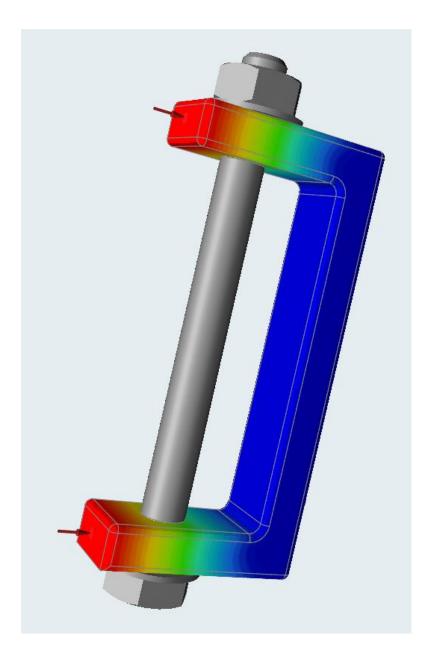


Tutorial: Analysis of a Bolt with Pretension

(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'.

Category	Preference	Value			
Workspace	Units				
Keyboard Shortcuts	Model units	MKS (m kg N s)			
Inspire	Display units	MKS (m kg N s)			
Modeling Space					
Geometry					
Recovery					
Mouse Controls					
Run Options					
Materials					
Visualization					
Sketching					
Mass Calculation					
Collision Detection					
Units					
Casting					
Analysis					
Stamping					
Analysis					
-					
<u>U</u> se Defaults			<u>О</u> К	<u>C</u> ancel	Apply

Figure 1 Setting the unit System

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

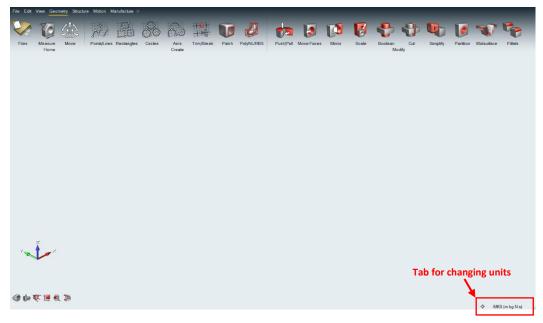


Figure 2 Tab for Changing Units

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

File	Edit View	Geometry	Structure	Motion	Manufacture	+
	New	I	Recent File	es		
	INEW		1 tower2D.			е а
	<u>O</u> pen		2 Exercise 3 Exercise	1.stmod		-14
2	Import		4 Valve ne Existing File			ы- ца
1	Revert					
7	PDM	Þ				
H	Save					
Ø	Save <u>A</u> s					
B	Save Selec	ted				
Ø	Screen Cap	ture				
?	Help	►				
5	Licensing					
					r <u>e</u> ferences	Exit

Figure 3 Opening the .stmod file

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

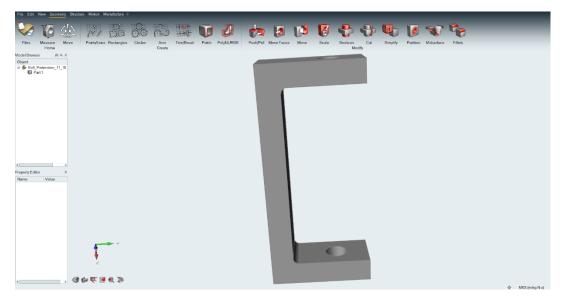


Figure 4 Interface after opening the file

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.

File Edit View Geometry Structure Motion Manufacture +			
😪 🐼 🎄 🕺 🖧 🗞	nd 🗱 🗊 🥔 🍖	📵 📢 🛃 🛟 🖏 🐚	🔞 🐨 🗞
Files Measure Move Points/Lines Rectangles Circles	Arcs Trim/Break Patch PolyNURBS Push/Pull	Move Faces Mirror Scale Boolean Cut Simplify	
Home Model Browser Ø. 9, ×	Create	Modify	
Modifhesen Opert ∋ Ø Ø Paet		Method 1 2 Metric/AD.320 Desgn Syste Steel (AD.320) Steel (AD.320) Gourd Steel (AD.320) Steel (AD.320) Gourd Steel (AD.320) Steel (AD.320) Hold AD Prin Steel (AD.330) Steel (AD.330) Hold AD Prin Steel (AD.330) Steel (AD.330) Bittle 1 Ammun (2014) Steel (AD.330) Bittle 1 Ammun (2014) Ammun (2014) Bittle 1 Ammun (2014) Ammun (2014) Bittle 1 Ammun (2014) Ammun (2014)	
Propeny Editor × Norme Value * General Name Part 1		Sc Cut Chi-X [Banim (F-B21)) (1) Copy Cut-C Tanim Alay(T-64-V) (2) Easte Chi-V Magnesum Alay Easte Chi-V Magnesum Alay Easte International State (State) ConvertiBiodies to Parts Save the Erole Vertice State (Medium Carbon)	
Material Ster (ASI: Derign - Gound - Reget C. Mass Pro. - Autrcal. 2070 kg			
Vehine 00005555 Vehine 0 0005555 Vehine 0 0005555 Vehine 0 000555 Vehine 0 0005 Vehine 0 005 Vehine 0 0005 Vehine 0 0005		Ster (9CANE 10) Ster (FX L3 20) Binal (FX L3 400 10) Plate (AS) Plate (Naro)	6 1955 (min 114)

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.

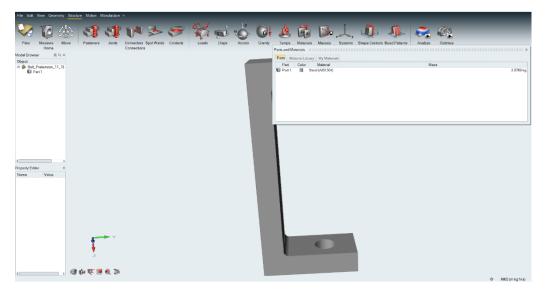


Figure 6 Checking the material

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

Parts and Materials					
Parts Material Library My	Materials				
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.

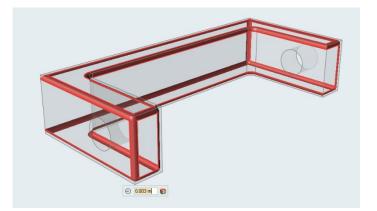


Figure 9 Edges to be filleted

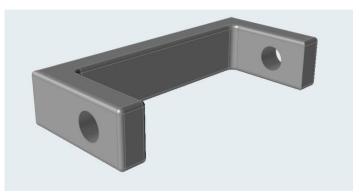


Figure 10 Modified model



- 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

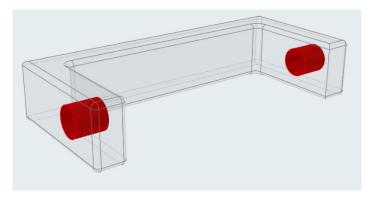


Figure 12 Cylindrical Holes to be Selected

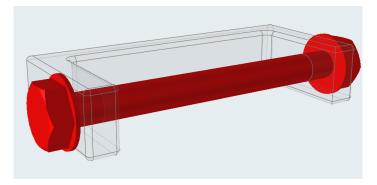


Figure 13 Fastened Holes

• 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.

🛆 Altair | University

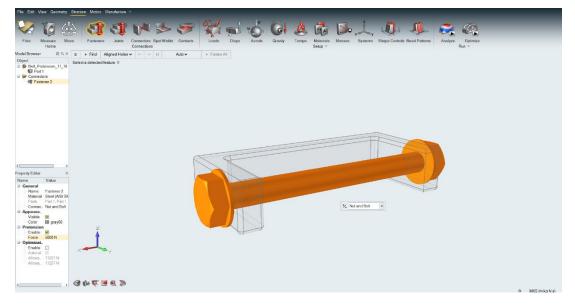


Figure 14 Enabling Pretension

• 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.

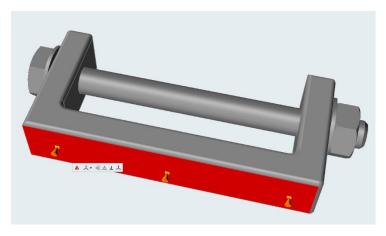


Figure 16 Supports Created



• 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.

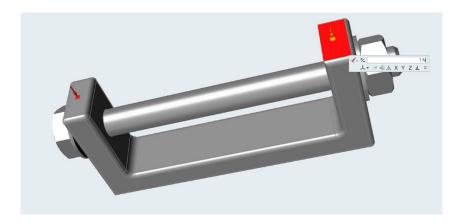


Figure 18 Imposed loads

- 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.

Run Analysis :::::	×				
Name:	Bolt_Pretension_11_18				
Element Size:	0.005 m				
Normal Modes ⇒					
Buckling Modes \gtrsim					
Speed/Accuracy: ;	*				
	○ Faster				
6	More accurate				
Contacts 🛠					
<i>_</i>	Sliding only				
1	 Sliding with separation 				
Gravity ≽					
Load Cases ⇒					
Restore •	Export • FRun Close				

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.

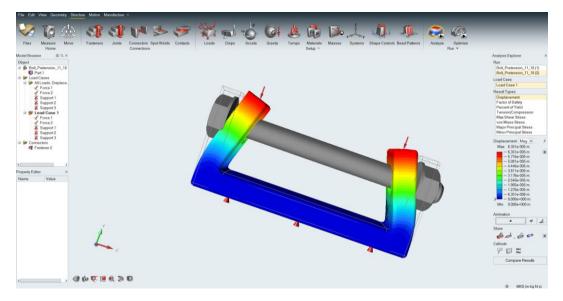


Figure 22 Screenshot for Displacement Animation

- 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 × 10⁸ Pa

