

Mechanics of Materials: Torque Wrench Analysis

Jonathan Distler

December 2025

1 Project Goal

The goal of this project is to develop a methodology to determine the necessary requirements for a torque wrench to meet high factors-of-safety and high response sensitivity. The system must satisfy the following requirements:

- A strain output of 1 mV/V
- A factor-of-safety against yield of at least 4
- A factor-of-safety against fatigue of at least 1.5
- A factor-of-safety against crack growth of at least 2

2 Process

The pipeline for the work was as follows. First, I screened multiple materials (Aluminum, Steel, and Titanium) using Granta's filters to identify a semi-cheap material capable of meeting the necessary performance requirements (Figures 1–3). Then, I created a MATLAB script that parsed different torque wrench dimensions and materials (along with their mechanical properties) to find a feasible design. Finally, I created a parameterized torque wrench model in Fusion 360, allowing for iterative design improvements if needed.

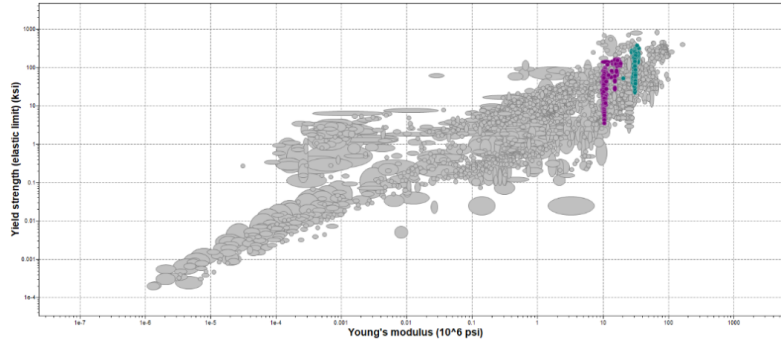


Figure 1: Young's Modulus vs. Yield Strength

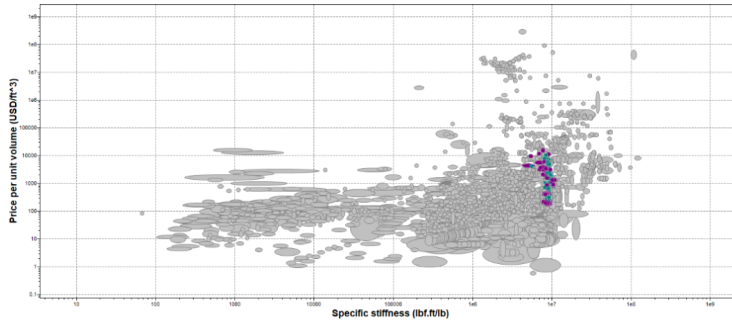


Figure 2: Stiffness vs. Price

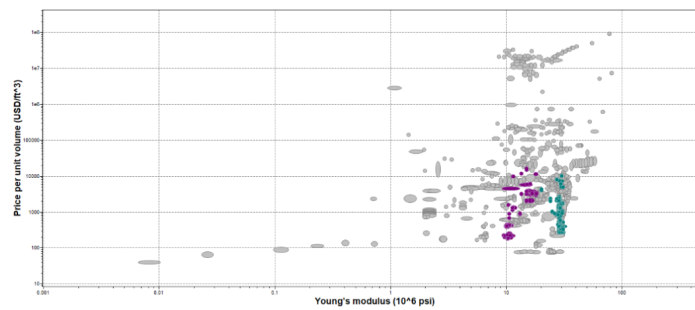


Figure 3: Young's Modulus vs. Price

The following is the MATLAB code used to iterate over candidate designs and materials in order to identify configurations that satisfy all requirements.

```

% Script for Mechanics of Materials Final Homework
% Goal is to define all of the variables
% and equations for a torque wrench with a pure moment applied
% Then, I plan on iterating over dimensions to see which dimensions
% Still provide viable results for our predefined factors of safety

% double check which variables can be iterated over
% need a way to define if a material is ductile or brittle
% can search for a steel, aluminum, or titanium alloy
% How to get the 1.0mV/V for 600 in lbf input? What equation is that?
% need to use half-bridge, could ask TA at OH
% maybe do von mises for stress

% main goals, need to know how to get strain (which equation?)
% also, need to double check the equation for fatigue stress, as he didn't
% define any of the necessary variables
% remake centered list

% Variable arrays for iteration
Ms = [600]; % applied moments (lbf-in)
Ls = [12, 15, 18, 9, 21, 7, 24, 13, 16, 19, 26]; % lengths (in)
hs = [1.2, 1.5, 1.8, .9, 1.8, 2.1, .7, .5]; % heights (in)
bs = [1.5, 1.8, 2.0, 1.2, .9, 2.1, .6, .5, 2.4]; % thickness (in)
cs = [0.0003, 0.0005, 0.0007, .0001]; % gauge offsets (in)

% Material property arrays (unchanged)
Es =
[32e6, 29.4e6, 10.2e6, 31.9e6, 10e6, 29e6, 14.5e6, 26.5e6, 16.4e6, 15.2e6, 29.4e6, 29.9e
6, 26.5e6, 28.4e6, 28.4e6];
nus =
[0.29, 0.29, 0.29, 0.29, 0.29, 0.29, 0.29, 0.29, 0.29, 0.29, 0.29, 0.29, 0.29];
sus =
[370e3, 160e3, 29e3, 396e3, 7.98e3, 39.2e3, 50e3, 245e3, 174e3, 100e3, 281e3, 327e3, 271e
3, 188e3, 147e3];
KICs =
[115e3, 31e3, 15.3e3, 14.5e3, 29.1e3, 17.3e3, 62.8e3, 118e3, 72.8e3, 45.5e3, 51e3, 20.7e3
, 118e3, 103e3, 122e3];
sfatigue=
[115e3, 38.3e3, 3.83e3, 110e3, 3.84e3, 15.1e3, 35.5e3, 118e3, 93.7e3, 59.2e3, 98.8e3, 79
.3e3, 118e3, 66e3, 62.8e3];
names = ["M42 Steel", "Tool steel, low carbon, AISI P21 (mold)", "Aluminum,
A390.0, permanent mold cast, F", "Tool steel, molybdenum alloy, AISI M46
(high speed)", "Aluminum, commercial purity, 1-0", "Drawing quality, YS140,
cold rolled", "Titanium, alpha alloy, Ti-0.2Pd (grade 7)", "Maraging steel,
250, maraged at 482°C", "Titanium, alpha-beta alloy, Ti-6Al-2Sn-4Zr-6Mo
(6-2-4-6)", "Titanium, commercial purity, Grade 4", "Low alloy steel, AISI
9260, oil quenched & tempered at 425°C", "Tool steel, AISI A9 (air-hardening
cold work)", "Maraging steel, 250, maraged at 482°C", "Low alloy steel, SAE
4335M, cast, quenched & tempered", "Low alloy steel, SAE 8630, cast, quenched
& tempered"];

```

Figure 4: MATLAB Code Sample

```

mattypes =
[ 'ductile', 'ductile', 'brittle', 'ductile', 'brittle', 'ductile', 'brittle', 'ductile', 'brittle', 'ductile', 'ductile', 'ductile', 'ductile', 'ductile', 'ductile' ];

function bool=torque_calcs(M,L,h,b,c,E,nu,su,KIC,sfatigue,name,material)
% Requirements
strain_output_goal = .001; % V/V goal
fos_yield_goal = 4;
fos_fatigue_goal = 1.5;
fos_crack_growth_goal = 2;

F = M/L; % point force equivalent
I = (b*h^3)/12; % moment of inertia

% Max normal stress in psi
sigma = (M*(h/2))/I;
fos_yield = su/sigma;

% Deflection
u = (F*L^3)/(3*E*I);

% Fatigue FoS
fos_fatigue = sfatigue/sigma;

% Crack growth, f=1.12
f = 1.12;
a = .04;
Kapp = f*sigma*sqrt(pi*a);
fos_crack_growth = KIC/Kapp;

% Strain at gauge
M_gauge = F*(L-c);
sigma_gauge = (M_gauge*(h/2))/I;
strain_gauge = sigma_gauge/E;
strain_output = strain_gauge;

if (strain_output>strain_output_goal && fos_yield>fos_yield_goal &&
fos_fatigue>fos_fatigue_goal && fos_crack_growth>fos_crack_growth_goal)
    bool = true;
    disp('Name')
    disp(name)
    disp('Material Type')
    disp(material)
    disp('Moment, Length, h, b, c')
    disp([M L h b c]) %Indices of the respective arrays for which the
output is true
else
    bool = false;
end

```

Figure 5: MATLAB Code Sample

```

end

% Nested loops for variable iteration
for i = 1:length(Es)
    for M = Ms
        for L = Ls
            for h = hs
                for b = bs
                    for c = cs
                        torque_calcs(M,L,h,b,c,Es(i),nu(s(i)),sus(i),KICs(i),sfatigues(i),names(i),mattypes(i));
                    end
                end
            end
        end
    end
end
end
end

```

Figure 6: MATLAB Code Sample

```

Name
Maraging steel, 250, maraged at 482°C
Material Type
ductile
Moment, Length, h, b, c
600.0000 12.0000 0.5000 0.5000 0.0003

Name
Maraging steel, 250, maraged at 482°C
Material Type
ductile
Moment, Length, h, b, c
600.0000 12.0000 0.5000 0.5000 0.0005

Name
Maraging steel, 250, maraged at 482°C
Material Type
ductile
Moment, Length, h, b, c
600.0000 12.0000 0.5000 0.5000 0.0007

```

Figure 7: MATLAB Code Output

Using the MATLAB-generated dimensions, I created a full torque wrench CAD model in Fusion-360. The dimensions are listed below: length of 12”, width of .5”, and height of .5”.

```

Name
Maraging steel, 250, maraged at 482°C
Material Type
ductile
Moment, Length, h, b, c
600.0000 12.0000 0.5000 0.5000 0.3000

FOS-Yield, U, FOS-Fatigue, Kapp, FOS-Crack Growth, Strain Output
1.0e+04 *
0.0009 0.0000 0.0004 1.1434 0.0010 0.0000

```

Figure 8: Used Dimensions and Factors of Safety

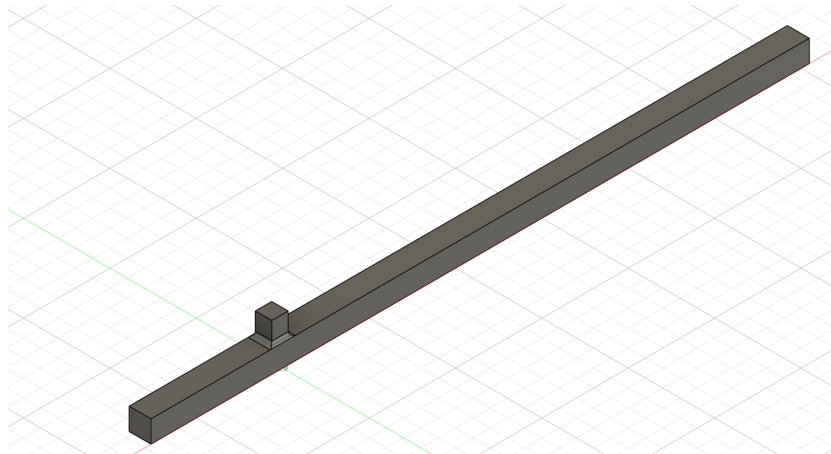


Figure 9: Torque Wrench CAD (Orthographic View)

Then, I followed a tutorial to save the CAD as a .STEP file and then implemented it in ANSYS. I constrained the top part of the drive and applied a

distributed load of force (equivalent to the applied moment of 600 lb-in) on the front face of the torque wrench. Then, I gathered the following outputs:

- Maximum Normal Stress = 1.1×10^6 psi
- Strain at the Strain Gauge = 3.277×10^{-2} m/m or 3.28 mV/V
- Deflection of the Load Point = .039 in

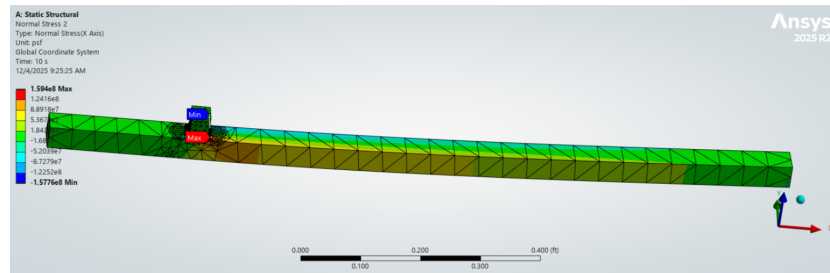


Figure 10: Maximum Normal Stress

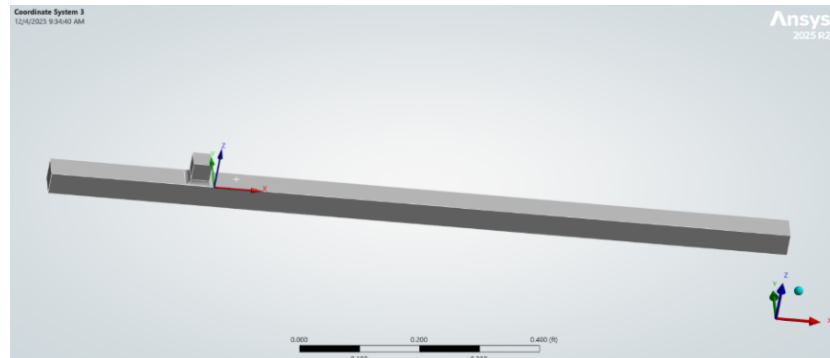


Figure 11: Coordinate System for Strain Probe of Strain Gauge

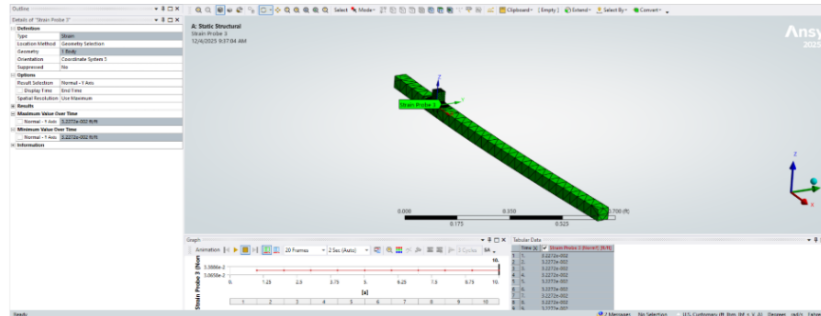


Figure 12: Strain Probe of Strain Gauge

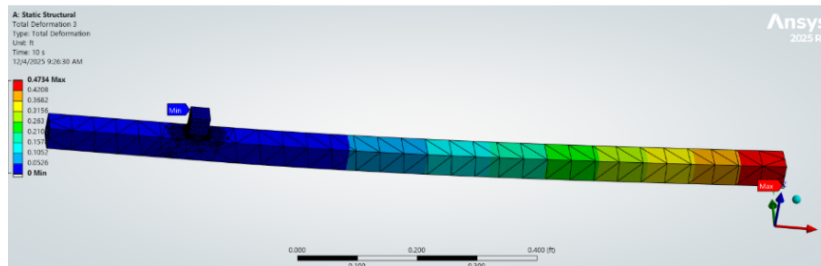


Figure 13: Deformation of Point Application

3 Reflections

- **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?** There is clear curvature to the lines, however, they do maintain a general straightness, and for the most part can be approximated as being parallel and roughly straight. Therefore, I would argue that beam theory is a reasonably accurate model, as all of lines remain approximately straight and parallel, therefore the section (to a first-degree approximation) is plane.
- **How do the FEM and hand calculated maximum normal stresses compare? If they differ significantly, why?** The calculated stress was 1.1×10^6 psi, whereas the calculated maximum stress was 28800 psi. This is an obvious discrepancy that arises due to the stress concentrations from the top part of the drive. The hand-calculations didn't factor these stress concentrations into the beam-bending equations. The normal stress approaches a singularity around the stress concentration which explains the high normal stress. Although it was a secondary goal to minimize the stress concentration (as per the goal of the fillets), this was minimized via

mesh changes around the extrusion (although tighter meshes around the extrusion typically result in higher normal stresses).

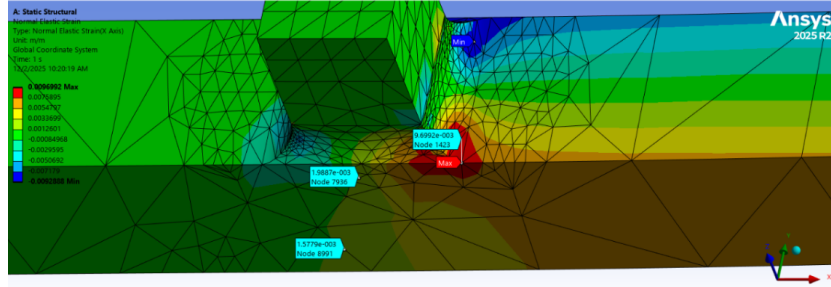


Figure 14: Stress Concentration

- **How do the FEM and hand calculated displacements compare? If they differ, why?** The calculated displacement was .2" whereas the FEM displacement (taken as the maximum displacement) was .039". This is obviously a huge difference and this could be a result of needing a finer mesh, not applying a true moment as was the case in the system setup, or again, the geometry of the top part of the drive which was shown earlier to have significantly increased the maximum stress, and this could drive the deformation to be much smaller at the tip.

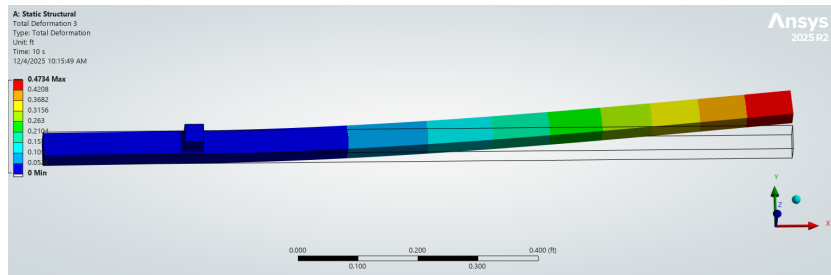


Figure 15: Deformation

4 Results

- **CAD Model With Dimensions** The following are the parametric dimensions of my design (as Fusion only limits 2 measurements at a time), as well as the orthographic picture of my design:

User Parameters					
☆ User Parameter	L	in	12 in	12.00	Length of torque wrench
☆ User Parameter	b	in	0.5 in	0.50	Height of torque wrench
☆ User Parameter	h	in	0.5 in	0.50	Thickness of torque wrench
☆ User Parameter	h2	in	0.5 in	0.50	Height of the extrusion
☆ User Parameter	revolve	in	0.1 in	0.10	Height of revolution around h2
☆ User Parameter	w	in	(3 / 8) * 1 in	0.375	Width of the extrusion

Figure 16: Parameters for Torque Wrench CAD

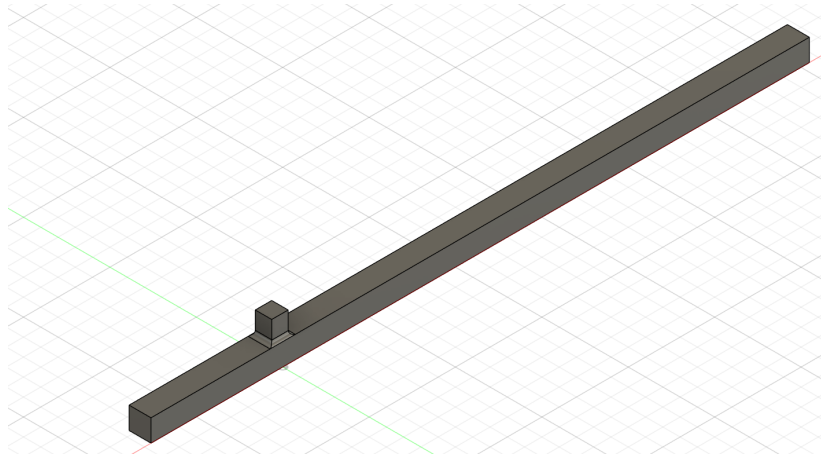


Figure 17: Torque Wrench CAD (Orthographic View)

- **Describe material used and its relevant mechanical properties.** I decided on using Maraging steel, 250, maraged at 482°C. Its relevant mechanical properties are as follows: $E=26.5e6$ psi, Tensile Strength=271 ksi, Poisson's Ratio=.31, Fatigue Strength=118 ksi (per 10^7 cycles), Fracture Toughness=118 ksi*in⁵.
- **Diagram communicating how loads and boundary conditions were applied to your FEM model.** The diagram below shows how I applied the loads and boundary conditions. The yellow-highlighted area represents the top part of the drive. In ANSYS, I constrained this such that it was static, meaning that it was incapable of moving, this was done via a face selection of the five highlighted faces. I made sure to take special care not to select the fillet and below. Then, I applied a distributed force-load across the loaded-face. I converted the applied moment to the relevant applied force for the given distance of the width, and then applied the force as a distributed force, as was recommended by Dr. Bhaskaran.

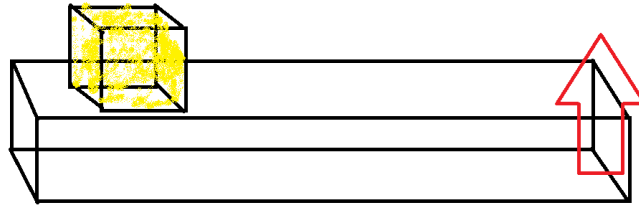


Figure 18: ANSYS Constraints Model

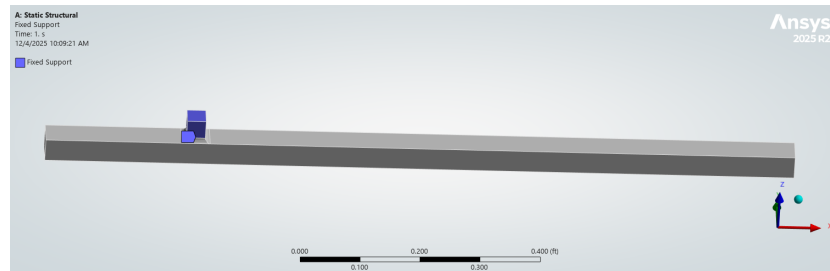


Figure 19: ANSYS Fixed Support

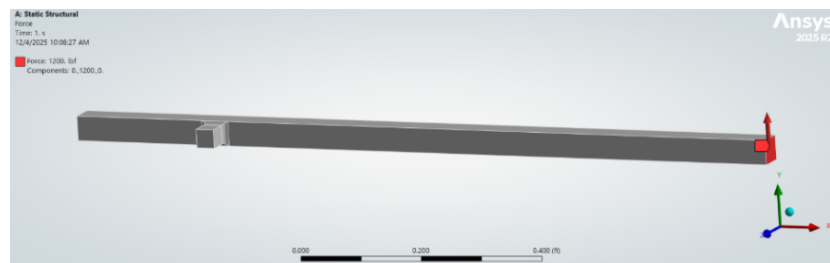


Figure 20: ANSYS Applied Moment

- **Normal strain contours (in the strain gauge direction) from FEM.** The following is the normal strain contours in the frame of the strain-gauge, as defined by a non-global coordinate system:

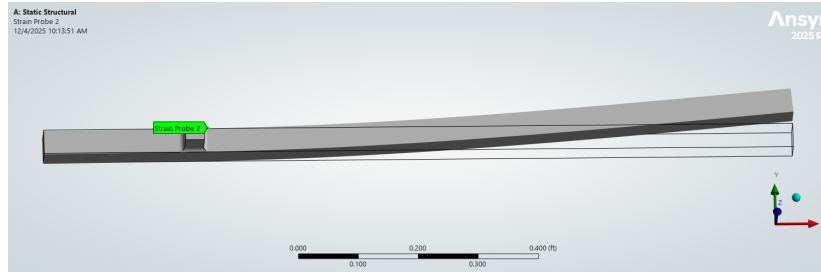


Figure 21: Strain Gauge's Frame

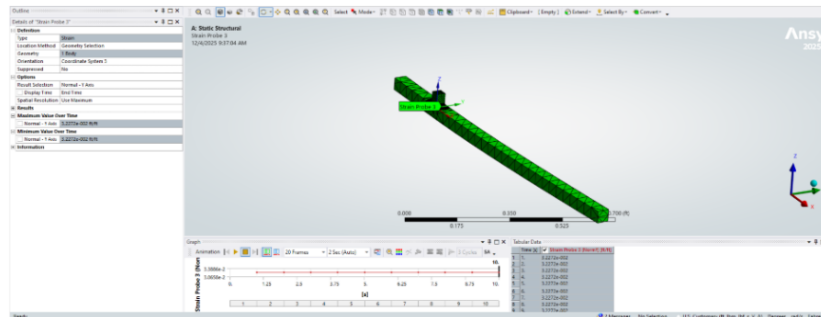


Figure 22: Normal Strain in the Strain Gauge's Frame

- Contour plot of maximum principal stress from FEM

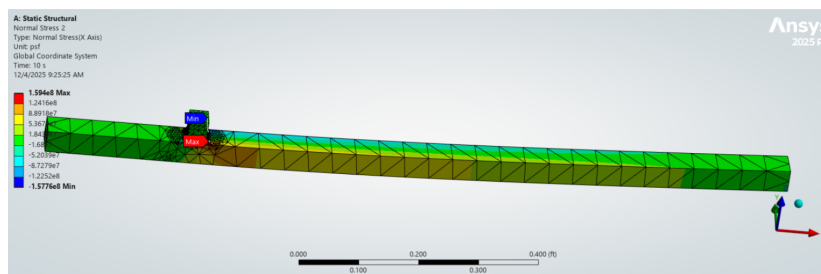


Figure 23: Maximum Principal Stresses

- Summarize results from FEM calculation showing maximum normal stress (anywhere), load point deflection, strains at the strain gauge locations As discussed underneath the reflections section, the following were the FEM outputs:
 - Maximum Normal Stress = 1.1×10^6 psi
 - Strain at the Strain Gauge = 3.277×10^{-2} m/m or 3.28 mV/V

– Deflection of the Load Point = .039 in

Overall, these were on the same order of magnitude predicted by my hand-calculations.

- **Torque wrench sensitivity in mV/V using strains from the FEM analysis** The resulting strain in mV/V was 3.28 which was higher than the threshold of 1.0 defined in the problem-statement, thus meeting the threshold. This value was much higher than the threshold likely as a result of the stress concentrations around the extrusion, which is where the strain gauge was located.

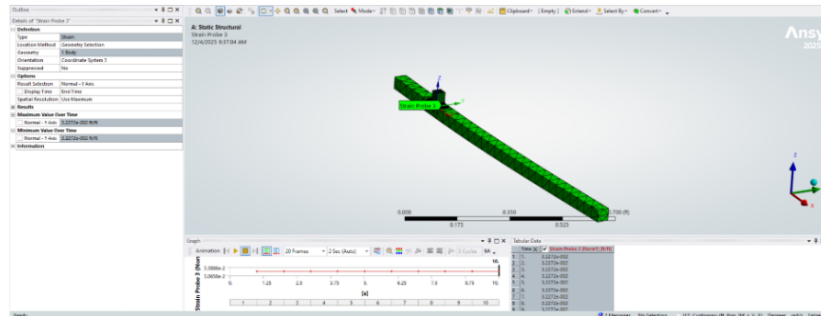


Figure 24: Strain Probe at Strain-Gauge Application Point (3.28 mV/V)

- **Strain gauge selected (give type and dimensions). Note that design must physically have enough space to bond the gauges** It was suggested to use a half-bridge Strain-Gauge by Dr. Zehnder. The following was the Strain-Gauge I decided on, BF350-2HA-C Half-Bridge Strain Gauge (<https://www.aliexpress.us/item/3256803563592738.html>). Even though the tolerance of location (.3") is small, all that matters is that the resistance bridge of the strain gauge is at the correct location; its sensitive grid dimensions (the actual resistor) area is 2mm x 3.7mm and the overall size is 9mm x 5.6 mm. The idea is that as long as the sensitive grid dimensions are less than the distance between the strain gauge and the top of boundary-application point, then it is feasible. .3" is less than 2mm and 3.7mm both, thus this could accurately fit in the intended location and not interfere with the boundary condition. However, there may be an overhang of wires parallel to the wrench. One possible issue could be that the resistance might not be large enough to support a viable voltage difference across the strain gauges.