ME478 – Final Project

Emily Mun
Eric Lee

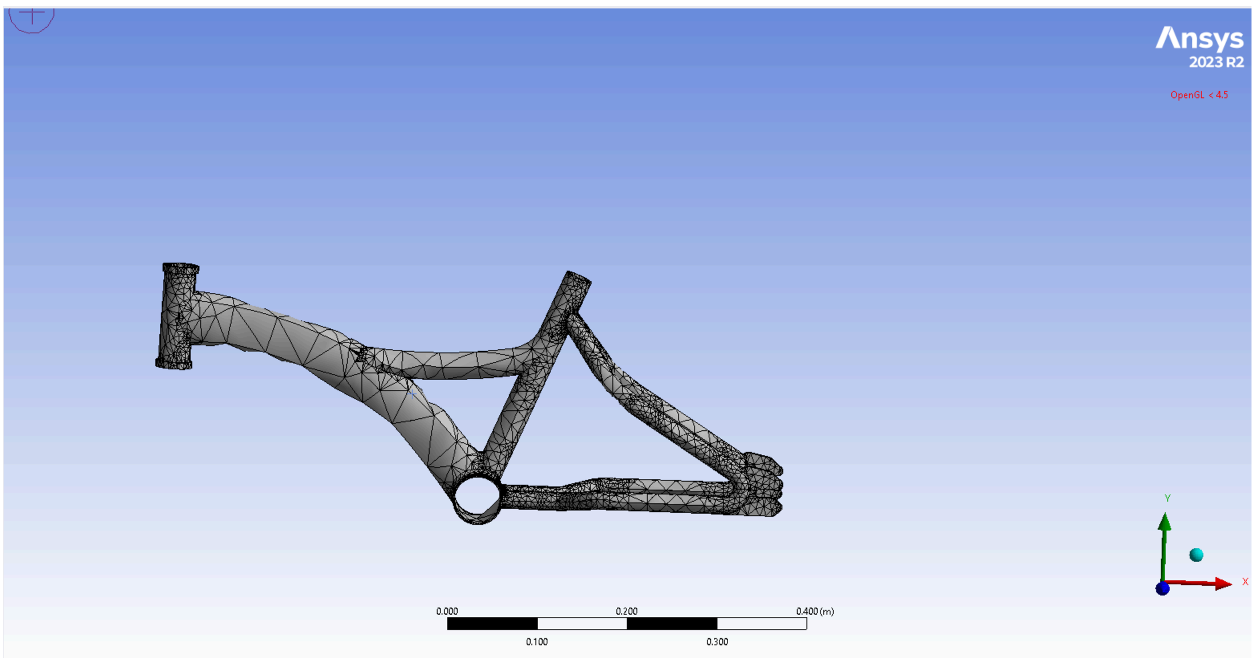
1. Investigate the stresses in an aluminum bike frame of your choice subjected to a load corresponding to a 60 kg rider (you can also download a bike frame file from a site like grabcad.com). Consider vertical loading at the top of the saddle tube. For simplicity, exclude the fork and the wheels in your analysis. What safety factor would you use? Do you trust this model? Explain. It may be good to consider stress levels, buckling, vibration resonances but you can exclude fatigue consideration.

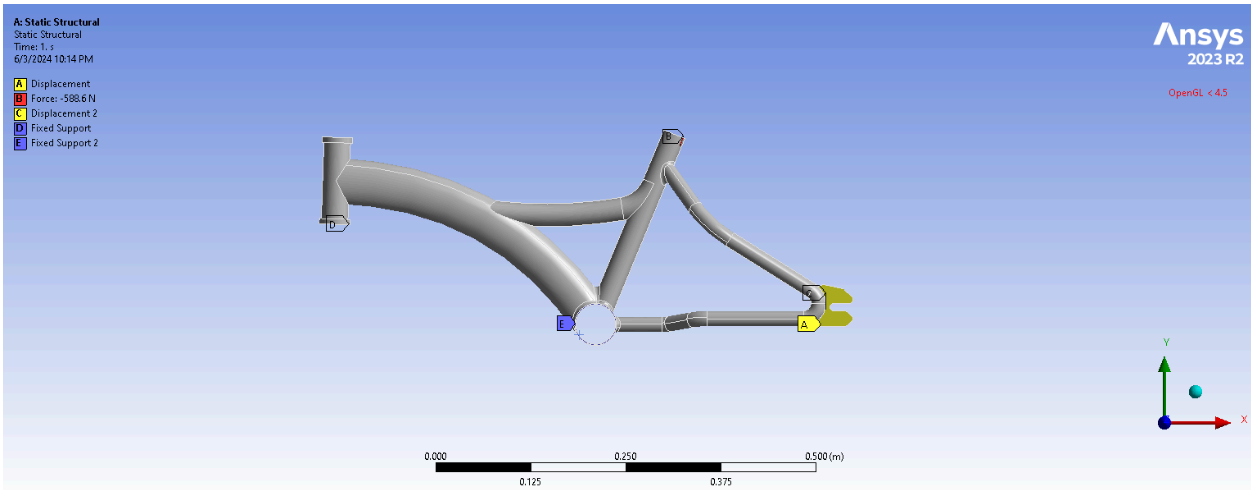
Problem 1

$F_a = mg = (60kg)(9.81\,m/s^2) = 588.6\,N$

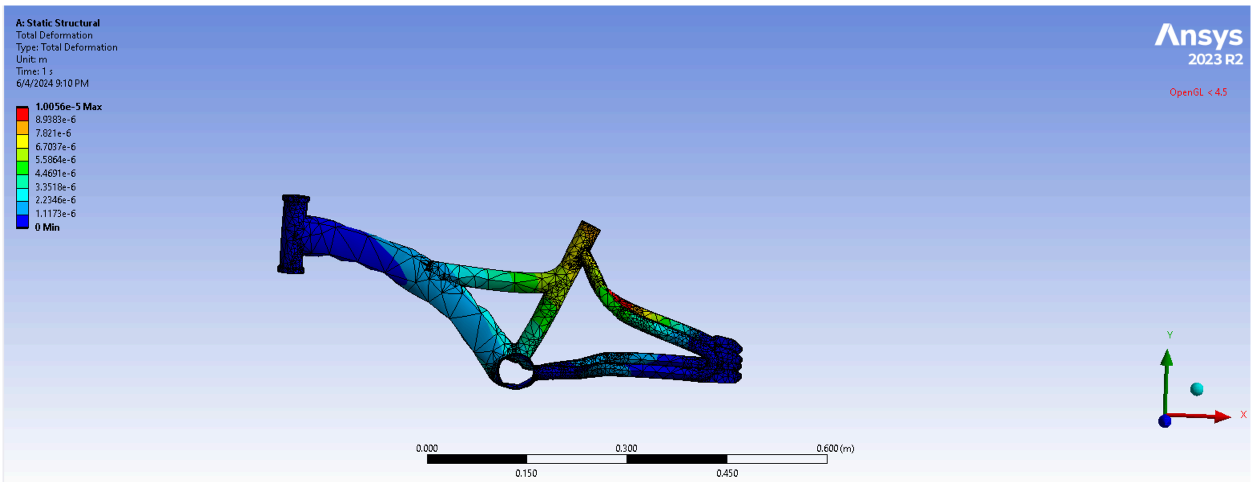
Mesh element order: quadratic

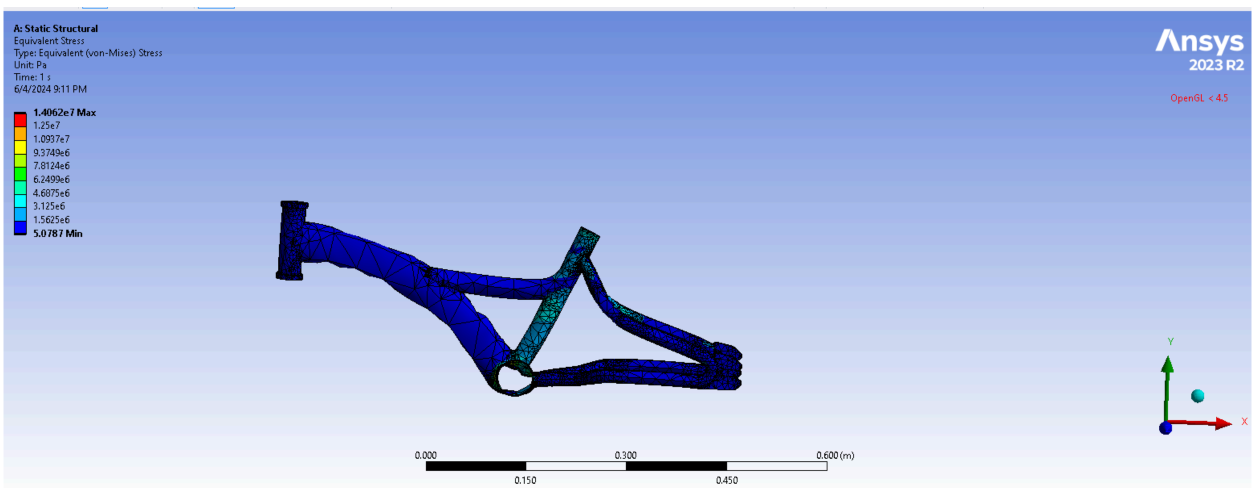
Mesh element size: 0.05m



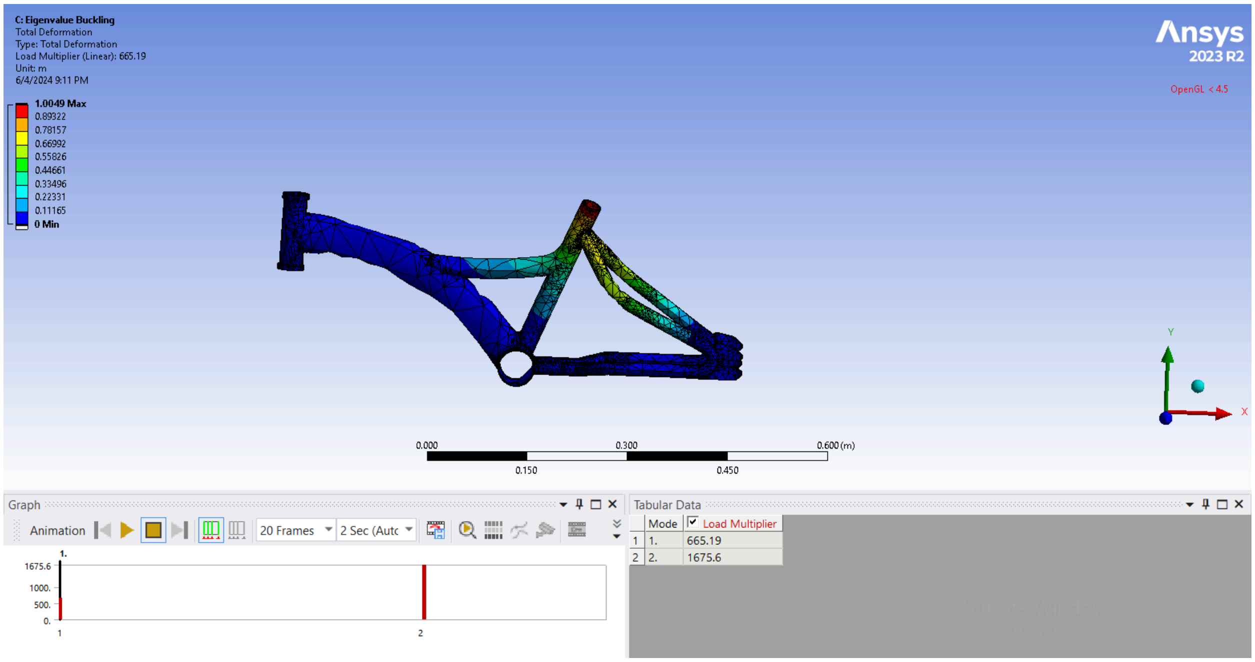


A force of -588.6N was applied on the top surface of where the bike seat would go.

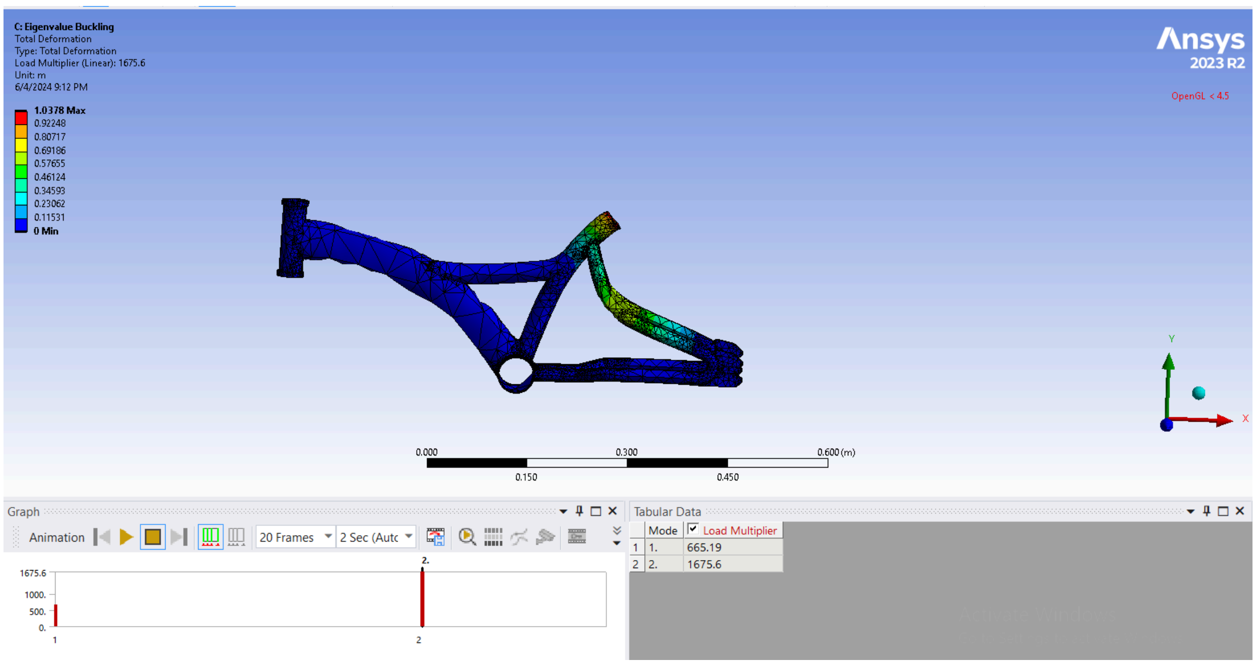




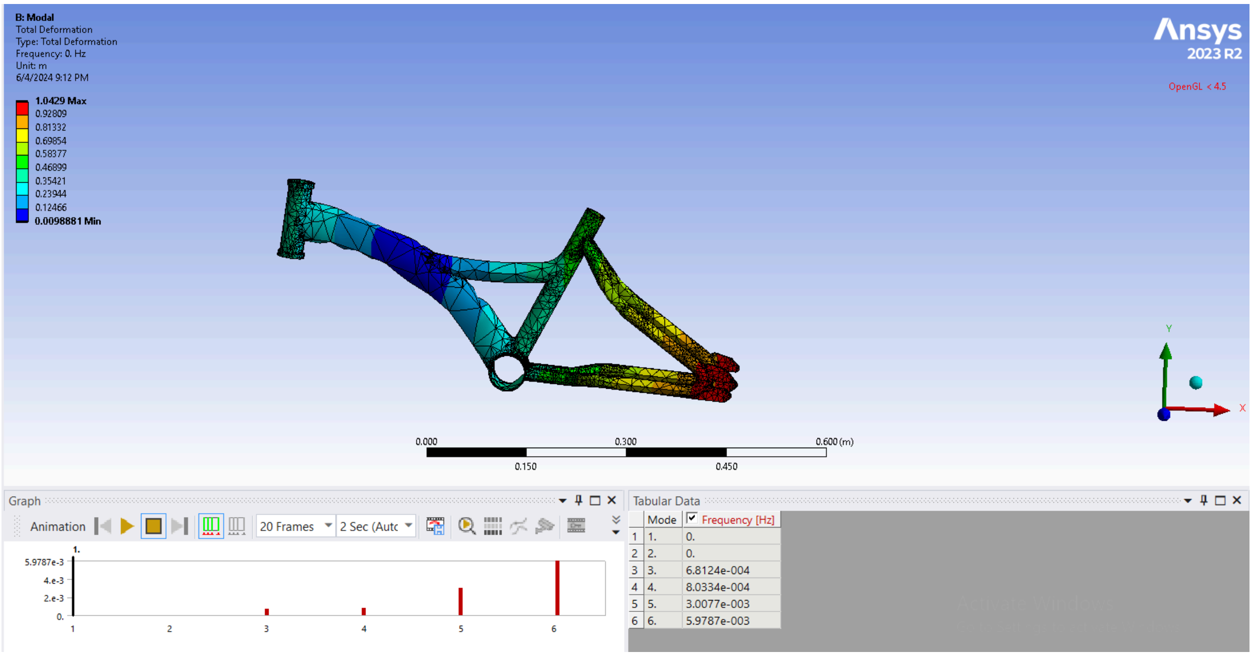
Buckling Analysis:
Mode 1


C: Eigenvalue Buckling
Total Deformation
Type: Total Deformation
Load Multiplier (Linear): 665.19
Unit: m
6/4/2024 9:11 PM
1.0049 Max
0.89322
0.78157
0.66992
0.55826
0.44661
0.33496
0.22331
0.11165
0 Min
Ansys
2023 R2
0.000
0.300
0.600 (m)
0.150
0.450
Graph
Animation
20 Frames
2 Sec (Auto
Tabular Data
Mode
Load Multiplier
1 1. 665.19
2 2. 1675.6

Mode 2


C: Eigenvalue Buckling
Total Deformation
Type: Total Deformation
Load Multiplier (Linear): 1675.6
Unit: m
6/4/2024 9:12 PM
1.0378 Max
0.92248
0.80717
0.69186
0.57655
0.46124
0.34593
0.23062
0.11531
0 Min
Ansys
2023 R2
0.000
0.300
0.600 (m)
0.150
0.450
Graph
Animation
20 Frames
2 Sec (Auto
Tabular Data
Mode
Load Multiplier
1 1. 665.19
2 2. 1675.6

Vibration responses:



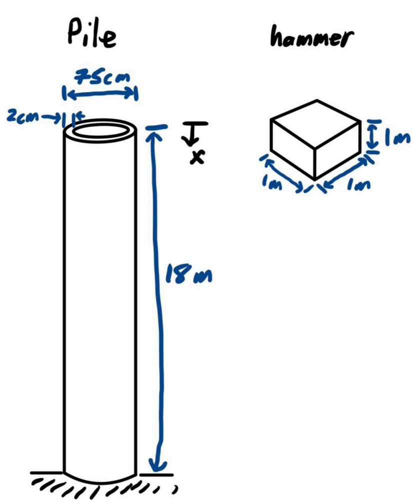
$FoS = Max\ stress/Working\ or\ Design\ Stress = 2.5E8\ Pa/1.4062E7\ Pa = 17.8$

Compressive yield = 2.5E8 Pa for structural steel

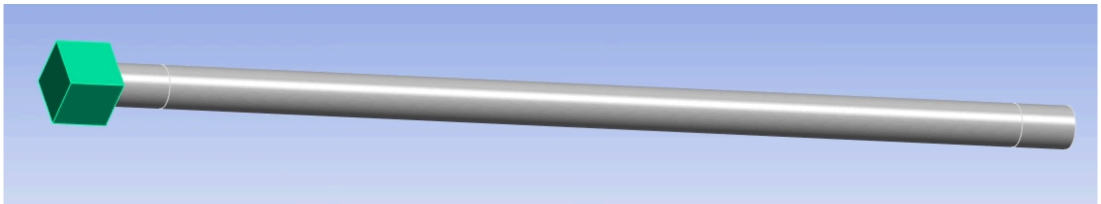
We would use a safety factor of 17.8 based on the compressive yield strength of structural steel and the max design stress.

We trust this model as a small mesh size and a quadratic mesh element order was used. A 62 kg person is about 132.77 lbs. Going off of the real world, a bike should be able to withstand the weight of a person greater than 132.77 lbs for a decent lifespan. Based on the safety factor, the bike is very safe so the bike frame should not fail with a 62 kg person riding it. This reflects the real world, and thus we trust this model.

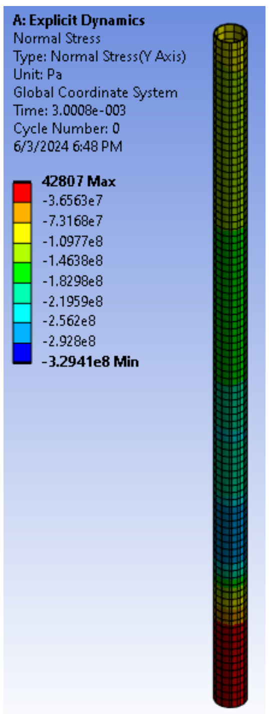
Problem 2



Structural steel was used for the pile. The pile was a thin feature with the dimensions shown. The hammer was made of a similar material, except with a density of 6000 kg/m^3 so the mass of 6000 kg needed a 1 m^3 volume. This was generated using a 1x1x1m cubic hammer geometry. The hammer-pile interface was frictionless and the bottom of the pile was fixed. A quadratic, 0.2m element size mesh was generated. The simulation time was set to the wave propagation time of T=L/c=(10 m)/(6000 m/s)=0.003s. The normal stress was utilized for the axial stress, which was measured at 1m and 17m using two probes. The geometry is shown:

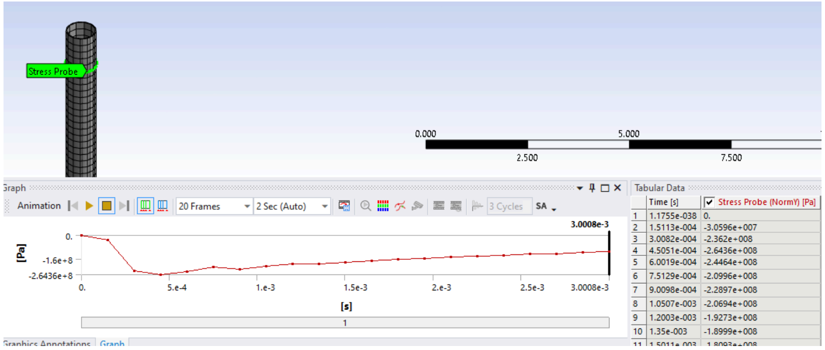


The overall stress distribution at the end of the simulation:

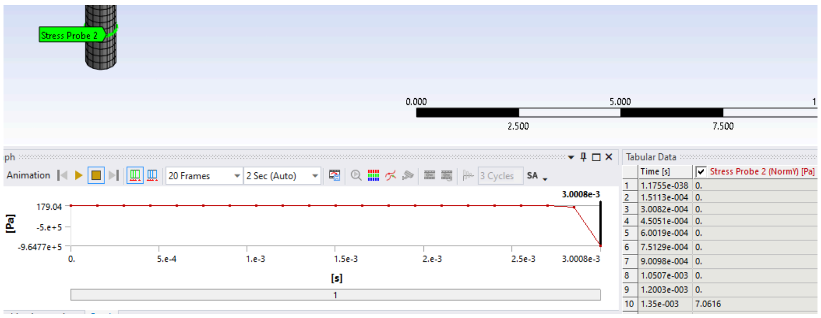


Note that the blue represents the compressive axial stress wave propagating along the pile. The max stress was 42.8 kPa and the minimum was -329.41 MPa. Though the wave starts as an impulse, it widens and the max amplitude drops as it travels, hence the min stress is only found at the moment of impact. The wave propagation speed was less than predicted as the wave did not finish traveling along the pile before the simulation ended.

The stress plots at the probes:



	Time [s]	Stress Probe (NormY) [Pa]
1	1.1755e-038	0.
2	1.5113e-004	-3.0596e+007
3	3.0082e-004	-2.362e+008
4	4.5051e-004	-2.6436e+008
5	6.0019e-004	-2.4464e+008
6	7.5129e-004	-2.0996e+008
7	9.0098e-004	-2.2897e+008
8	1.0507e-003	-2.0694e+008
9	1.2003e-003	-1.9273e+008
10	1.35e-003	-1.8999e+008

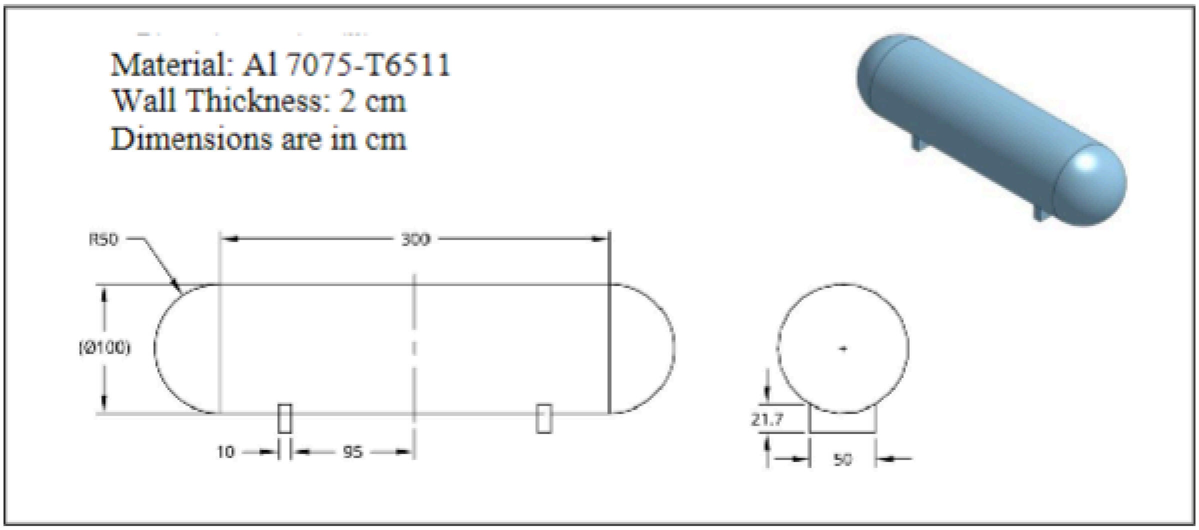


	Time [s]	Stress Probe 2 (NormY) [Pa]
1	1.1755e-038	0.
2	1.5113e-004	0.
3	3.0082e-004	0.
4	4.5051e-004	0.
5	6.0019e-004	0.
6	7.5129e-004	0.
7	9.0098e-004	0.
8	1.0507e-003	0.
9	1.2003e-003	0.
10	1.35e-003	7.0616

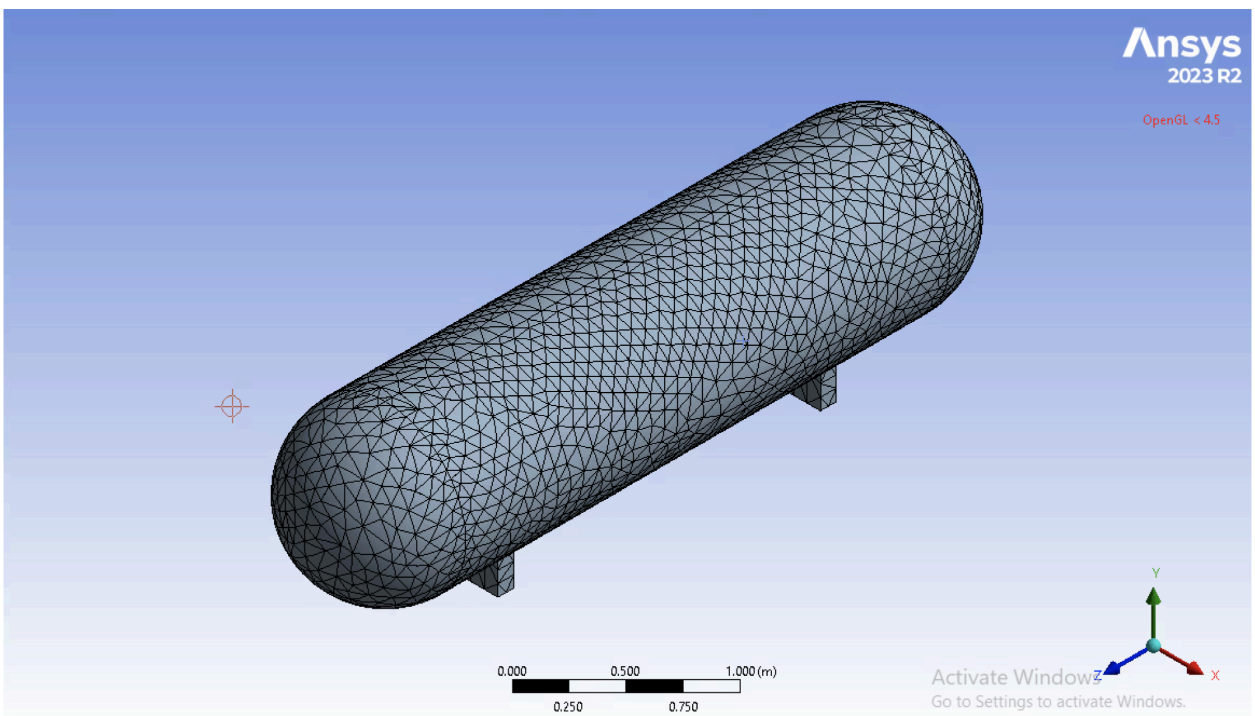
At 1m, the probe starts off unstressed, before the wave quickly propagates through it causing a sharp decrease to -264.36 Mpa, showing that the wave amplitude has already dropped significantly even 1m into the pile. It then slowly recovers however it does not reach 0 Pa by the time the simulation ends. It is this sluggish recovery which likely causes the widening of the wave. At 17m, it remains unstressed for most of the simulation due to the wave not reaching it. When it does, it starts decreasing. If the simulation continued, it would experience similar behavior as shown in the 1m probe albeit with a smaller min stress.

Problem 3

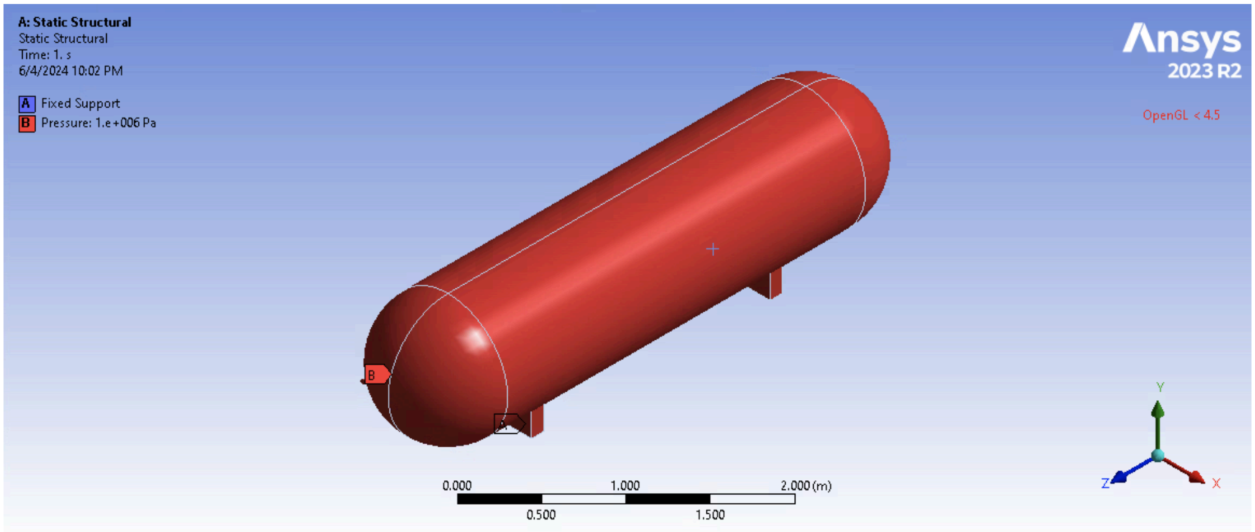
3. Determine the maximum safe water depth for the shown underwater vessel with a safety of 1.8 (to avoid buckling). Also determine how thick the wall thickness would need to be to withstand a depth of 4,000m with a safety factor of 1.25.



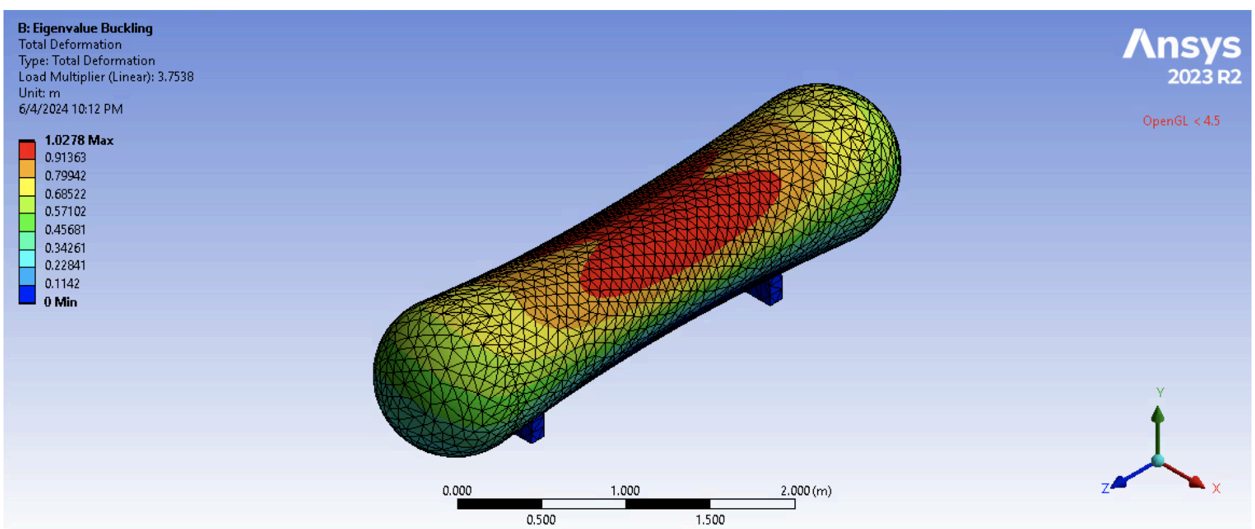
Mesh element order: quadratic
Mesh element size: 0.09m

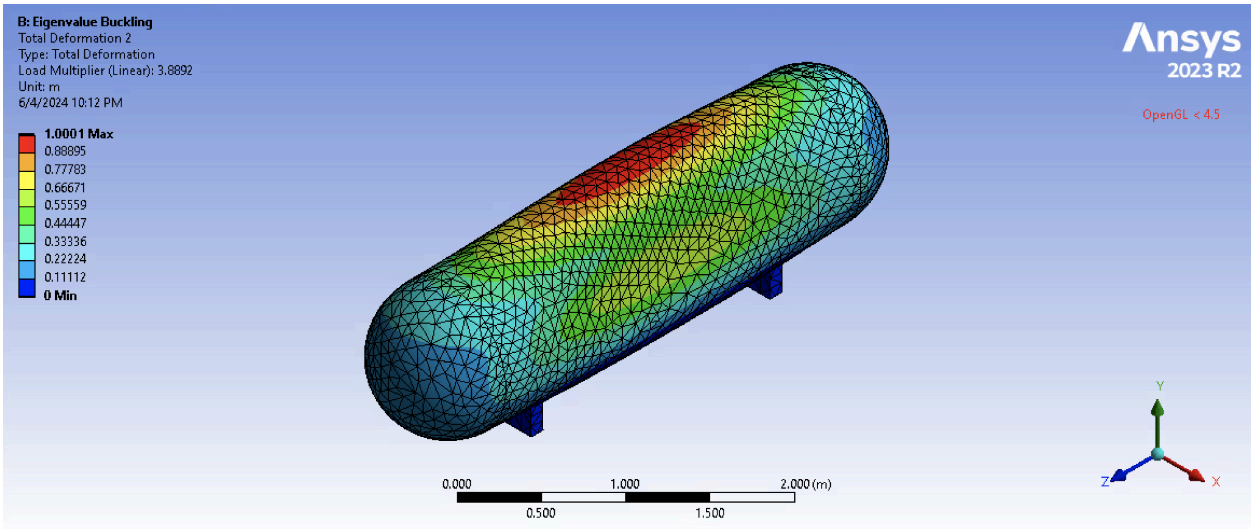


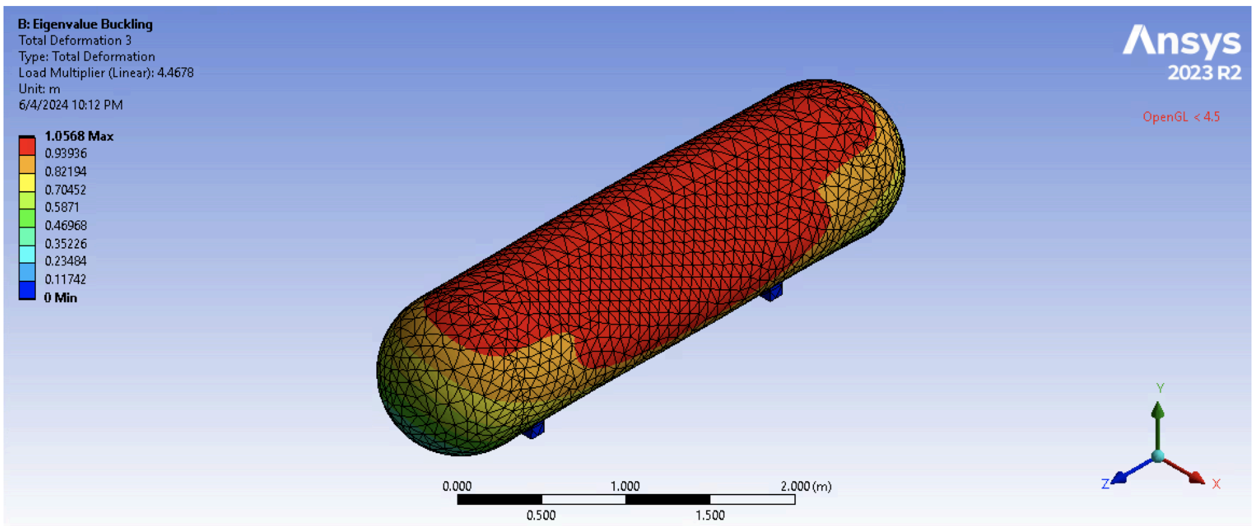
Fixed supports applied on vessel supports

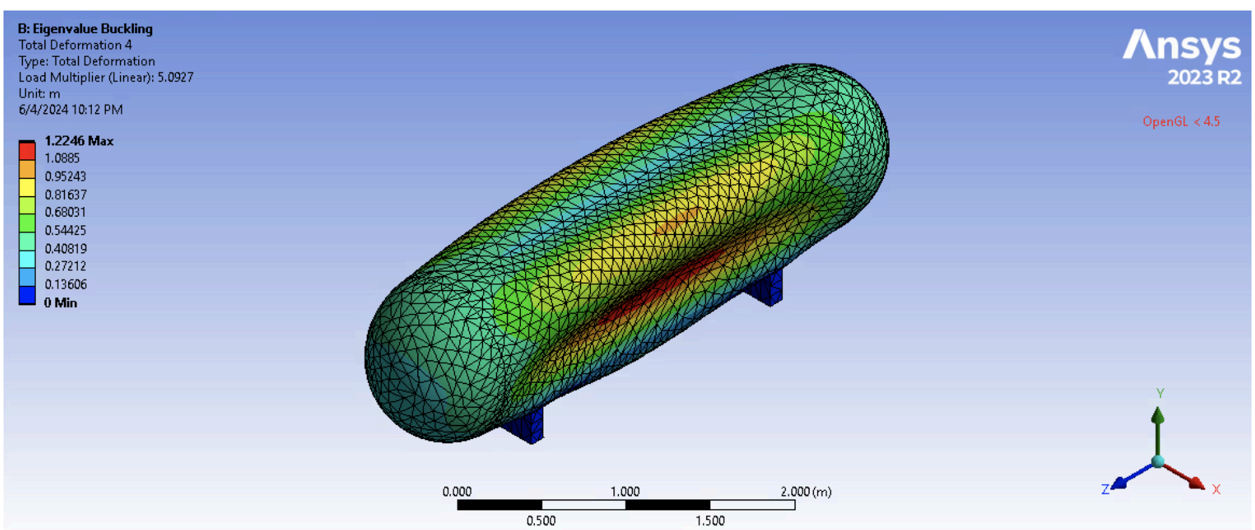


4 modes of deformation:









Calculations to find max water depth:

$1MPa(5.0927)/1.8\ (SF) = 2.829277778\ MPa$

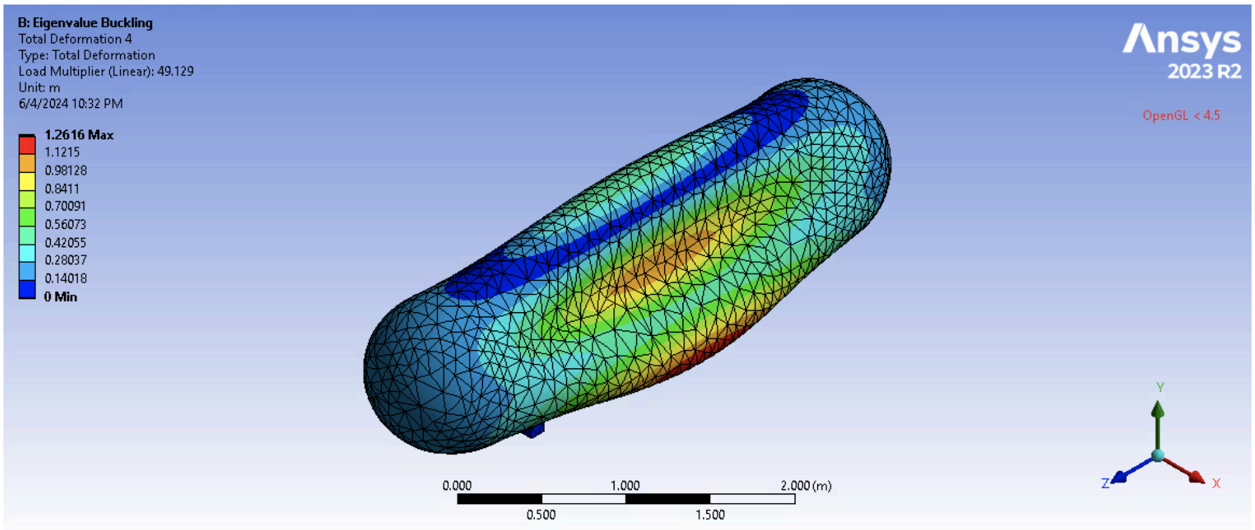
$P = \rho gh \rightarrow h = P/(\rho g) = 2.829\ MPa/(1000\ kg/m^3 * 9.81\ m/s^2) = 288.4075207 \approx 288.4\ m$

Max safe depth = 288.4 m

Calculations to find wall thickness at depth of 4000m

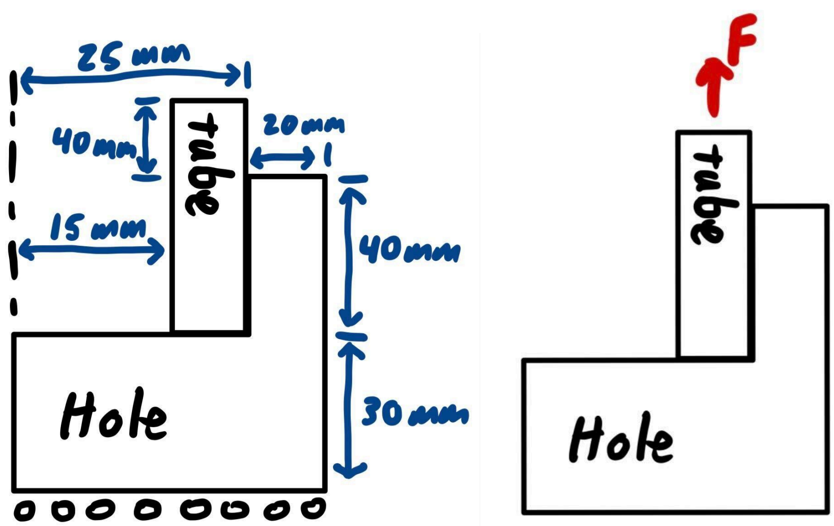
$P = \rho gh = (1000\ kg/m^3)(9.81\ m/s^2)(4000m) = 39240000\ Pa$

$P * SF = 39240000\ Pa(1.25) = 49050000\ Pa$



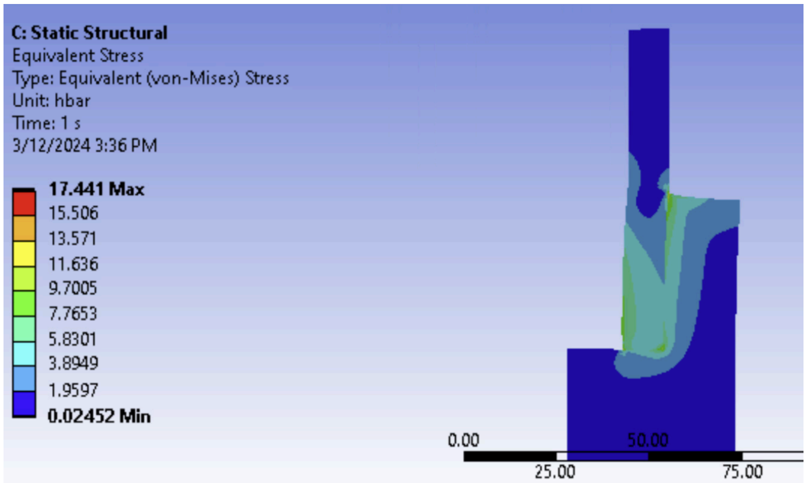
This results in a thickness of 45 mm through a guess and check until a resultant pressure close to 49.05 MPa on the analysis.

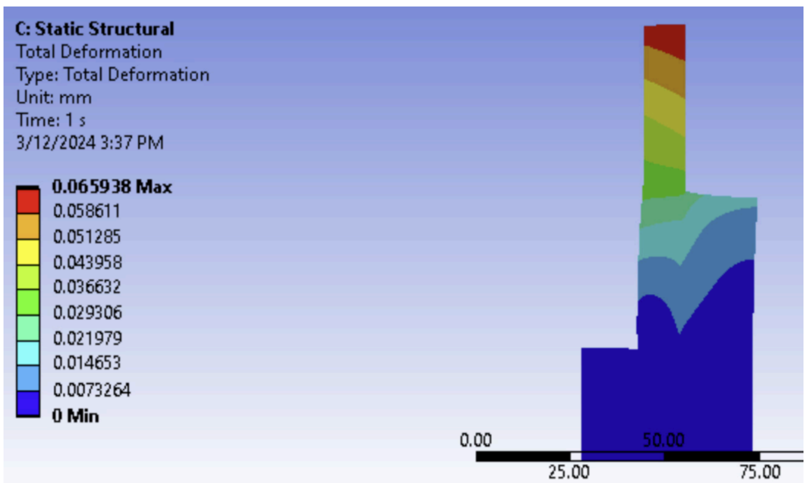
Problem 4



The scenario was modeled as a 2D surface using symmetry constraints to reduce the simulation to one half. Rollers were attached to the button of the hole to prevent vertical movement. A quadratic, 1mm element size mesh was generated. Thermal conditions of 62 and 22 Centigrade were placed on the tube and the hole respectively to generate the 40 Centigrade temperature differential desired. The contact region was specified to be frictional with a coefficient of friction of 0.4.

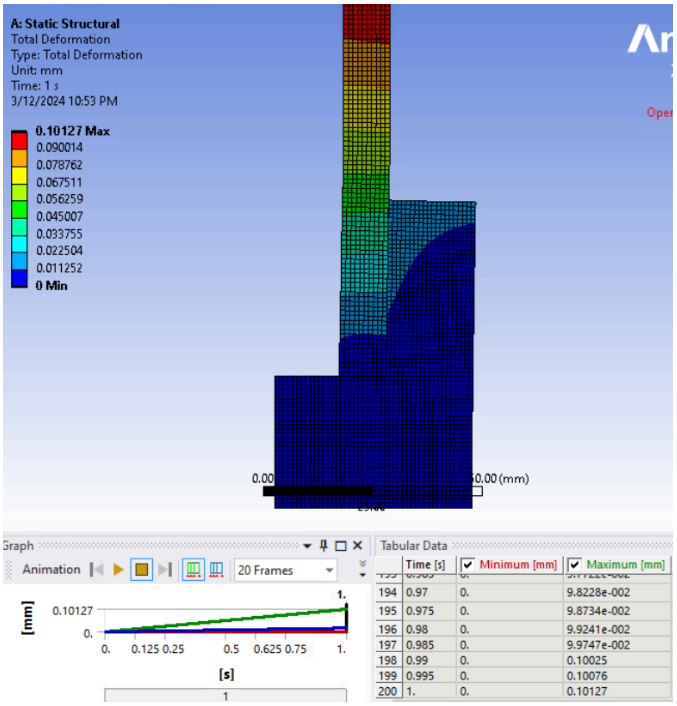
The following thermal expansion stresses and deformations were generated:





A max stress of 17.441 hectoBar and minimum of 0.03452 HectoBar ws generated. The highest stress was at the corner of full contact, and the tube experienced a higher average stress. Near the hole base and top of tube, the stress was negligible. The displacement was negligible at the bottom of the tube, and reached its maximum of 0.065938mm at the top of the tube. The shape of the tube is straight at the undeformed top then curves inward indicating that as the tube expanded the hole impeded it and squeezed the tube which generated the shown stresses.

A ramped pull-out force was then applied to the tube. The result is shown below:



Note that the maximum deformation is small, from the material strain, until t=0.99s when it jumps from 9.747E-2 mm to 0.1mm, indicating a larger deformation resulting from slippage. Referencing the ramped force graph for the corresponding force at that time, we get approximately 26 kN as the required force for pull out.

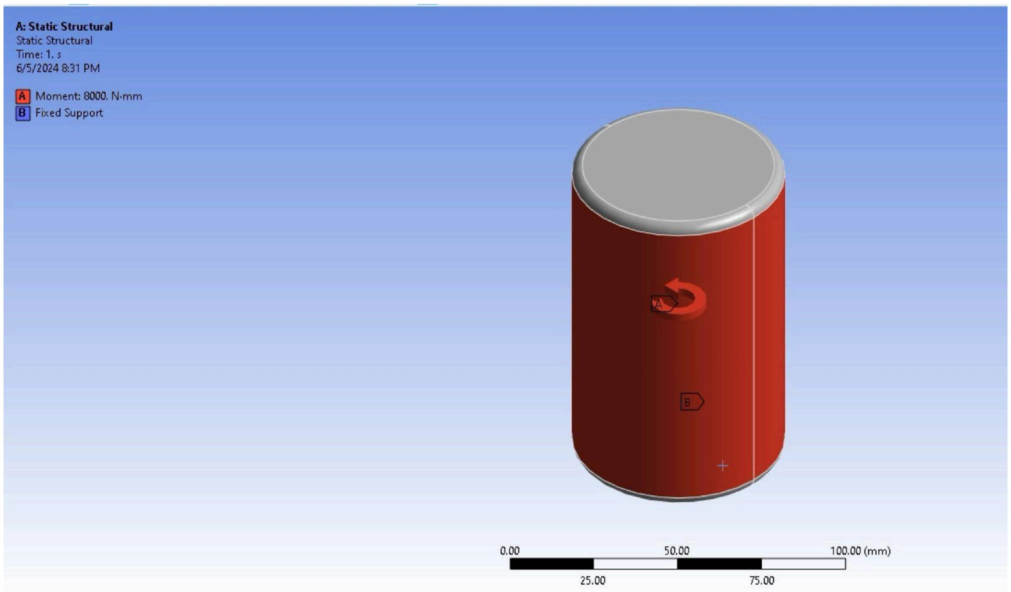
Problem 5

5. Applying a twist on a thin beverage can introduce a tensile stress in a principal direction and compressive stress in another principal direction. Excess comprehensive stress may cause the skin to buckle. Assume that the can is made of AA3004, which has a Young's modulus of 69 MPa and a Poisson's ration of 0.34.

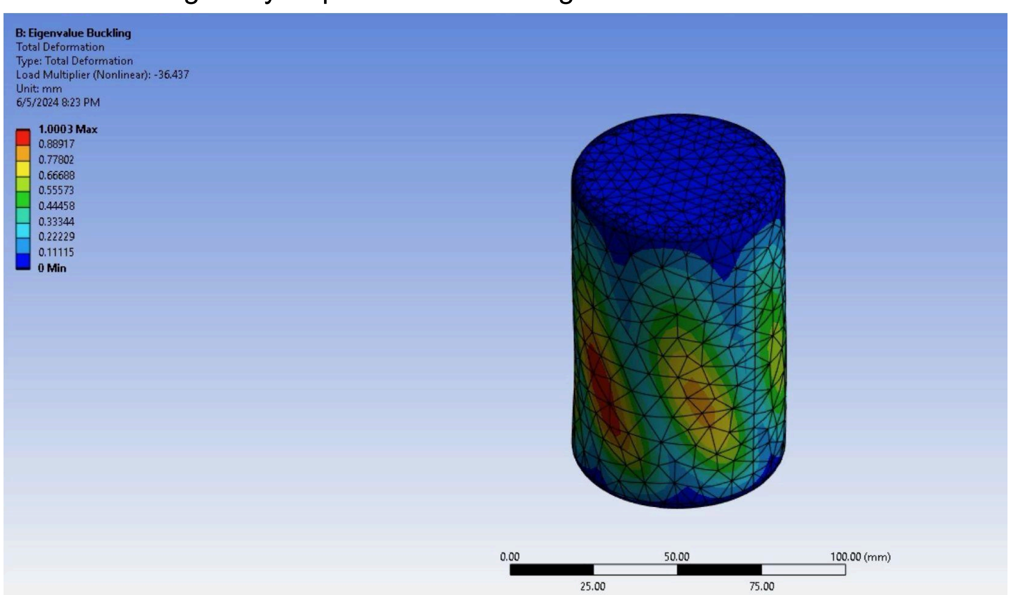
a) Predict the torque that causes the skin to buckle using a linear buckling analysis. Determine the buckling modes.

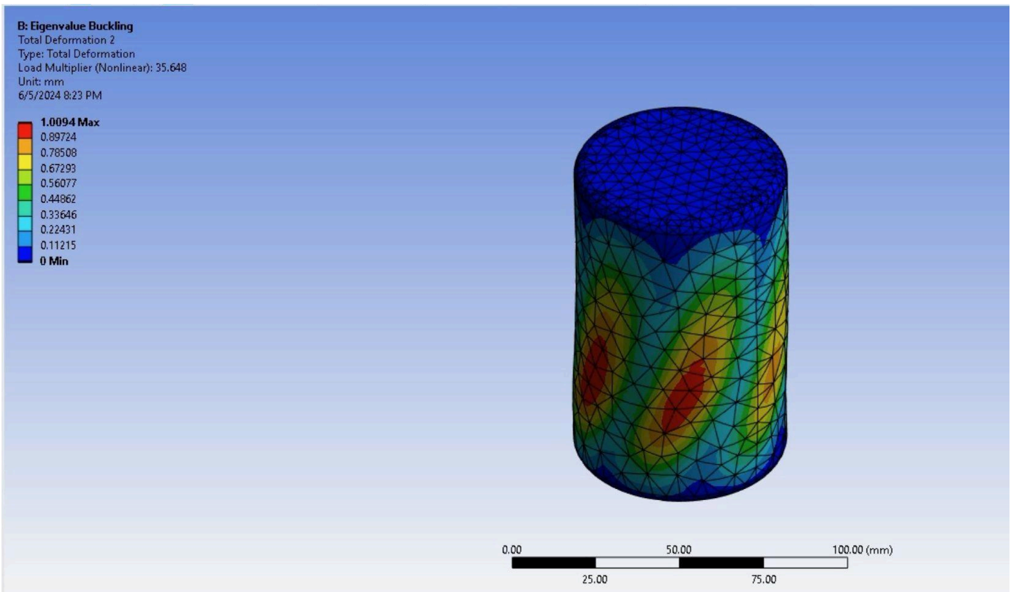
b) Compare the buckling torque that you got in part a) with the buckling torque you would get by performing a nonlinear buckling analysis.

a)
Can model imported into Ansys from SolidWorks
Mesh element order: quadratic
Mesh element size: default
Fixed support applied on the bottom surface with moment applied around the surface of the can at 200 N-mm torque.



Linear Buckling analysis performed resulting in 2 modes.

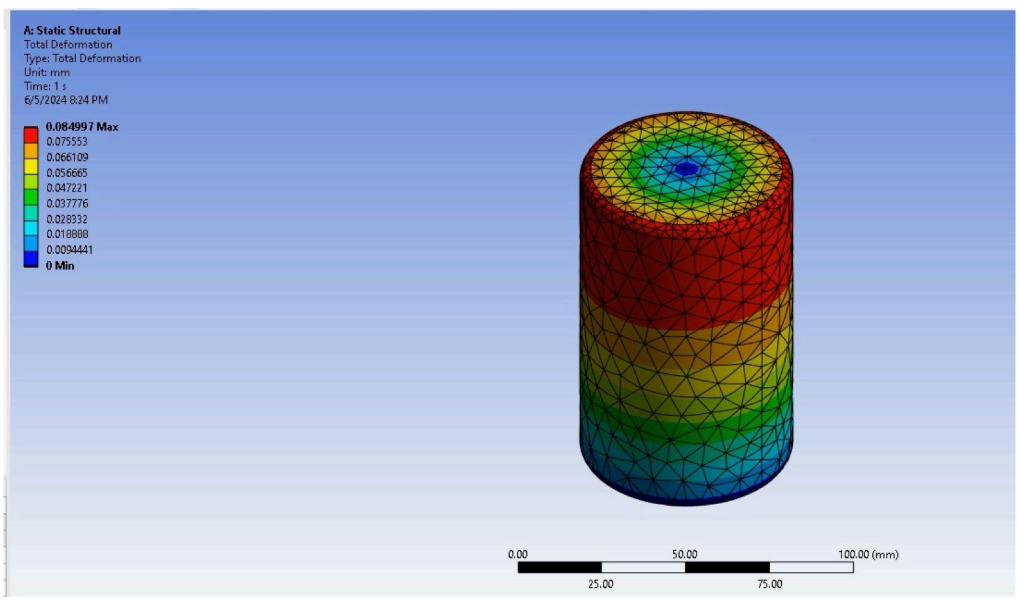


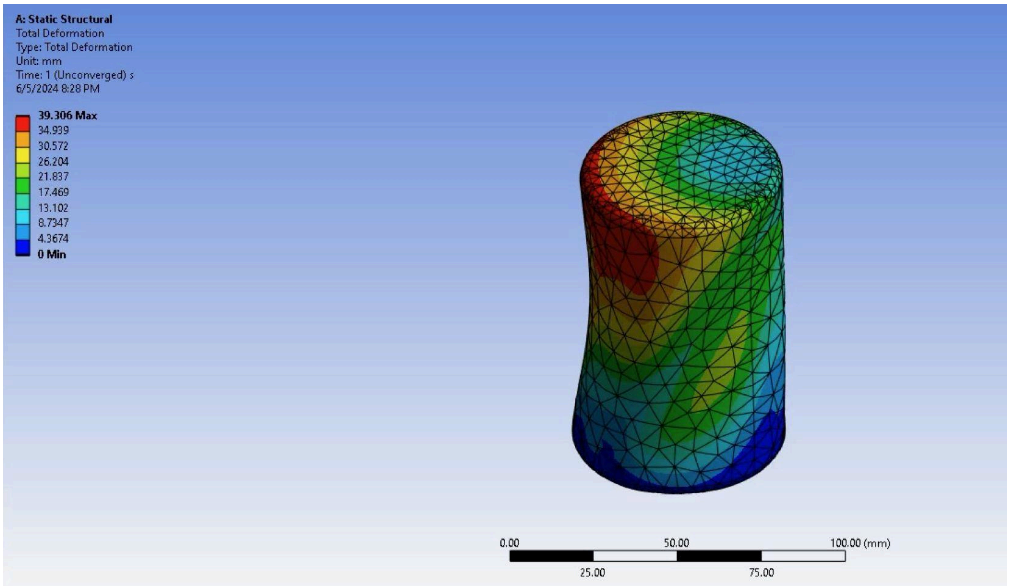


Mode 1 yields a load multiplier of 36.437 and mode 2 yields a load multiplier of 35.648. Based on the model, a torque of 7287.4 N-mm and 7129.6 N-mm will cause the skin to buckle for the respective modes.

b)

Non-linear analysis was performed that resulted in no buckling and buckling on the can. First, a moment of 7000 N-mm was applied – resulting in no buckling – and then a moment of 8000 N-mm was applied, which resulted in buckling.





The can buckles past a torque of 7000 N-mm, which agrees with the torques found using the linear buckling analysis.