

Figure 19 The Analyze Icon

- Let the element size be 5 mm.
- Select More Accurate for Speed/Accuracy.
- Select Sliding Only for the Contacts.
- Click run.

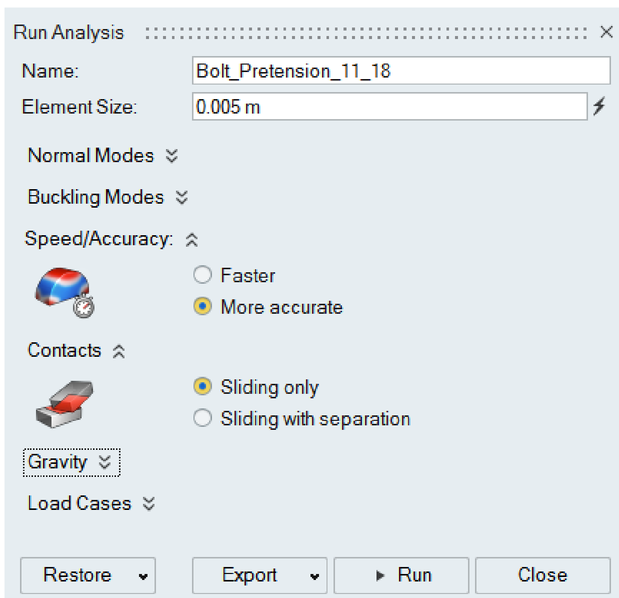


Figure 20 Settings for the Analysis

- Once the analysis is completed, a green flag appears next to the analysis icon. Click on it to access the results.

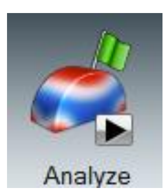


Figure 21 Click here for the analysis results

5. Results

- Click on **Displacement** in the **Result Types** menu to see the displacement distribution.
- The maximum displacement is 6.351×10^{-2} mm.

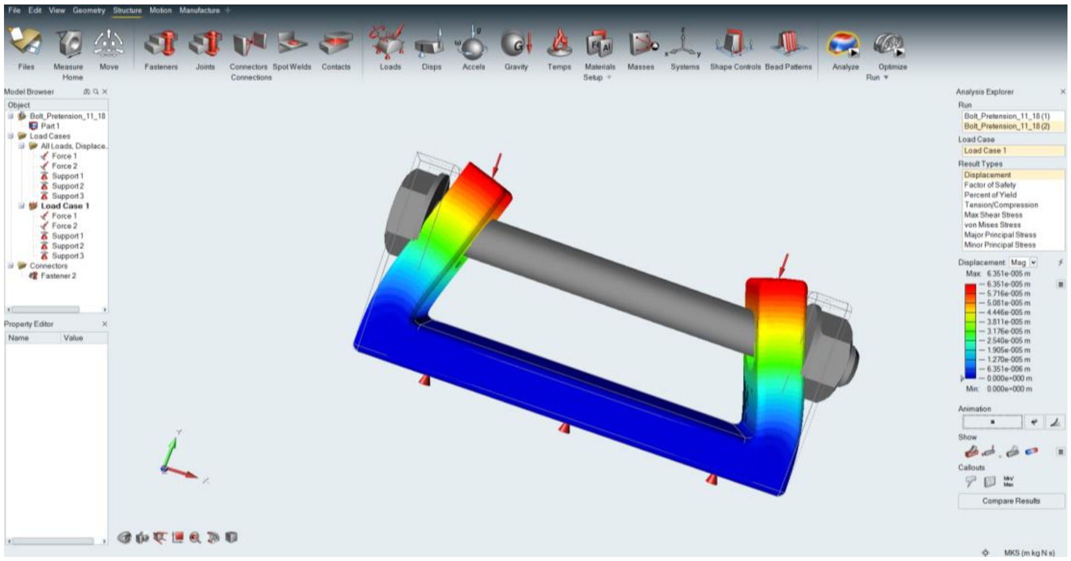


Figure 22 Screenshot for Displacement Animation

- Click on Von Mises Stress in the Result Types menu to see the Von Mises Stress distribution.
- The maximum Von Mises Stress is 1.023×10^{8} Pa

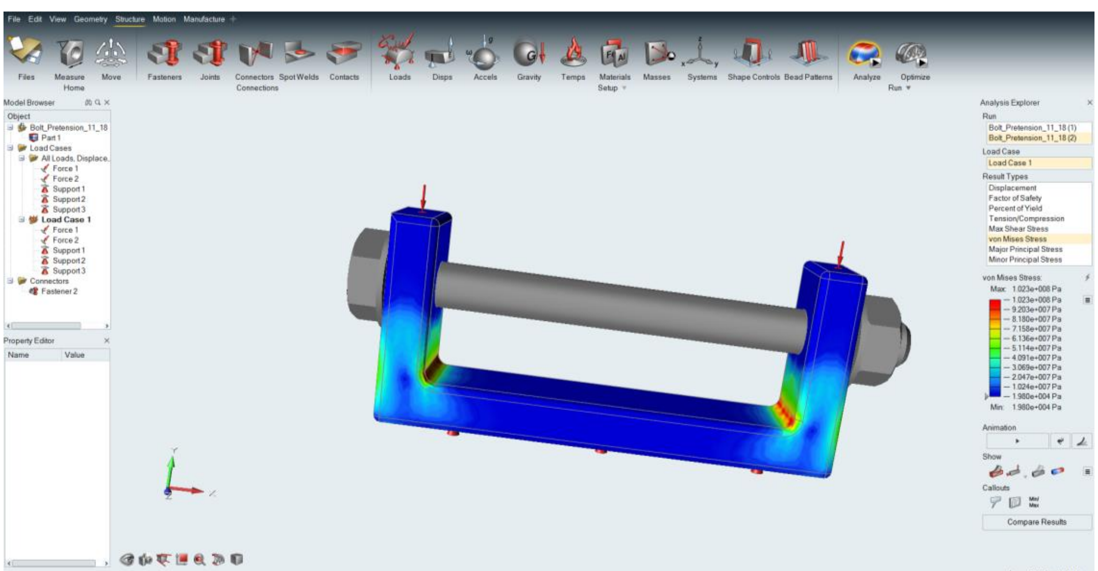


Figure 23 Von Mises Stress Distribution