MAE 3270, Fall 2025, Last HW, Design Project, Due Dec 8

This assignment pulls together several topics such as materials selection, fracture, fatigue, strain gauges, stress analysis and finite element analysis **This assignment counts as 3% of the course grade and cannot count as one your drops**. This will also be your portfolio assignment for 3270; do a good job so that you'll have something you can be proud of. We will integrate use of CAD and FEM tools along with hand calculations.

We will work on this together in lecture during the last three lecture meetings. You are encouraged, but not required to work in pairs. Everyone should submit their own assignment for their portfolio, noting who you worked with.

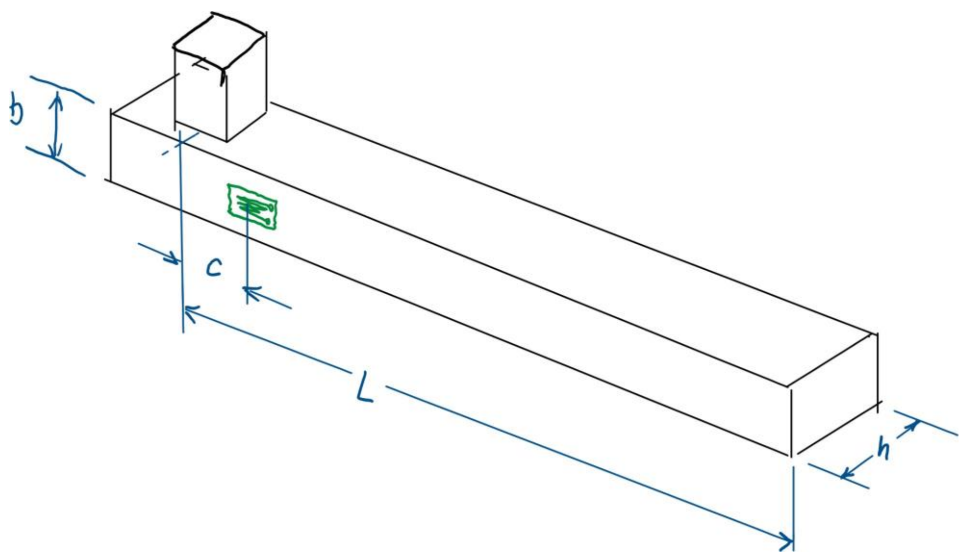


Figure 1: Schematic of torque wrench. Details of the drive dimensions are given in Figure 2. For first cut design you can assume that the load is applied at distance L from the drive. c is the distance from the drive to the strain gauge. The wrench handle itself is shown as rectangular with thickness b and width h. Your design need not be rectangular nor does it have to be of uniform shape and cross section. Note that actual wrench would be longer than L so that you there is room for the user to grip the wrench handle and apply the load.

1 Overview and Learning Goals

Your assignment is to design a non-ratcheting, 3/8 inch drive instrumented torque wrench rated for 600 in-lbf. A bare bones geometry is shown in the figure. Torque will be transduced using strain gauges bonded to the outer surfaces of the wrench at high strain locations. The basic analyses needed for the design can be performed by hand using your results from HWs 11, 12 and 13. I am also asking you to perform a finite element analysis of your final design.

The overall idea is to

- Learn to use ANSYS via an FEM analysis of a provided "baseline" design.

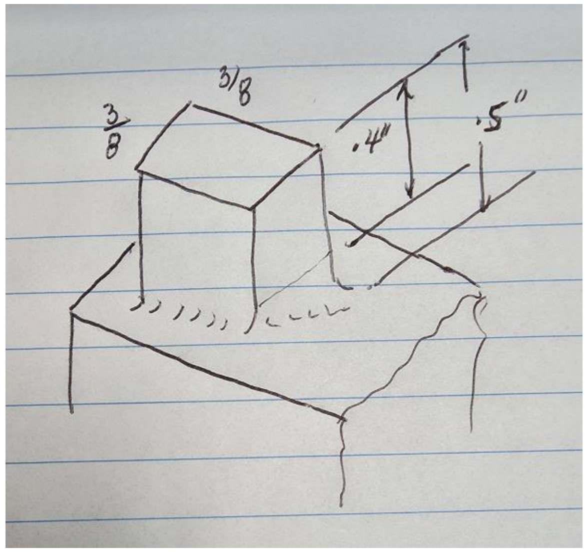


Figure 2: Details of drive dimensions. You can assume clamped boundary conditions in the upper 0.4 inches of the drive.

- Use hand, i.e. analytical calculations, to iterate the dimensions and material to come up with an improved design that meets all requirements
- Build a CAD model of your design. Your design need not be exactly what you came up with by hand calculations but the basic dimensions of the design should be guided by the hand calculations.
- Import the CAD model to ANSYS, perform the stress analysis, check that the design meets the requirements and determine the torque wrench sensitivity.

The learning goals are to

- Be able to perform first cut design considering multiple constraints.
- Be able to compare and understand results from FEM and hand calculations.

2 Design Using Hand Calculations

The torque will be transduced by strain gauges on the sides of the torque wrench. The design goal is to maximize the voltage output of the wrench (mV/V) at the rated torque. The design is required to attain at least 1.0 mV/V output at the rated torque of 600 in-lbf. Higher output will lead to more sensitivity and improved signal to noise ratio. The constraints are that the wrench must not fail due to static loading, crack growth or fatigue. The wrench must sustain a fully reversed torque of $T = \pm 600$ in-lbf for 10^6 cycles. Design will include selecting an appropriate material and dimensions to meet or exceed the following requirements:

- attain at least 1.0 mV/V output at the rated torque of 600 in-lbf.

- safety factor of $X_o = 4$ for yield or brittle failure (you pick which criterion based on whether you are using a brittle or ductile material)
- safety factor of $X_K = 2$ for crack growth from an assumed crack of depth 0.04 inches (1 mm).
- fatigue stress safety factor of $X_S = 1.5$.
- material must be a steel, aluminum or titanium alloy.

I strongly suggest you write a Matlab or Python script to perform your hand calculation design iterations.

Below is *one* design. You can use these results to check your numerical calculations. Note that the torque wrench output is low and that the design does *not* meet the sensitivity specification. You can, and are required, to do better.

Load, dimensions and mechanical properties:

```
M = 600;              % max torque (in-lbf)
L = 16;               % length from drive to where load applied (inches)
h = 0.75;              % width
b = 0.5;              % thickness
c = 1.0;              % distance from center of drive to center of strain gauge
E = 32.E6;            % Young's modulus (psi)
nu = 0.29;             % Poisson's ratio
su = 370.E3;          % tensile strength use yield or ultimate depending on material (psi)
KIC = 15.E3;          % fracture toughness (psi sqrt(in))
sfatigue = 115.e3;    % fatigue strength from Granta for 10^6 cycles
name = 'M42 Steel'; %  material name
```

Stress and deflection analysis

```
load point deflection = 0.091 in
max normal stress = 12.80 ksi
```

Safety factor results:

```
safety factor for strength = 28.9
safety factor for crack growth = 2.95
safety factor for fatigue = 8.98
```

Strain gauge results:

```
strain at gauge = 375 microstrain
output = 0.38 mV/V at 600 in-lbf using half bridge
```

3 Analysis of Baseline Design

Following the example in lecture, import a model of the baseline design in Ansys and perform an FEM analysis under a torque of T = 600 in-lbf (lateral force of F = T/L). Use the results from this model to address the questions below which ask you to compare hand and FEM calculations.

4 Your Design

Using the code you developed for the hand calculations, iterate the design, including material and dimensions to come up with an improved design that meets all design requirements.

4.1 CAD

Build a CAD model based on your improved design. We recommend Autodesk Fusion for ease of integration with ANSYS.

4.2 FEM Analysis

Import your CAD model into ANSYS. (Instructions on how to do this will be provided.) Load with a torque of T = 600 in-lbf and solve the model. Perform at least one mesh refinement study, i.e. reduce the mesh size and re-run the analysis.

5 What to include in your HW and portfolio

Your submission will consist of two parts. A PDF file with the items below, submitted to Gradescope and a link to your portfolio submission, submitted via Canvas and the portfolio submission itself.

5.1 Baseline Design - PDF File Submitted to Gradescope

5.1.1 Results

1. Script used for hand calculation.

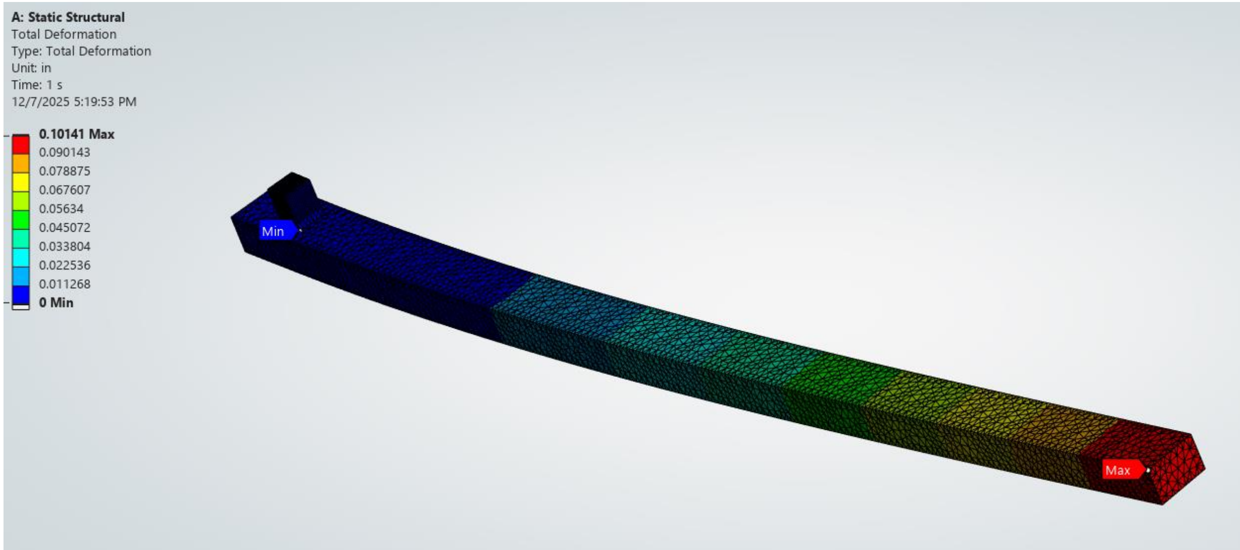
```
%Materials Torque Wrench Final
%Caroline Moskal, December 2, 2025
clear;

%Parameters
M = 600;%Max Torque(ft-lbs)
L = 16;%Length from drive to load(inch)
h = 0.75;%Width(inch)
b = 0.5;%Thickness(inch)
c = 1.0;%Distance from center of strain gauge to center of drive(inch)
%Material Properties
%name = M42_Steel;%Material
E = 32e6;%Young's Modulus
nu = 0.29;%Possion Ratio
su = 370e3;%Strength
KIC = 15e3;%Fracture Toughness
sFatigue = 115e3;%Fatigue Strength, Granta, 10^6 Cycles
%Givens
a = 0.04;%Crack
K = 2.0;%Strain Gauge Factor

%Deflection/Strain
I = b*h^3/12;
Force = M/L;
```

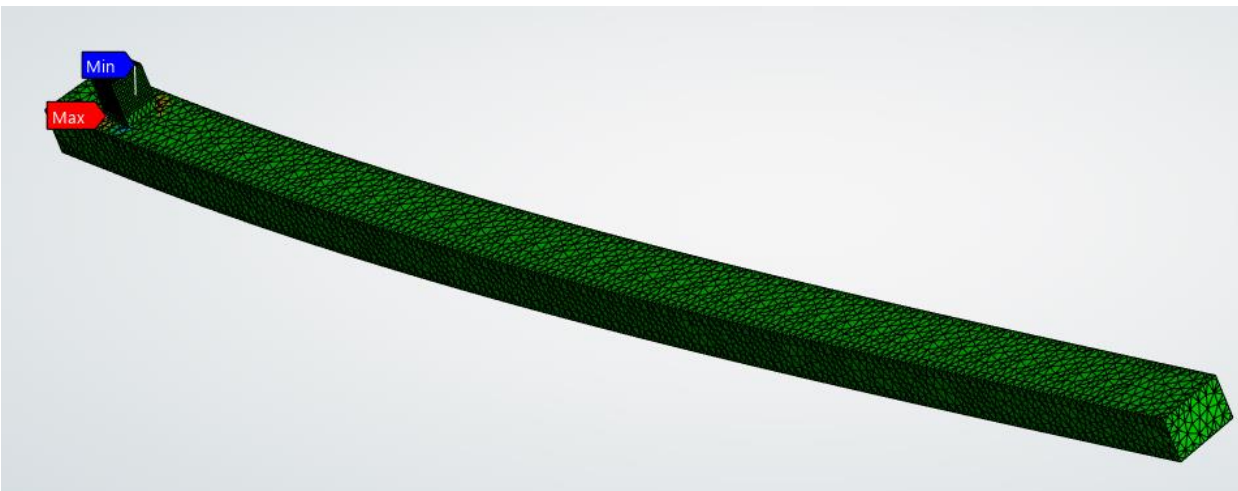
```
Deflection = M*L^2/(3*E*I);
%Deflection at Strain
Strain_Moment = (M/L)*(L-c);
Strain_Stress = 1e6*(Strain_Moment*(h/2)/(I*E));
%Strain_Deflection = (Force*(c^2)*(3*L-c))/(6*E*I);


%Strength
Max_Stress = (M*(h/2))/I;
%Crack Strength
KI = 1.12*Max_Stress*sqrt(a)*sqrt(3.14);

%Outputs
SF_Strength = su/Max_Stress;
SF_Crack = KIC/KI;
SF_Fatigue = sFatigue/Max_Stress;
Output = (K*Strain_Stress)/2000;

fprintf('Deflection                        = %.3f in\n', Deflection);
fprintf('Normal Stress                     = %.2f ksi\n', Max_Stress/1000);
fprintf('Safety Factor For Strength        = %.1f \n', SF_Strength);
fprintf('Safety Factor For Crack Growth    = %.2f \n', SF_Crack);
fprintf('Safety Factor For Fatigue         = %.2f \n', SF_Fatigue);
fprintf('Strain at Gauge,                  = %.0f microstrain\n', Strain_Stress);
fprintf('Output,                           = %.2f mV/V\n', Output);
```

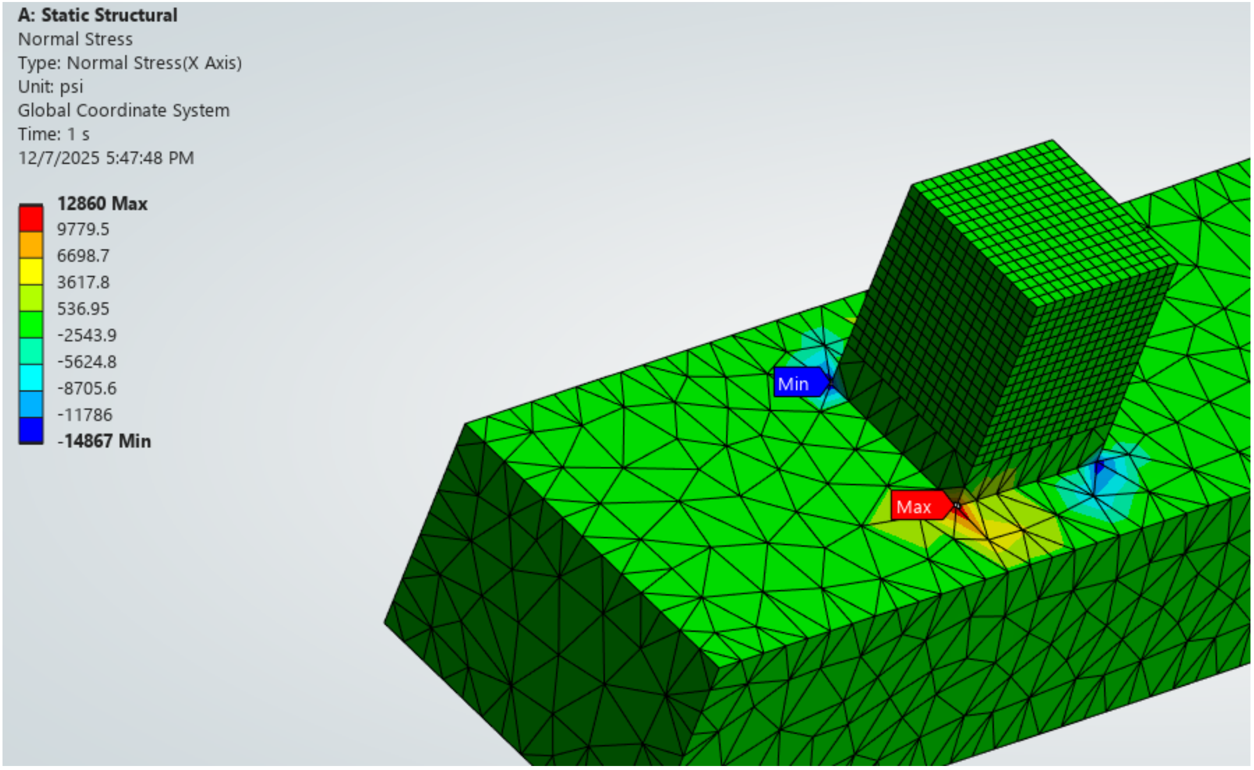
2. Results from hand calculation of base design showing maximum normal stress (anywhere), strains at the strain gauge locations and deflection of the load point.

```
>> MAE3270_Torque_Wrench
Deflection                     = 0.091 in
Normal Stress                  = 12.80 ksi
Safety Factor For Strength     = 28.9
Safety Factor For Crack Growth = 2.95
Safety Factor For Fatigue      = 8.98
Strain at Gauge,               = 375 microstrain
Output,                        = 0.38 mV/V
```

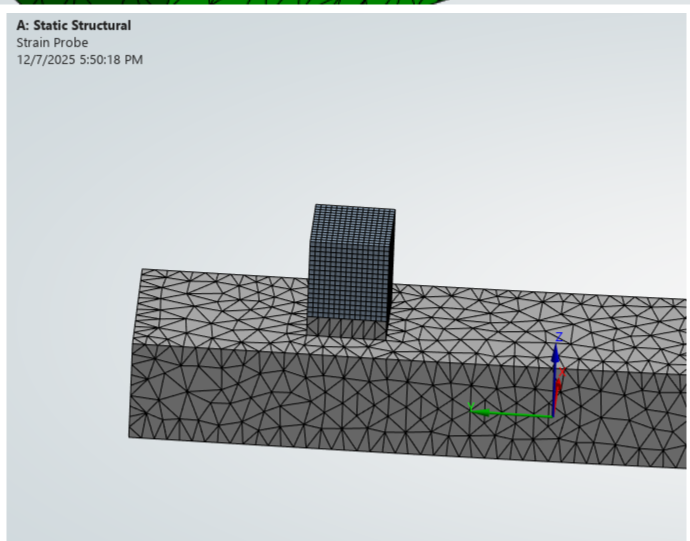
3. Results from FEM calculation of base design. From the FEM find the maximum normal stress (anywhere), strains at the strain gauge locations and deflection of the load point.


A: Static Structural
Total Deformation
Type: Total Deformation
Unit: in
Time: 1 s
12/7/2025 5:19:53 PM
0.10141 Max
0.090143
0.078875
0.067607
0.05634
0.045072
0.033804
0.022536
0.011268
0 Min
Min
Max


Min
Max



Maximum Value Over Time	
Normal - X Axis	-1.857e-006 in/in
Normal - Y Axis	1.8301e-006 in/in
Normal - Z Axis	-7.5758e-009 in/in
XY Shear	-3.8559e-006 in/in
YZ Shear	3.6009e-007 in/in
XZ Shear	-2.8196e-007 in/in
Equivalent (von-Mises)	3.5948e-006 in/in
Maximum Principal	2.6723e-006 in/in
Middle Principal	-2.4498e-008 in/in
Minimum Principal	-2.6823e-006 in/in
Intensity	5.3546e-006 in/in



5.1.2 Reflections

1. Beam theory assumes that plane sections remain plane. View the deformed mesh and check if mesh lines that cut across the beam handle remain as straight lines. Do you think that beam theory is reasonably accurate?

When looking at the deformation of the mesh, it seems that the mesh lines remain straight, this view is from the top section of the torque wrench. Since the lines are straight this indicates that Beam theory assumption is valid for the model. If the lines were curved or in some way discontinuous, it would indicate that the cross sections are no longer planner, and beam theory would not be valid.

2. How do the FEM and hand calculated maximum normal stresses compare? If they differ significantly, why?

The FEA and hand-calcs were pretty similar when comparing the max displacement and strain at the strain gauge. When comparing these numbers from the FEM and the matlab script:

Max Deflection: 0.1’’(FEM)-0.091’’(MATLAB)

Strain at Strain Gauge: 0.359microstrain(FEM)-0.375microstrain(MATLAB)

The only assumption is within the strain at the strain gauge, where the FEM is taking a Von Mises approximation, where the MATLAB uses an approximation of the rosette strain gauge.

3. How do the FEM and hand calculated displacements compare? If they differ, why?

The normal stress calculated values are very similar to each other. Although these values are very similar to each other, I would note that the max stress presented on the FEM is at a stress concentration near the boundary condition of the torque wrench drive. However, this is very similar to the assumption that was made in calculating the max stress at the outer edge of the torque of where the boundary condition is placed.
I would also like to note that the stress between the compressive and tensile area of the drive in the torque wrench are different, with the compressive stress being closer to 14867psi. Since the cross-section of the torque wrench being constant, this discrepancy most likely due to a unsymmetric mesh.

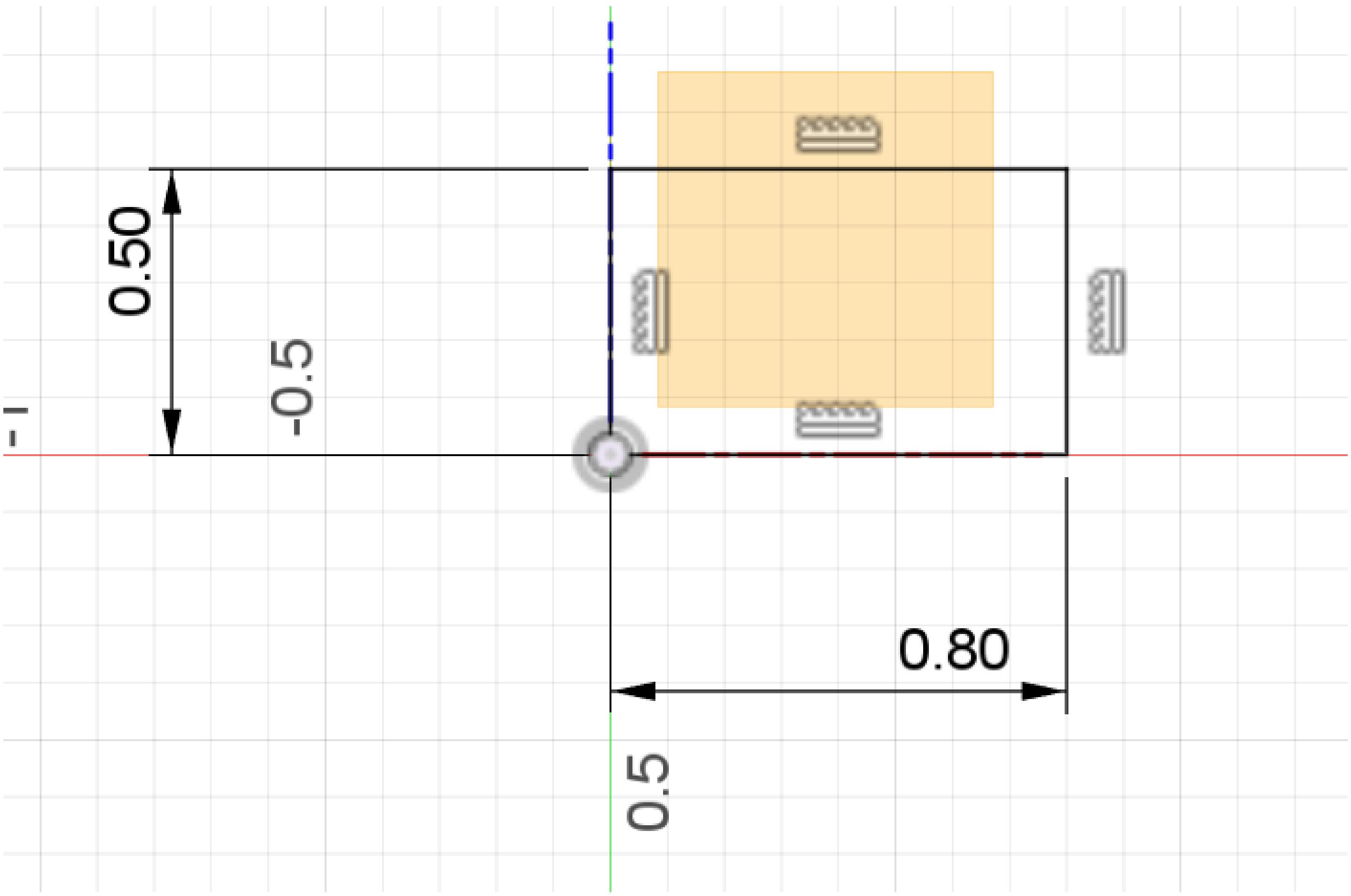
Max Normal Stress: 12860psi(FEM)-12800psi(MATLAB)

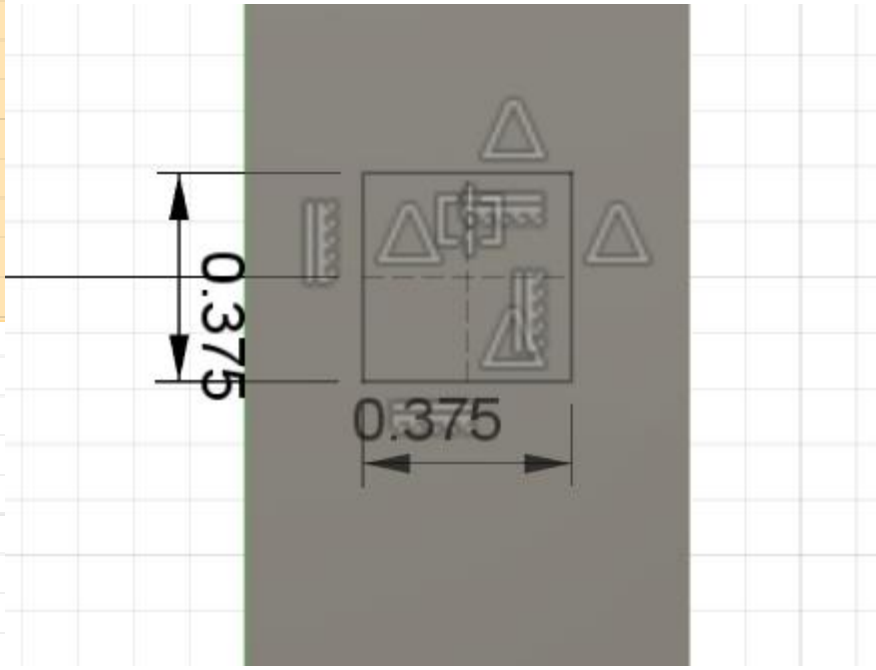
5.2 Your Design, Upload to portfolio

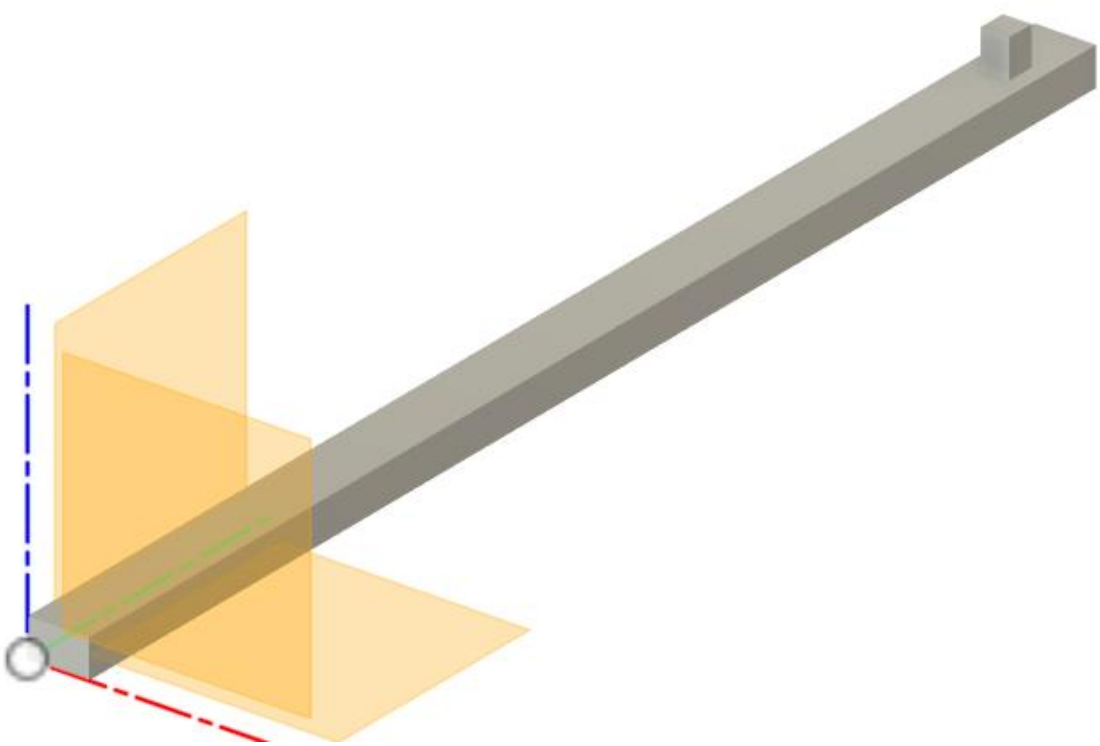
5.2.1 Results

1. Image(s) of CAD model. Must show all key dimensions.

```
L = 16;%Length from drive to load(inch)
h = 0.8;%Width(inch)
b = 0.5;%Thickness(inch)
c = 0.25;%Distance from center of strain gauge to center of drive(inch)
```



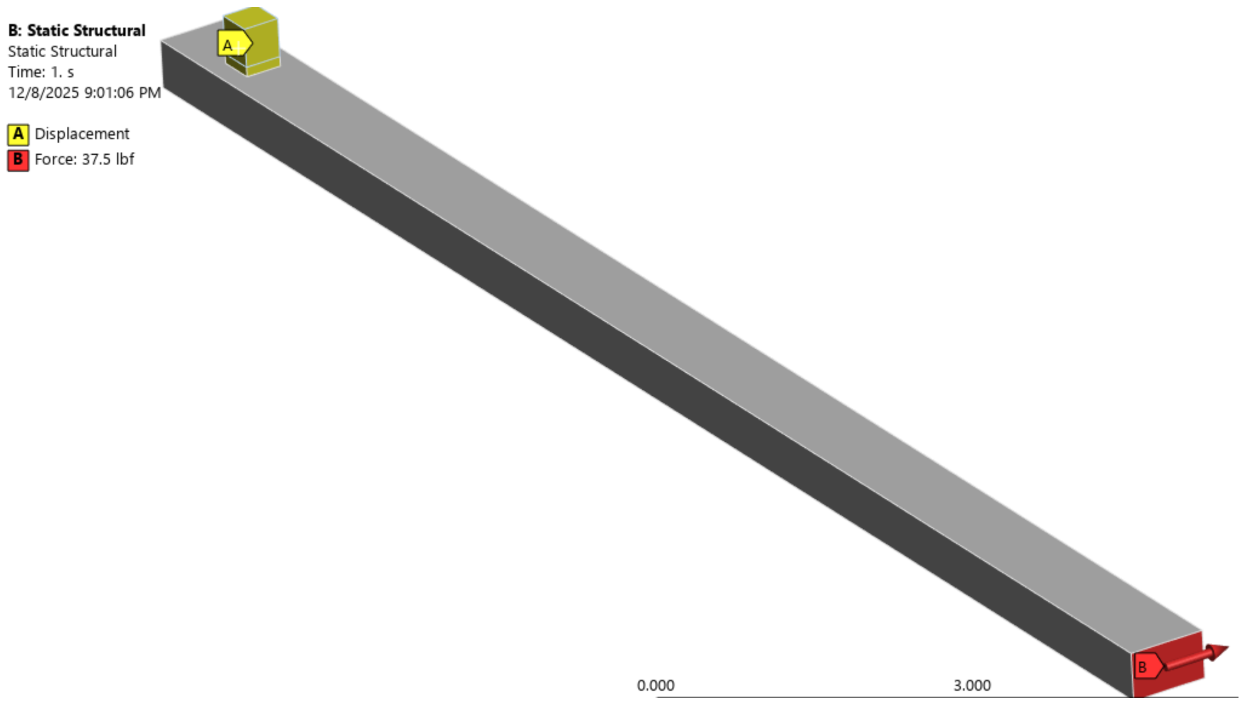


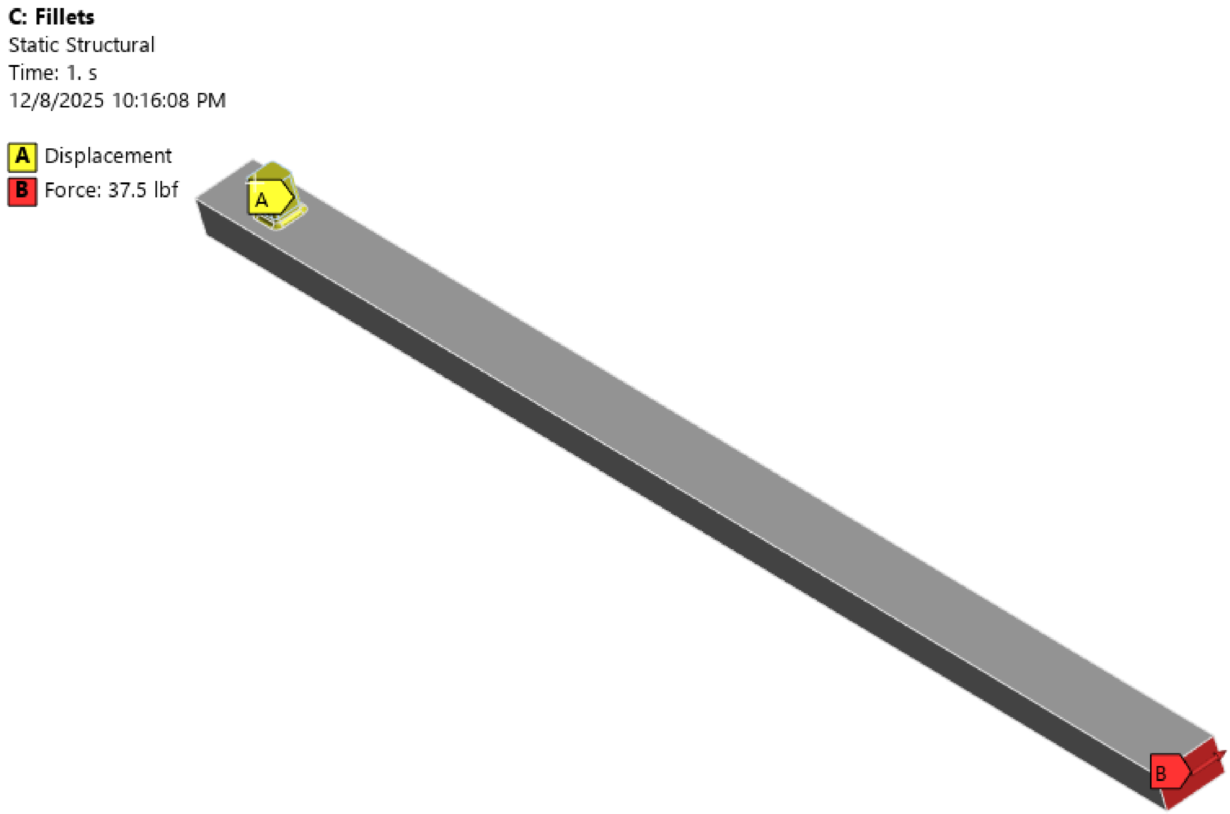


2. Describe material used and its relevant mechanical properties.
   Aluminum 7075
   Young's Modulus: 10.55e6 (psi)
   Poisson Ratio: 0.335

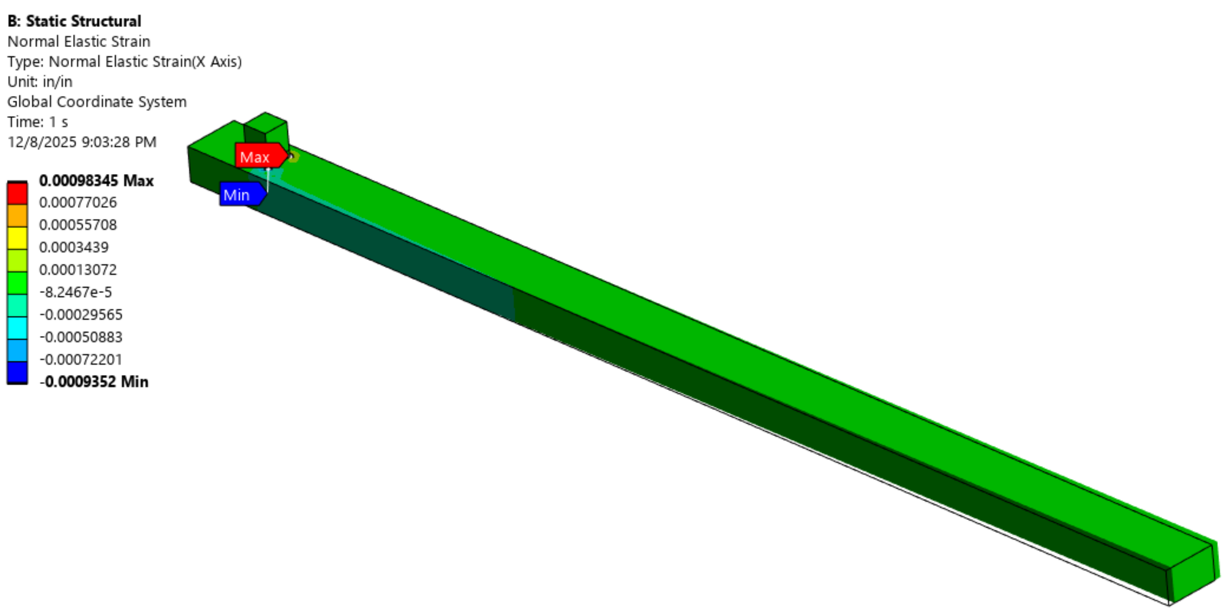
```
Strength: 50e3 (psi)
Fracture Toughness: 29.5e3 (psi*sqrt(inch))
Fatigue Strength (Granta, 10^6 Cycles) = 22.2e3 (psi)
```

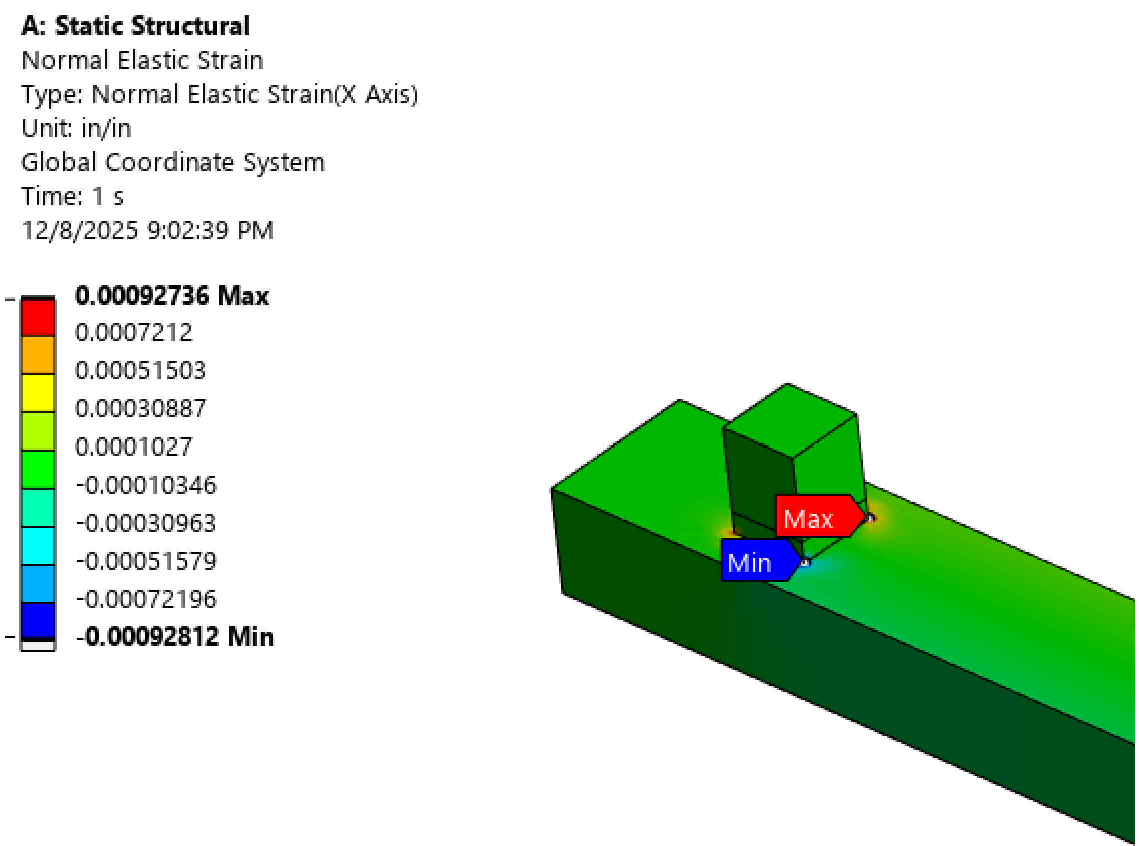
3. Diagram communicating how loads and boundary conditions were applied to your FEM model.

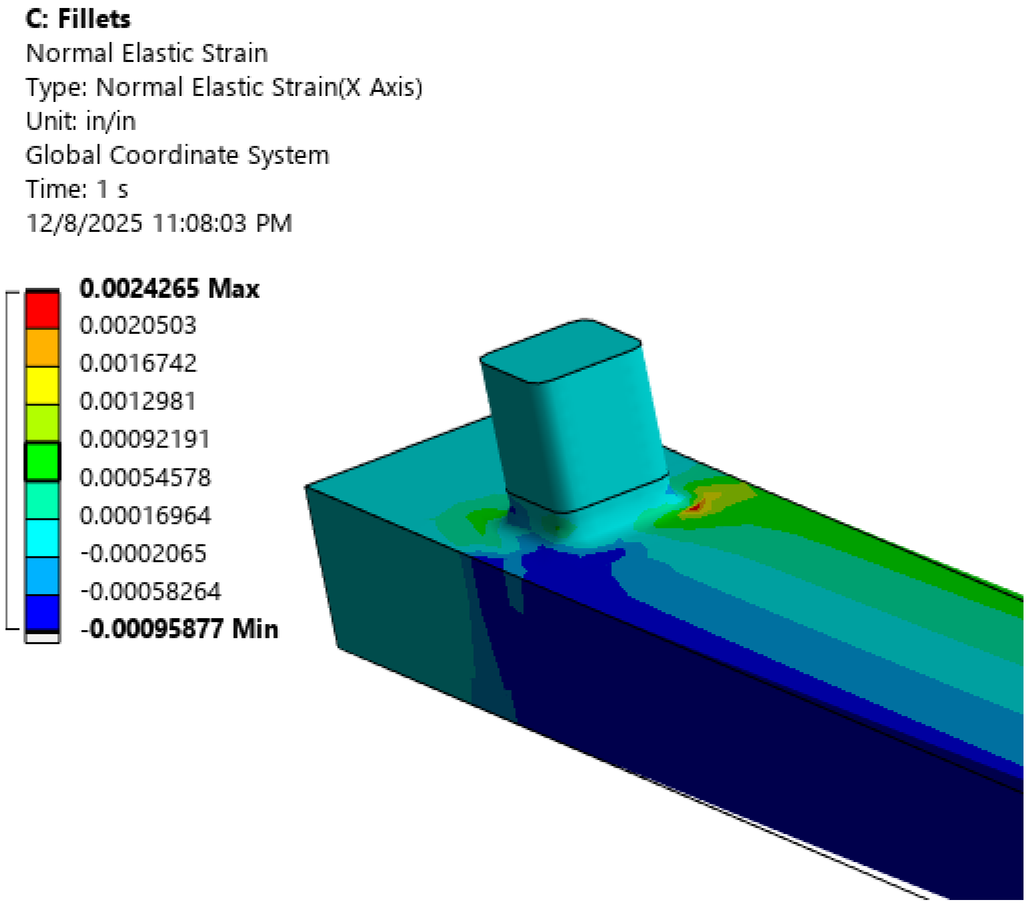




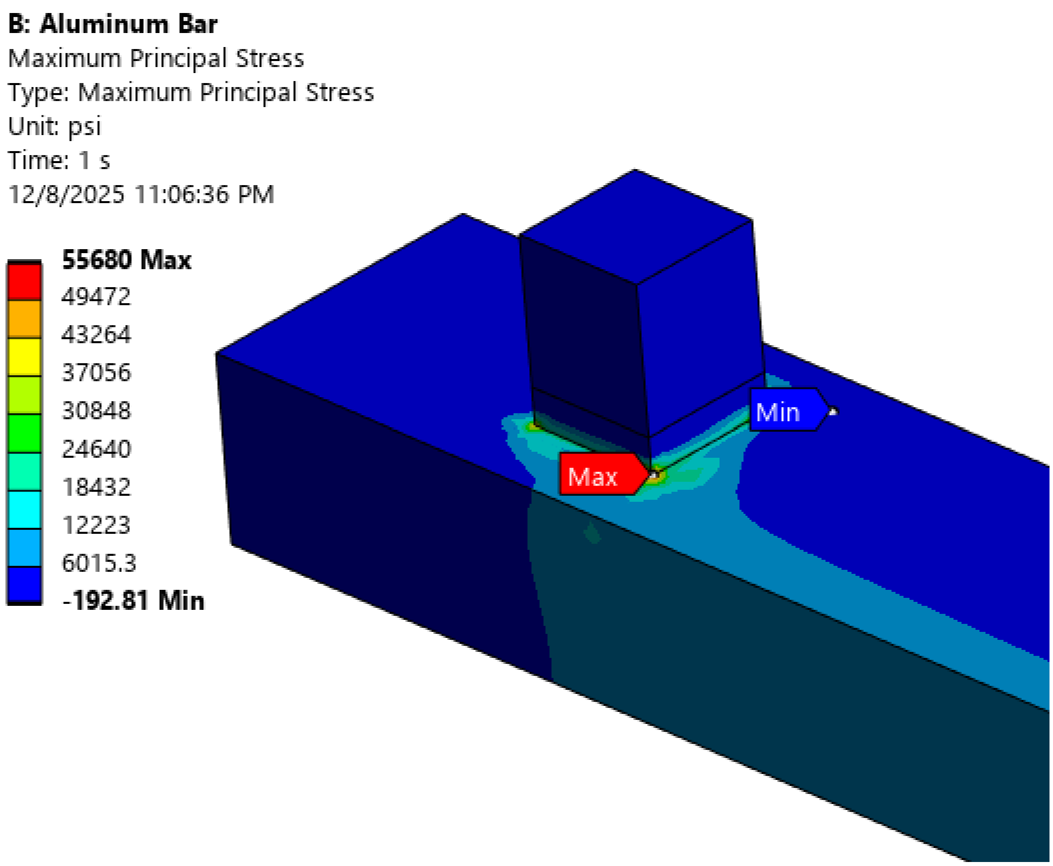
4. Normal strain contours (in the strain gauge direction) from FEM

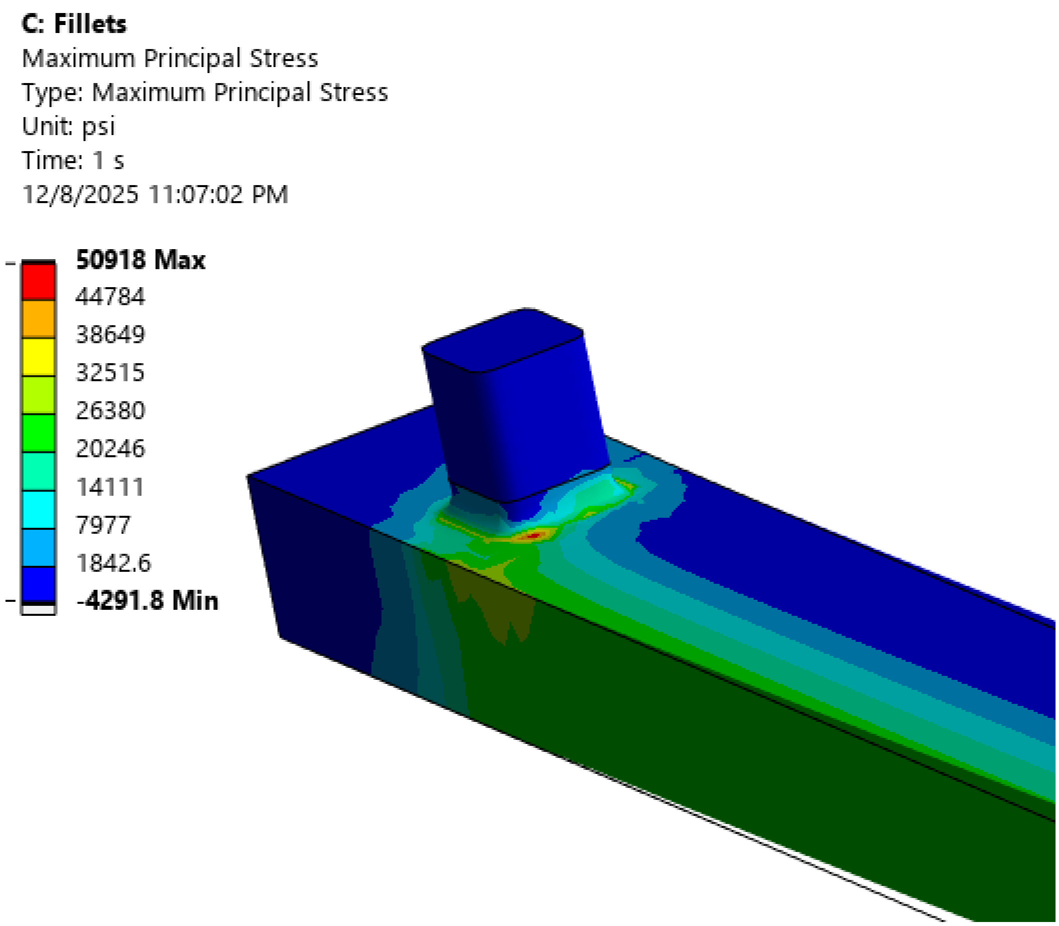



A: Static Structural
Normal Elastic Strain
Type: Normal Elastic Strain(X Axis)
Unit: in/in
Global Coordinate System
Time: 1 s
12/8/2025 9:02:39 PM
0.00092736 Max
0.0007212
0.00051503
0.00030887
0.0001027
-0.00010346
-0.00030963
-0.00051579
-0.00072196
-0.00092812 Min
Max
Min

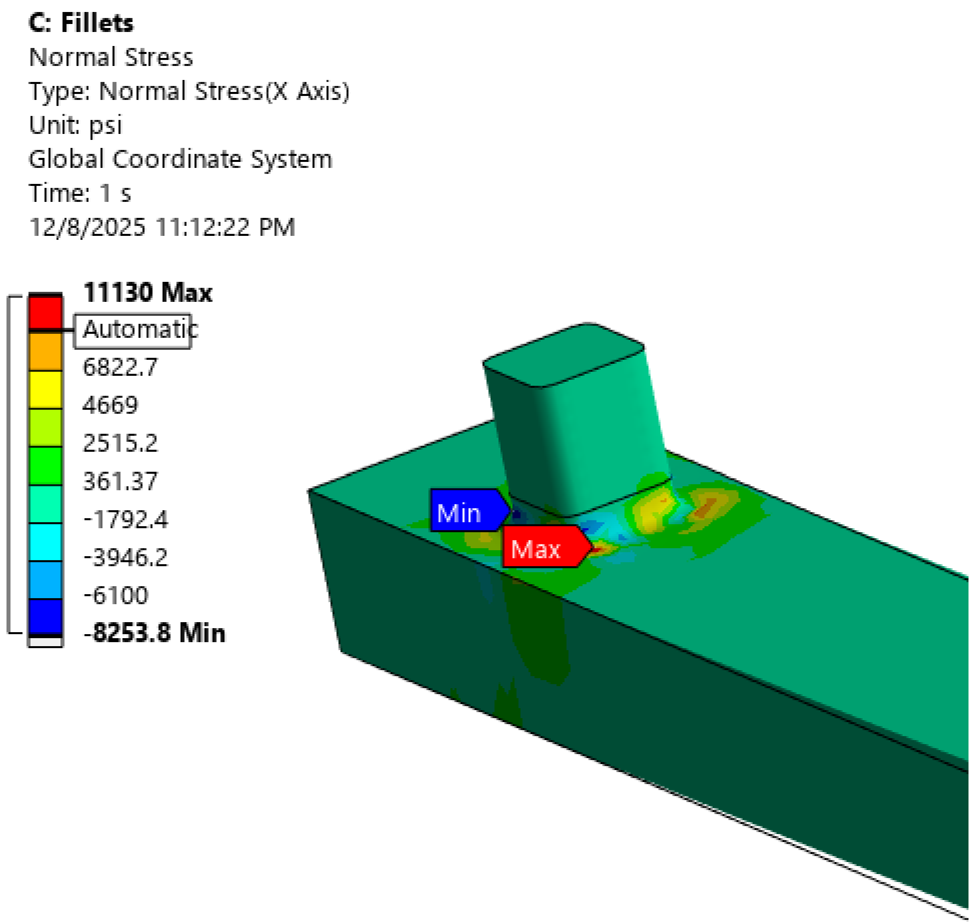


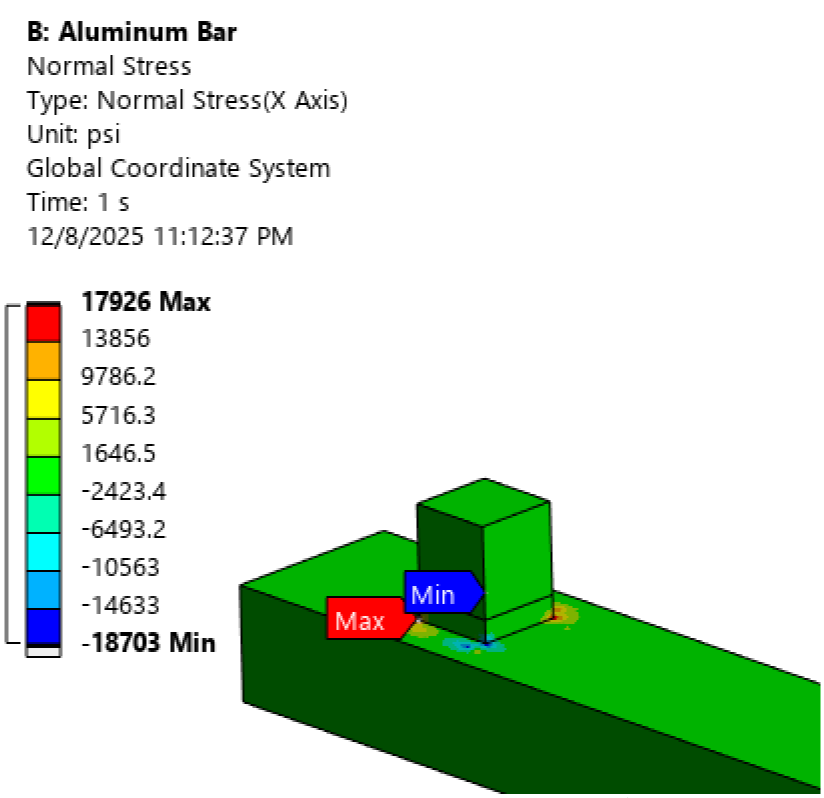
5. Contour plot of maximum principal stress from FEM

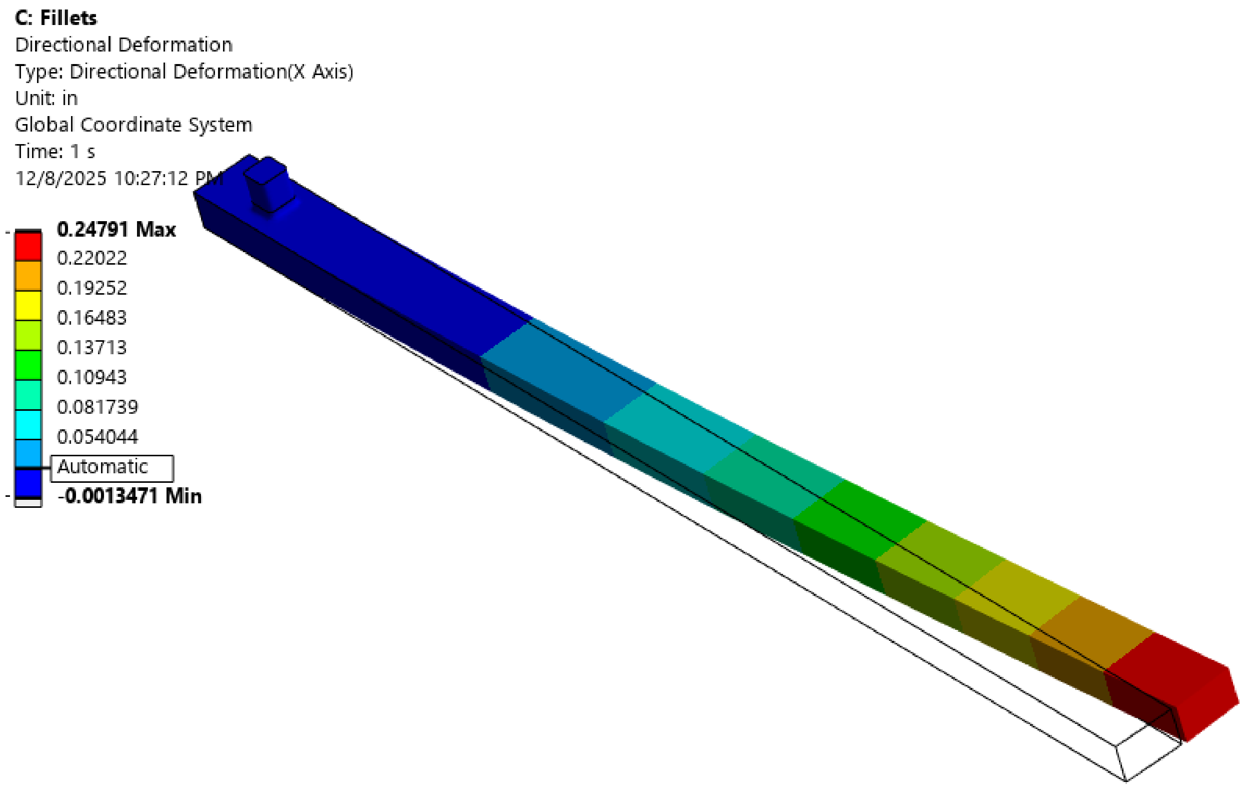

B: Aluminum Bar
Maximum Principal Stress
Type: Maximum Principal Stress
Unit: psi
Time: 1 s
12/8/2025 11:06:36 PM
55680 Max
49472
43264
37056
30848
24640
18432
12223
6015.3
-192.81 Min
Max
Min

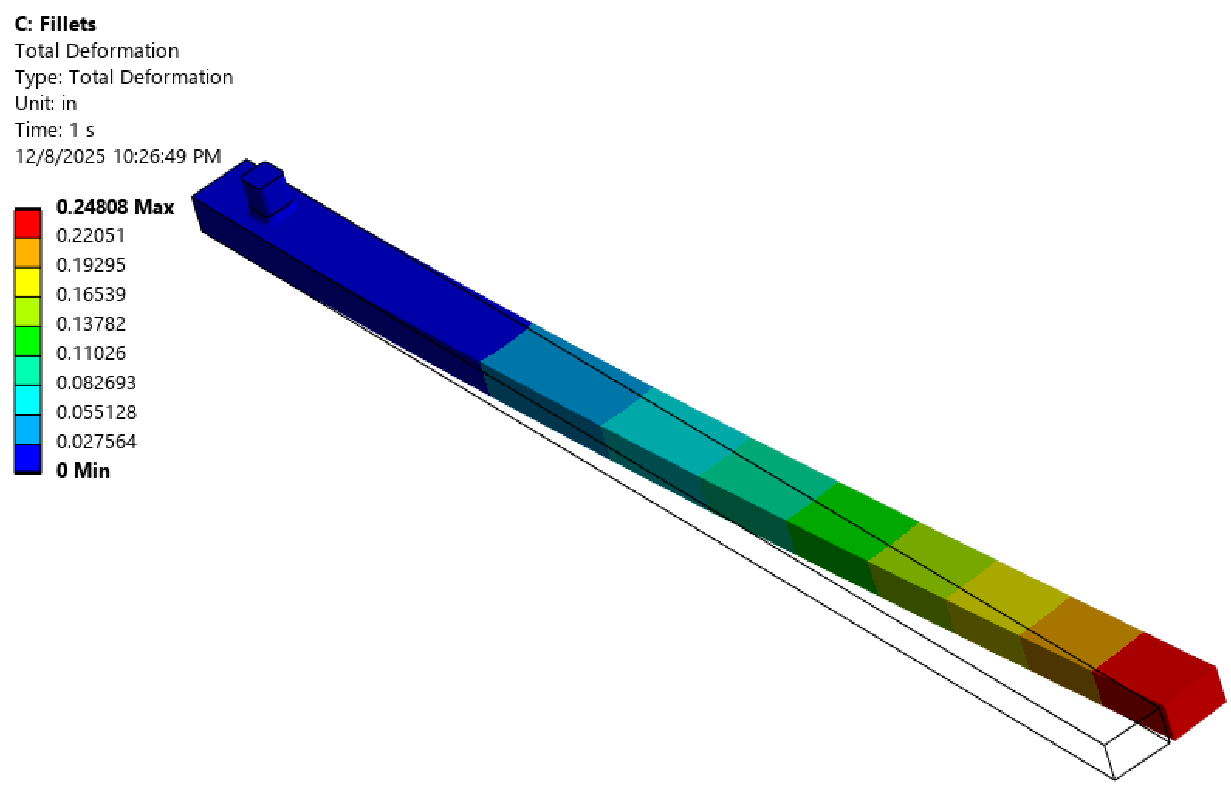


6. Summarize results from FEM calculation showing maximum normal stress (anywhere), load point deflection, strains at the strain gauge locations

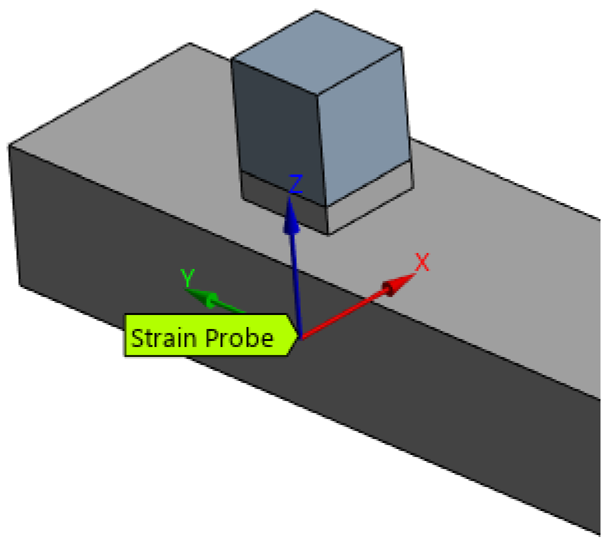

C: Fillets
Normal Stress
Type: Normal Stress(X Axis)
Unit: psi
Global Coordinate System
Time: 1 s
12/8/2025 11:12:22 PM
11130 Max
Automatic
6822.7
4669
2515.2
361.37
-1792.4
-3946.2
-6100
-8253.8 Min
Min
Max


B: Aluminum Bar
Normal Stress
Type: Normal Stress(X Axis)
Unit: psi
Global Coordinate System
Time: 1 s
12/8/2025 11:12:37 PM
17926 Max
13856
9786.2
5716.3
1646.5
-2423.4
-6493.2
-10563
-14633
-18703 Min
Min
Max


C: Fillets
Directional Deformation
Type: Directional Deformation(X Axis)
Unit: in
Global Coordinate System
Time: 1 s
12/8/2025 10:27:12 PM
0.24791 Max
0.22022
0.19252
0.16483
0.13713
0.10943
0.081739
0.054044
Automatic
-0.0013471 Min


C: Fillets
Total Deformation
Type: Total Deformation
Unit: in
Time: 1 s
12/8/2025 10:26:49 PM
0.24808 Max
0.22051
0.19295
0.16539
0.13782
0.11026
0.082693
0.055128
0.027564
0 Min

B: Static Structural
Strain Probe
12/8/2025 9:05:52 PM



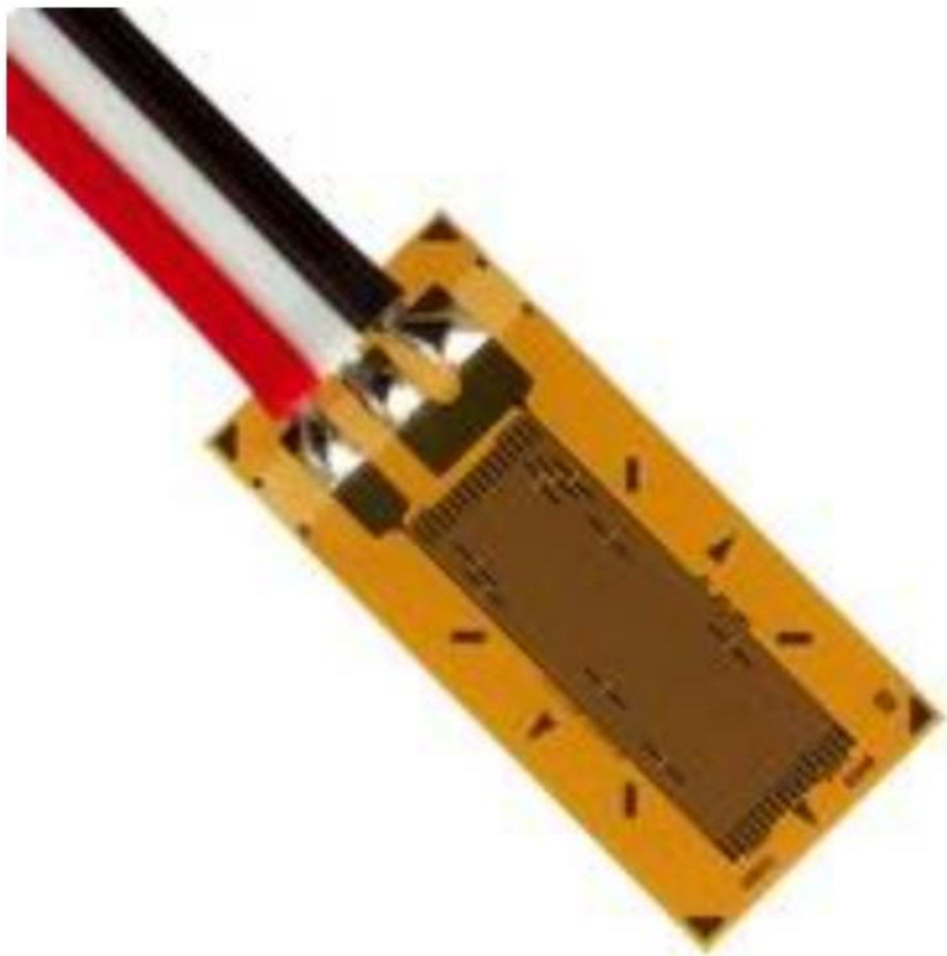
Maximum Value Over Time	
Normal - X Axis	-1.0257e-004 in/in
Normal - Y Axis	3.5183e-004 in/in
Normal - Z Axis	-1.1161e-004 in/in
XY Shear	-6.244e-007 in/in
YZ Shear	5.0588e-005 in/in
XZ Shear	-1.8257e-007 in/in
Equivalent (von-Mises)	3.5475e-004 in/in
Maximum Principal	3.5321e-004 in/in
Middle Principal	-1.0257e-004 in/in
Minimum Principal	-1.1299e-004 in/in
Intensity	4.6619e-004 in/in

Maximum Value Over Time	
Normal - X Axis	-1.3295e-004 in/in
Normal - Y Axis	2.2028e-003 in/in
Normal - Z Axis	-1.5063e-004 in/in
XY Shear	-4.4484e-005 in/in
YZ Shear	3.0609e-004 in/in
XZ Shear	-1.8882e-006 in/in
Equivalent (von-Mises)	2.7271e-003 in/in
Maximum Principal	2.215e-003 in/in
Middle Principal	-1.3381e-004 in/in
Minimum Principal	-1.5222e-004 in/in
Intensity	2.9761e-003 in/in

7. Torque wrench sensitivity in mV/V using strains from the FEM analysis

From the derivation done in class, the torque wrench sensitivity is a value that is calculated from the strain experienced at the strain gauge. This is determined by taking the strain and multiplying it the gauge factor, which was found to be 1000. With the strain at the gauge being 0.00221, the sensitivity would then be 2.21mV/V.

8. Strain gauge selected (give type and dimensions). Note that design must physically have enough space to bond the gauges.



https://www.utmel.com/productdetail/micromeasurementsdivisionofvishayprecisiongroup-mmf404146-1840402?srsltid=AfmBOoocQ4xh1xwT4nZjpHKU6IeMazIj0N58Pbe0p4IYrdau3ItUvB-6xJ8

The two main constraints that I looked at when finding a strain gauge was the physical size of it and the strain capacity. For this strain gauge, bounding dimension would be 0.18'' and with the torque wrench being 0.5'' in thickness, this sensor would fit perfectly. It also has a strain range of 3%, and with the max strain at that location measuring about 0.0005, corresponding to a 0.5% deformation, the sensor would work perfectly.

6 Notes and things to think about

- Your final design should mitigate stress concentrations to the extent you can

I added a filleted version of the torque wrench with fillets around the drive and the connection between the drive and body of the torque wrench. This filleted model compared to the original was seen to be more accurate to the values found in the matlab script.

Deflection = 0.227 in
Normal Stress = 11.25 ksi

Safety Factor For Strength = 4.4
Safety Factor For Crack Growth = 6.61
Safety Factor For Fatigue = 1.97
Strain at Gauge, = 1050 microstrain
Output, = 1.05 mV/V

- You could consider making the handle narrower at the locations of the gauges. That may give higher sensitivity but will also reduce safety factors. You will need to make some tradeoffs.

Although you could develop a more intense geometry to maximize strain when staying in range of safety factors. I decided to go for adapting the material choices and minimize the geometric changes from the original model. This is because it allows for easier manufacturing.