MAE 3270 Final Project Part I: Material Selection and FEA

Matthew Sonn; Matthew Bulger

December 6, 2025

Script Used for Hand Calculations:

```
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
% Find Load Point Deflection and Max Normal Stress:
F = M/L;
r = h/2;
I = (b*h^3)/12;
loadpointdeflection = (M*L^2)/(3*E*I)
maxnormalstress = M*r/I
% Find Safety Factors:
a = 0.04;
x = a/h;
Y = 1.12;
Kmax = Y*maxnormalstress*sqrt(pi*a);
sf_strength = su/maxnormalstress
sf_crackgrowth = KIC/Kmax
sf_fatigue = sfatigue/maxnormalstress
%Find Strain Gauge Results:
bendingstress = F*(L-c)*r/I;
strainatgauge = bendingstress/E
output = strainatgauge*1e3 %mV/V
```

Hand Calculation Results (Screenshot from MATLAB code that I wrote):

```
loadpointdeflection = 0.0910

maxnormalstress = 12800
```

```
sf_strength = 28.9062

sf_crackgrowth = 2.9516

sf_fatigue = 8.9844
```

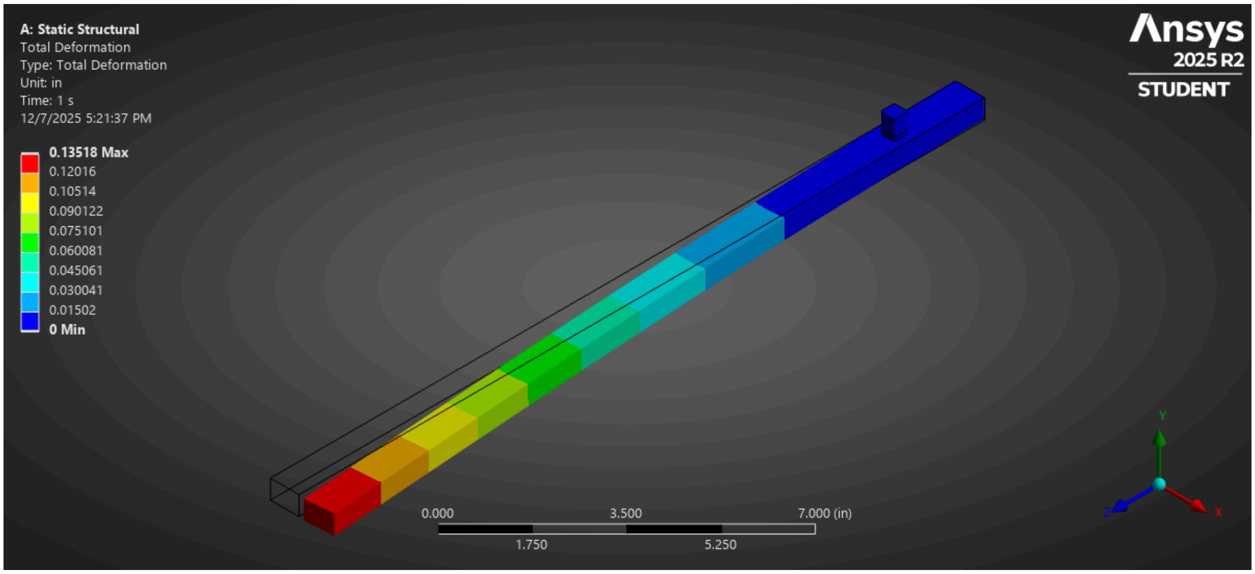
```
strainatgauge = 3.7500e-04

output = 0.3750
```

This matches the result that is given in the final project document.

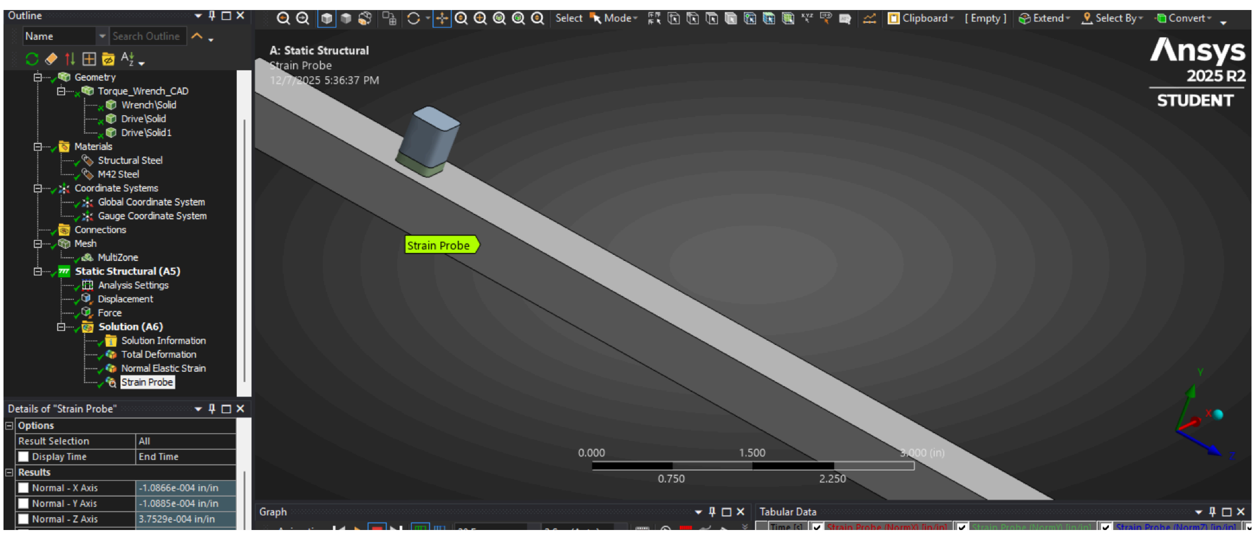
FEM Results:

Deflection of the load point:



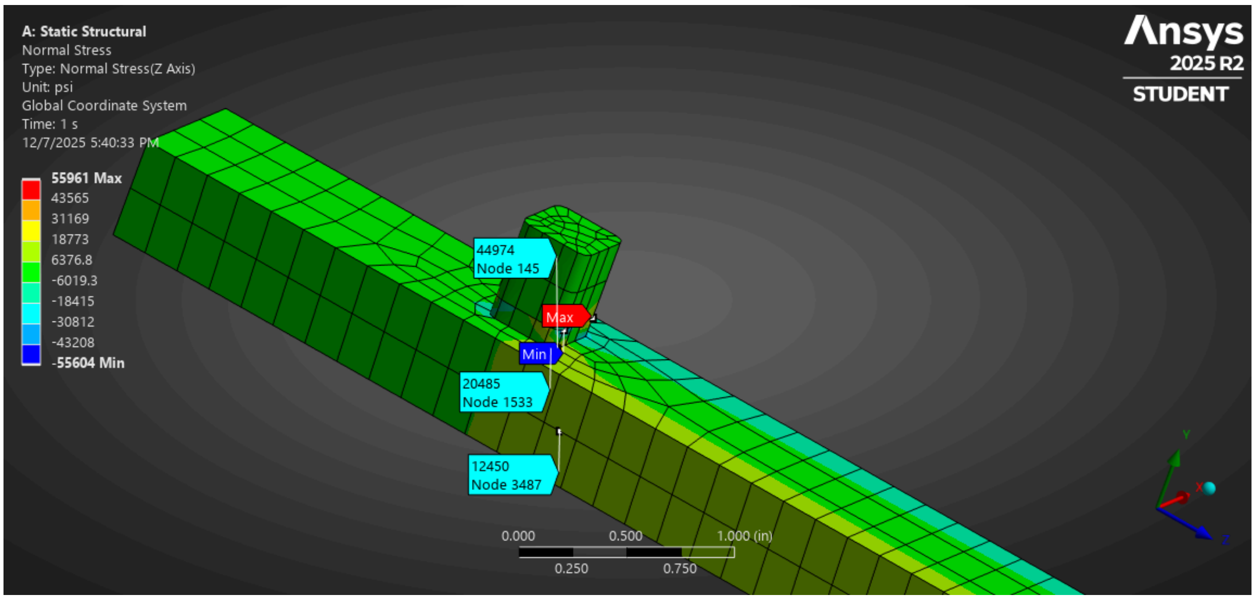
As can be seen in the screenshot above, the deflection of the load point calculated by Ansys is 0.13518 in. The hand calculations produced a value of 0.0910 in.

Strain at Strain Gauge Location:



As can be seen in the bottom left hand corner of the screenshot, the normal strain in the z direction, which would be the strain measured by the strain gauge, is 375 microstrain. This is exactly the same value that was calculated in the hand calculations.

<u>Maximum stress:</u>

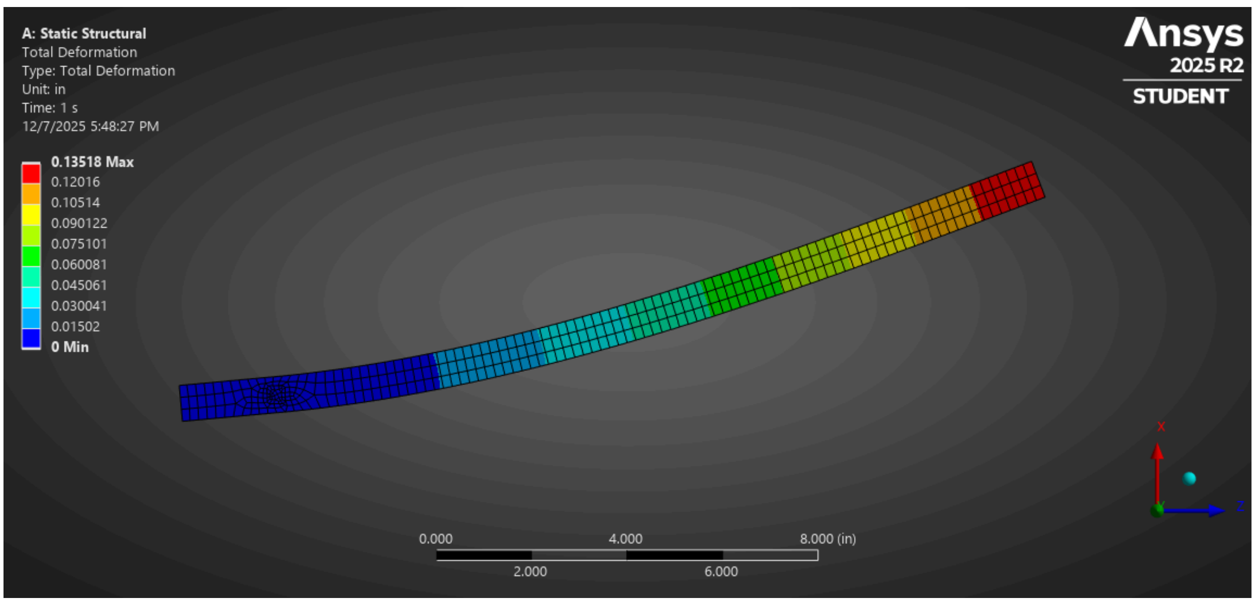


We see a maximum normal stress of around 55.961 ksi (from the legend on the right hand side) at the location that the clamped section of the drive meets the free section. We expect high stress concentrations here due to the sharp corners, however, our hand calculations did not take these stress concentrations into account. We can see that the stress on the x face is 12.45 ksi, which is very close to the calculated value of 12.8 ksi from the hand calculations.

<u>Reflection Questions:</u>

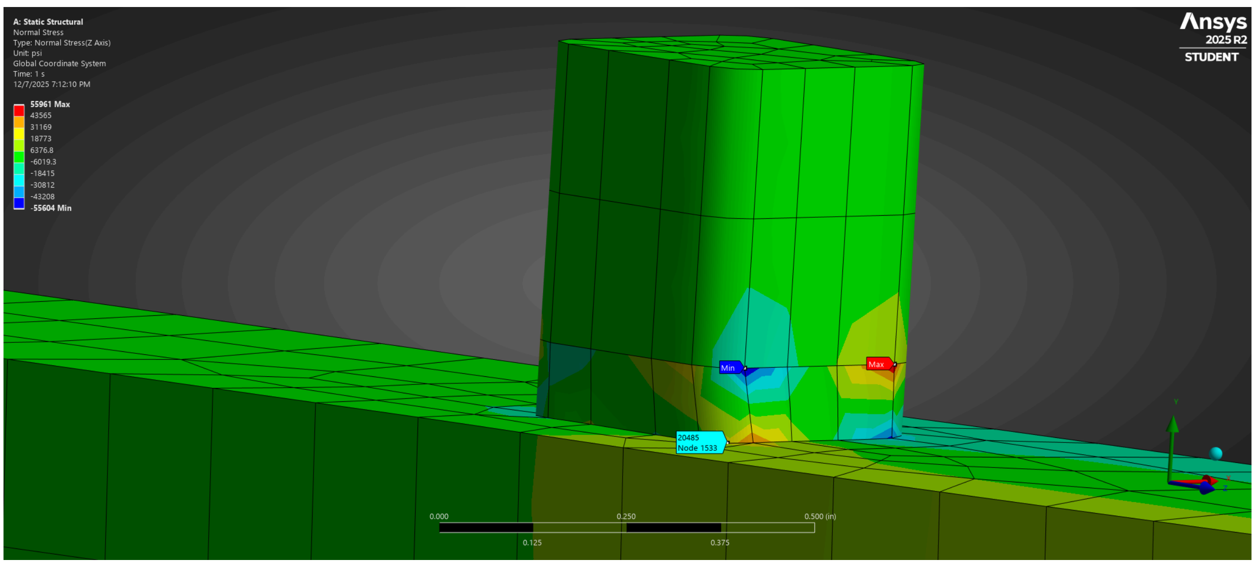
<u>Validity of plane sections remaining plane:</u>

Looking at the deformed mesh, we can see that straight lines in the mesh remain straight, and the mesh lines that were orthogonal before deformation remain orthogonal when deformed. This means that beam theory <u>is</u> reasonably accurate for this model. Not only can we qualitatively see this in the model, but we also see it quantitatively from the similar values obtained from the FEM and from using beam theory for hand calculations.



<u>Comparison of the FEM and Hand Calculations for Maximum Normal Stress:</u>

We saw in the previous section that the maximum normal stress calculated from the MATLAB script (12.8 ksi) was very different from the maximum normal stress that we actually saw in the FEM (55.96 ksi). This is because the maximum stress that we calculated by hand came from the simple Mr/I equation, which assumes that the maximum normal stress occurs along the x face of the torque wrench, and does not take stress concentrations into account. The maximum value of the normal stress along the x face is around 12.45 ksi from the FEM, which is very similar to the value of 12.8 ksi that we calculated by hand (only a 2.77% difference). However, we see that our highest values of normal stress come from the clamped boundary conditions in the upper 0.4 inches of the drive (see picture below), and this is almost 4 times the calculated value of maximum normal stress on the x face. We didn't account for this in our hand calculations, so that is why our values differ significantly.



<u>Comparison of the FEM and Hand Calculations for Displacements:</u>

Our hand calculated maximum deformation occurred at the loading point, as was confirmed in the FEM. By hand, we calculated a deformation of 0.0910 in, and the FEM gave us a value of 0.13518 in. This is a percent difference of 39%, which is a significant difference. One reason for this issue could be the fillets on the drive. When an FEM is made for the same geometry without fillets, a value of 0.101 in. is calculated as the maximum deformation, which is significantly closer to the value calculated using the hand calculations (from Prof. Bhaskaran's slides). The fillets could be causing an issue with the mesh, resulting in a slightly miscalculated maximum deformation.

On the other hand, the deformation measured using the strain calculation in the FEM at the strain gauge location (15 in. from the end of the arm) matched the hand calculations exactly. A strain of 375 microstrain was calculated at the strain gauge location both by hand and in the FEM model.

NEW DESIGN RESULTS:

```
loadpointdeflection = 0.3348
maxnormalstress = 2.9586e+04
```

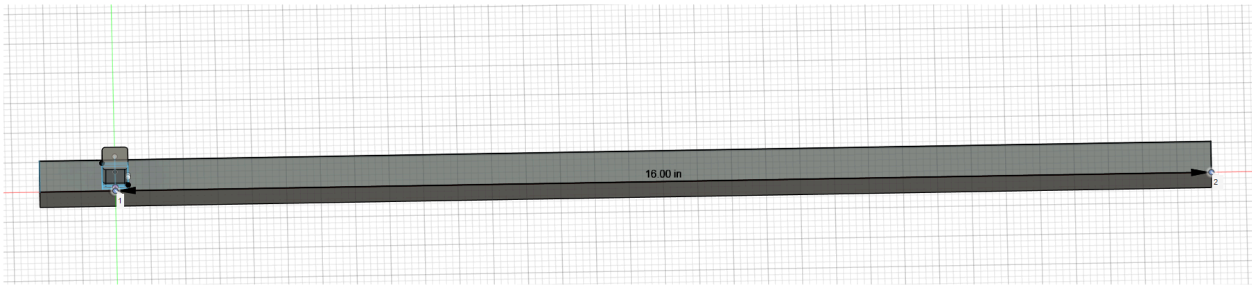
```
sf_strength = 4.3264
sf_crackgrowth = 2.0943
sf_fatigue = 2.3390
```

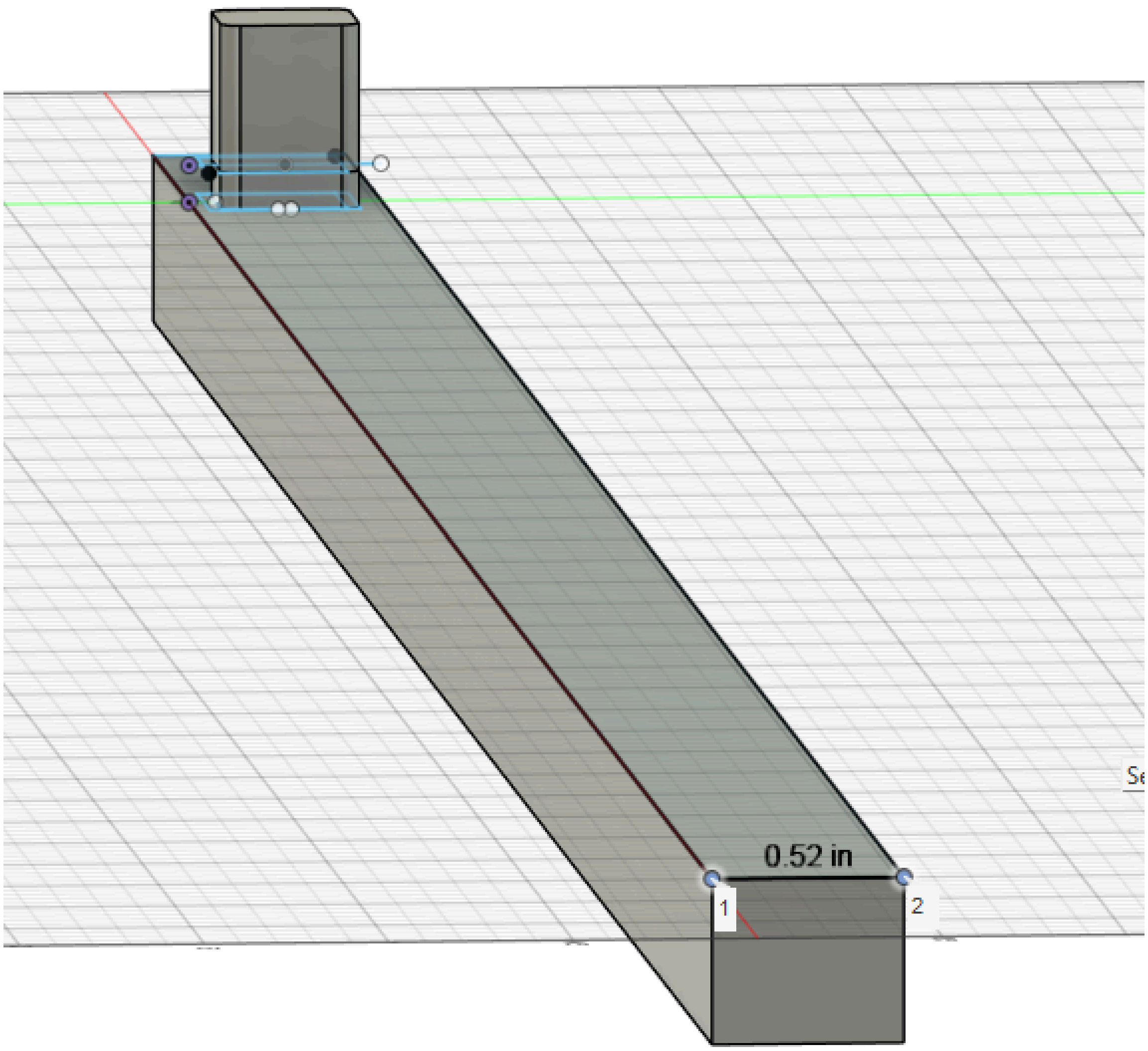
```
strainatgauge = 0.0010
output = 1.0043
```

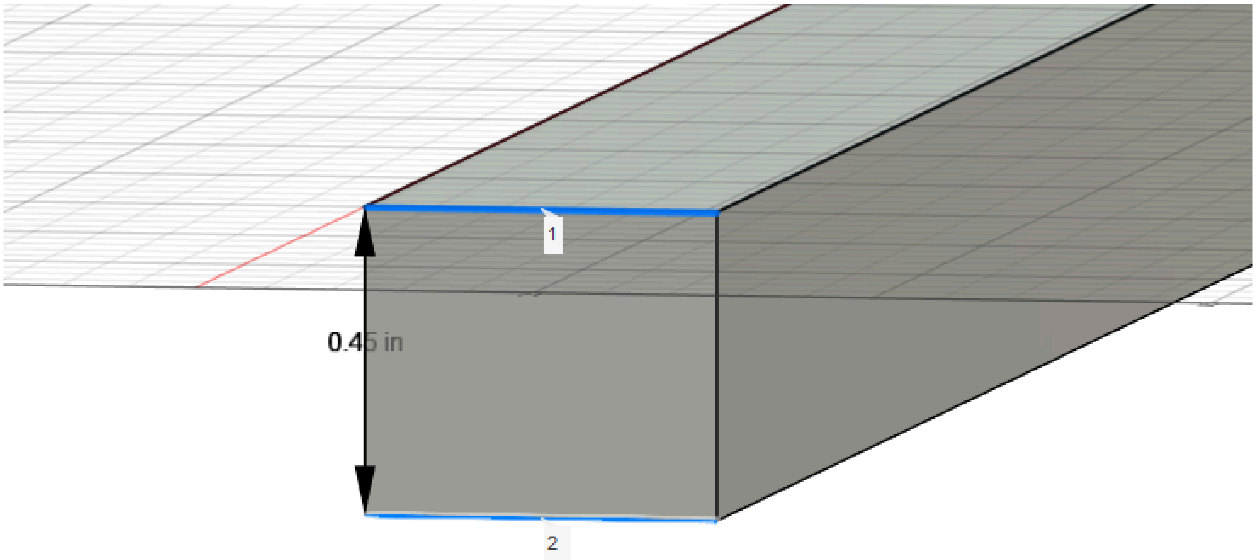
Dimensions of new design**Working material and dimensions (for our new and updated design just here for purposes):**

Material: Carbon steel, AISI 1080, oil quenched & tempered at 315 degrees Celsius

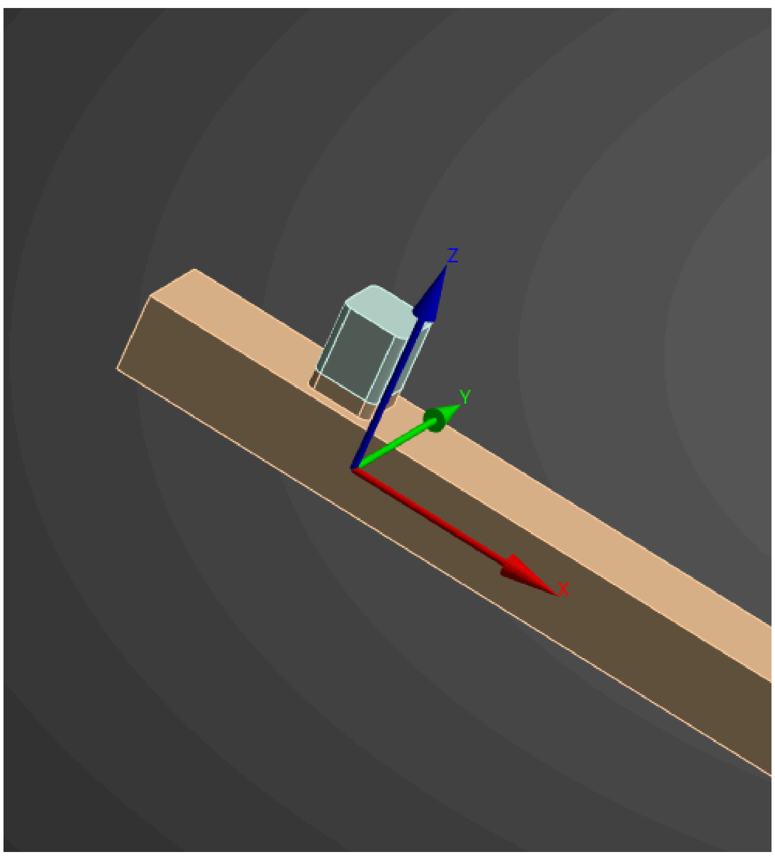
Dimensions: L = 16 in; h = 0.52 in; b = 0.45 in; c = 0.25 in


16.00 in


0.52 in
1
2



Guage Coordinates - C = .25



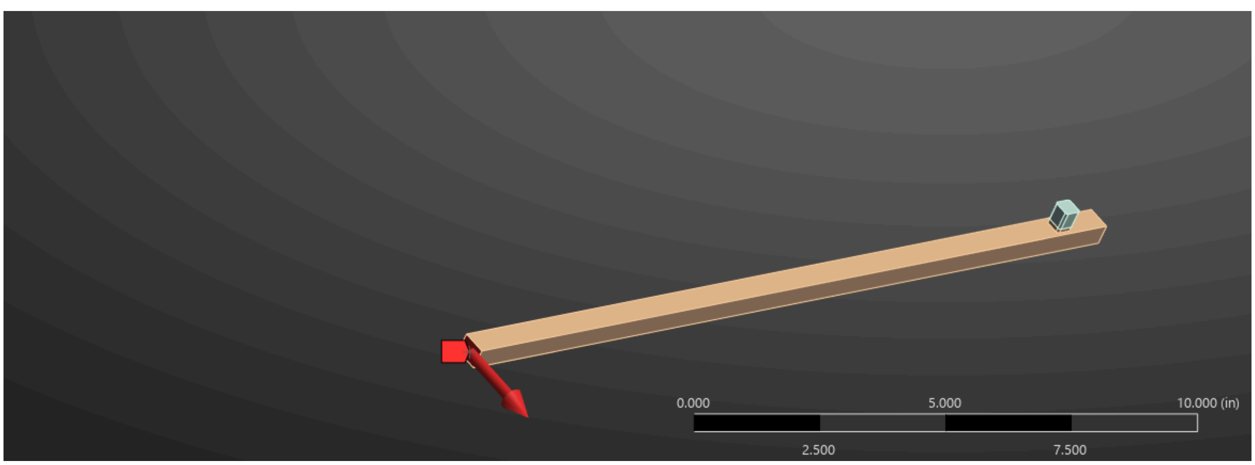
Describe the Material

Carbon steel, AISI 1080, oil quenched & tempered at 315 degrees Celsius

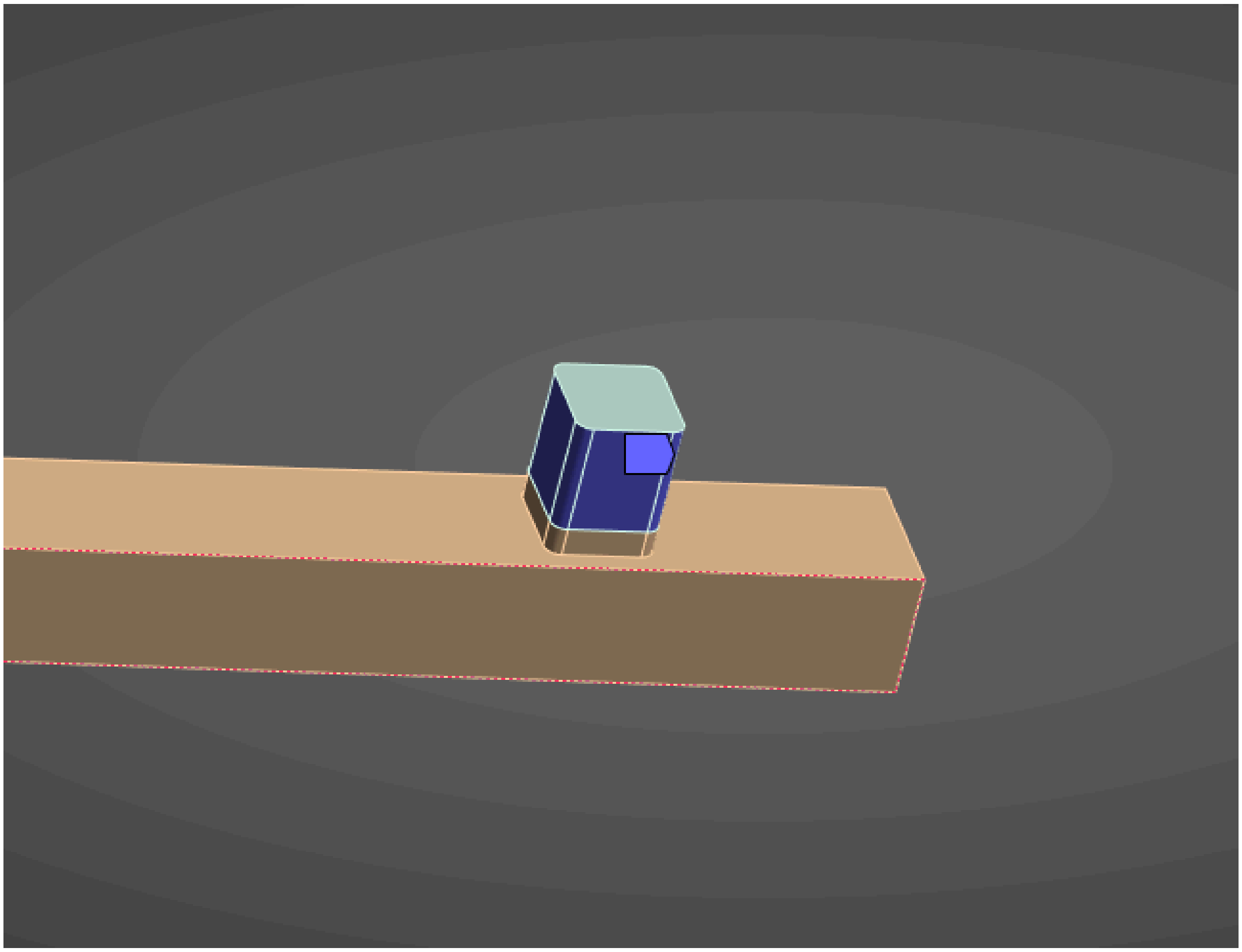
AISI 1080 is a high-carbon steel (≈0.80% C) known for developing very high hardness and strength after heat treatment. When oil-quenched from the austenitizing temperature and tempered at 315 °C, the microstructure becomes tempered martensite, providing an excellent balance of tensile strength (typically >1400 MPa), high yield strength, and good elastic modulus retention. Tempering at this intermediate temperature reduces brittleness while preserving the high hardness needed for resisting wear and maintaining dimensional stability under repeated loading.

Loading Conditions Diagram

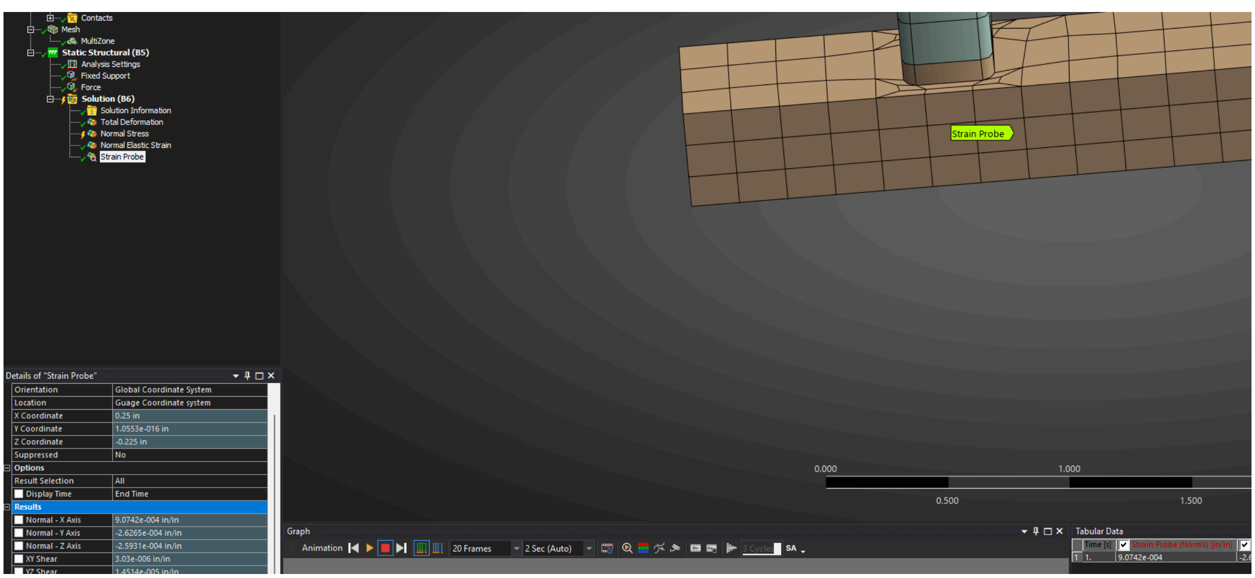
Force applied



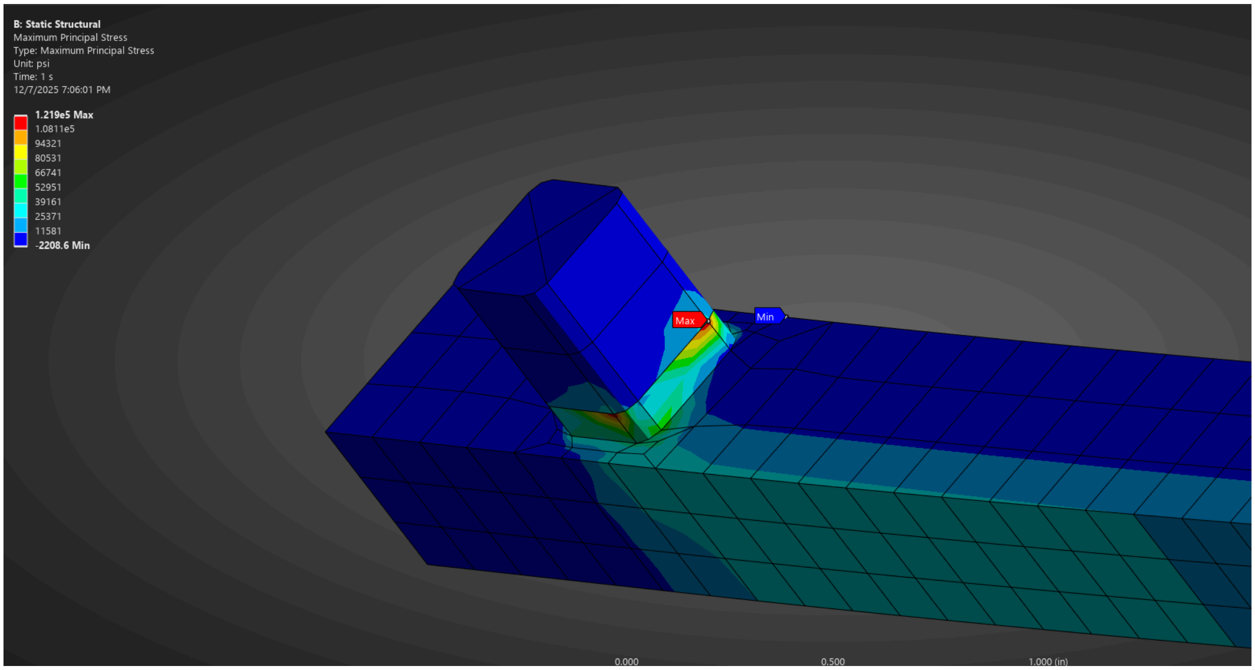
Clamped Socket Piece (all sides are fixed)



Normal Strain (9.0742E-4)

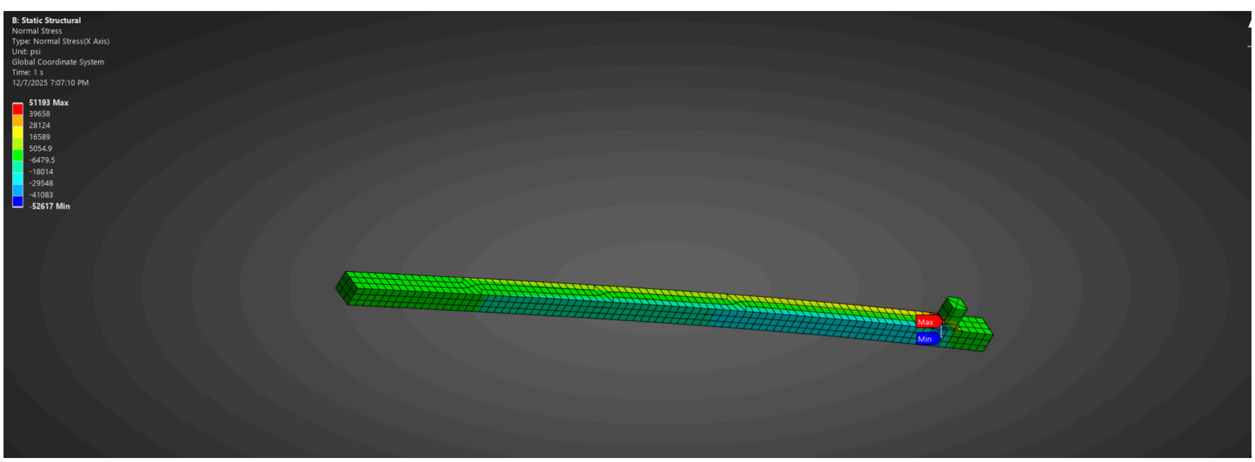


Contour of maximum principal stress - 1.219E5 PSI

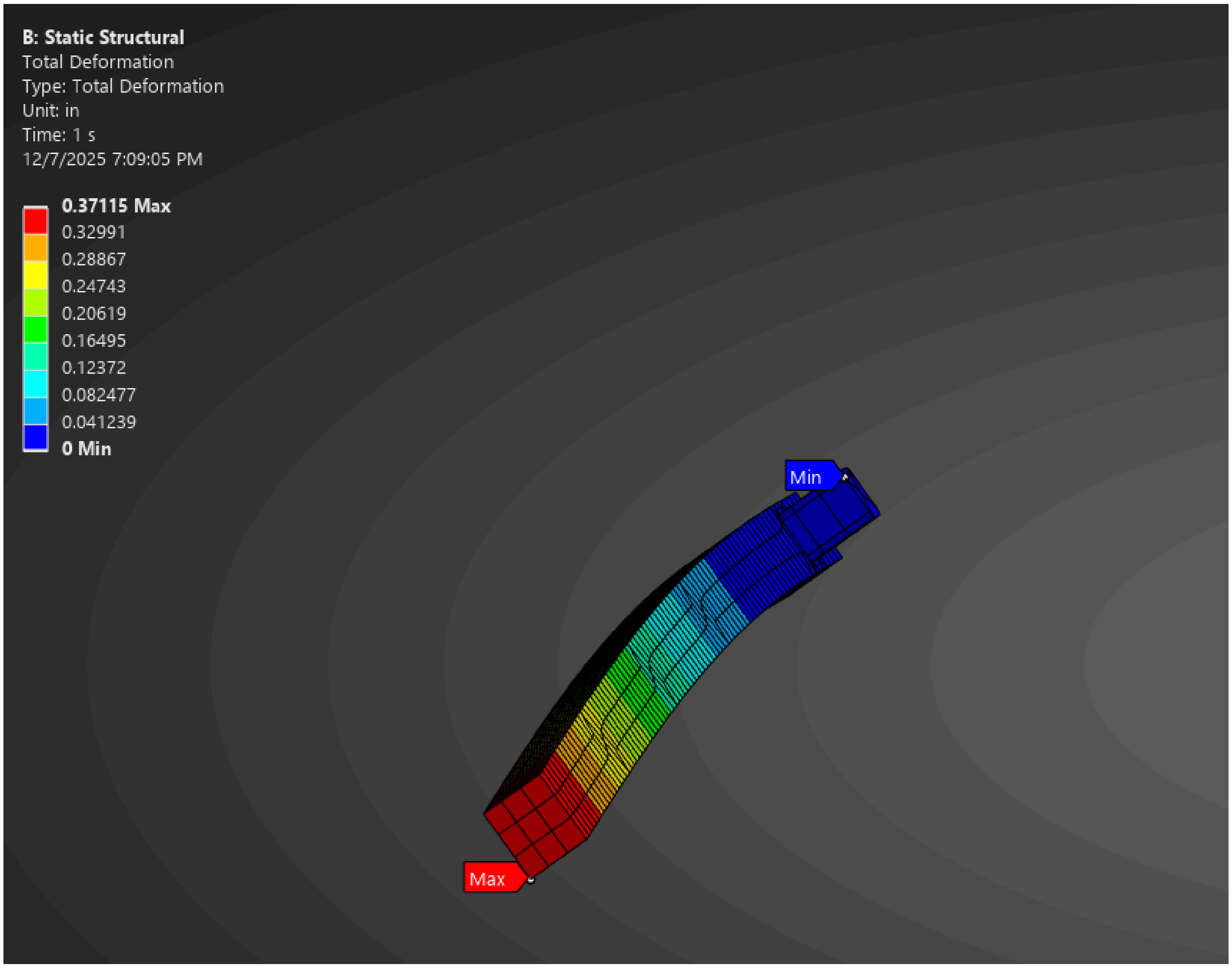


Summary of FEM results

Normal Stress - 51193 PSI or 51 ksi



Load point deflection - Max of .37115 in



Strain Probe Strains

	Time [s]	Strain Probe (NormX) [in/in]	Strain Probe (NormY) [in/in]	Strain Probe (NormZ) [in/in]
1	1.	9.0742e-004	-2.6265e-004	-2.5931e-004

Torque Wrench Sensitivity

To determine the torque-wrench output sensitivity, the strain at the strain-gauge location was taken directly from the finite-element model. The maximum principal strain available for measurement at the intended probe location was approximately 9.07e-4 in/in. This strain value corresponds to the applied torque used in the ANSYS simulation and therefore represents the full-scale operating strain for the sensor installation. A foil strain gauge bonded at this location produces an electrical output proportional to strain through the gauge factor, GF. For small strains, the bridge output of a Wheatstone bridge can be approximated with the following standard relationships:

Quarter-bridge output:

DeltaV/V = (GF * epsilon) / 4

Half-bridge output:

DeltaV/V = (GF * epsilon) / 2

Full-bridge output:

DeltaV/V = GF * epsilon

Using a typical gauge factor of 2.1 and the FEA strain epsilon = 9.07e-4, the predicted full-scale electrical outputs are:

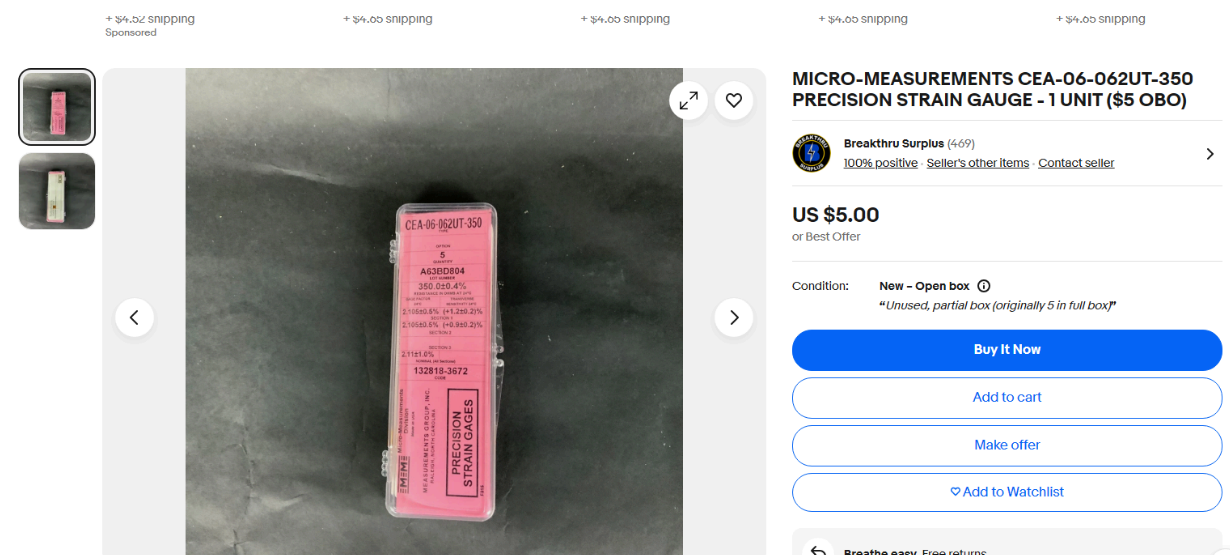
Full bridge: DeltaV/V = 2.1 * 9.07e-4 = 1.91e-3 (1.91 mV/V)

Half bridge: DeltaV/V = 2.1 * 9.07e-4 / 2 = 9.53e-4 (0.953 mV/V)

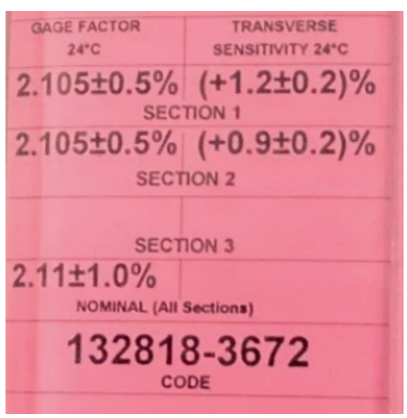
Quarter bridge: DeltaV/V = 2.1 * 9.07e-4 / 4 = 4.76e-4 (0.476 mV/V)

These values represent the torque-wrench sensitivity for the torque level applied in the FEA model. The sensitivity per unit torque can therefore be expressed as: Sensitivity = (DeltaV/V) / T_FEA, where T_FEA is the applied torque used in the simulation. This relationship allows direct conversion of measured bridge output to applied torque once the wrench is calibrated.

Strain gauge selection:

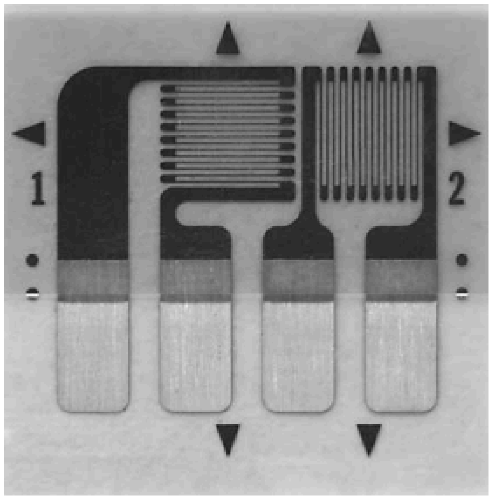


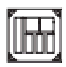
Example Device found on eBay



Guage Factor Zoom In

GAGE PATTERN DATA





actual size

GAGE DESIGNATION See Notes 1 and 4	RESISTANCE (OHMS) See Note 2	OPTIONS AVAILABLE See Note 3
CEA-XX-062UT-120	120 ± 0.4%	P2, SP35
CEA-XX-062UT-350	350 ± 0.4%	P2, SP35
CEA-XX-062UTA-350	350 ± 0.2%	P2, SP35

DESCRIPTION

Small general-purpose two-element 90° tee rosette. Exposed solder tab area 0.07 x 0.04 in (1.8 x 1.0 mm).

RoHS* COMPLIANT

GAGE DIMENSIONS

Legend
ES = Each Section
S = Section (S1 = Section 1)
CP = Complete Pattern
M = Matrix

inch / millimeter

Gage Length	Overall Length	Grid Width	Overall Width	Matrix Length	Matrix Width
0.062 ES	0.205 CP	0.080 ES	0.225 CP	0.31	0.31
1.57 ES	5.21 CP	2.03 ES	5.72 CP	7.9	7.9

Data Sheet for manufacturer-recommended alternative of comparable size

Description:

Four matched 350-ohm constantan linear foil gauges (e.g., Micro-Measurements CEA-series), .01 in gauge length, arranged in a full-bridge bending configuration with a GF of 2.11 +- 1.0%

The strain gauge selection was based on the available mounting width of less than 0.45 in (.31 in) and the need to measure the principal bending strain identified in the FEA. A linear foil gauge was required to align with the dominant strain direction, and a gauge length between 0.125 and 0.25 in was chosen to provide adequate strain averaging while keeping the backing width (~0.20

in) within the geometric limit. A 350-ohm constantan foil gauge was selected to minimize self-heating and ensure stable output. A full-bridge configuration would ideally be chosen to maximize sensitivity, improve temperature compensation, and provide a balanced bending-strain measurement suitable for torque wrench calibration.

Final Notes

While the peak normal stress is substantially higher than the expected value, it is understood that the design we made may have some over-concentration at points due to the limitations of the clamped surface and contact points with the rounded edge,s as described by Professor Bhaskaran in his own example. Our overall model, aside from the point maxes, is at or above our predicted stress from the hand calculations.

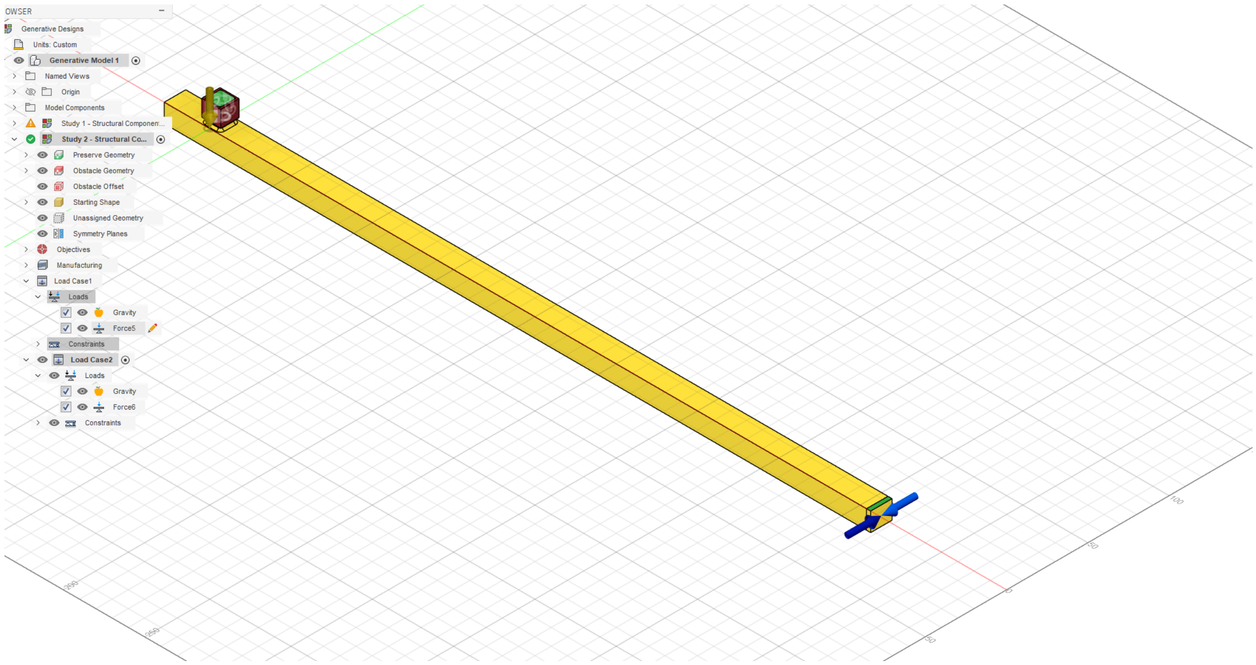
Generative Design Section

To further my own interests in working with generative design tools, I chose to challenge myself and create a torque wrench model using Fusion 360's built-in Generative design tool.

The steps I took are as follows:

- Import the design of the torque wrench,
- Identify and establish Preserve Geometry,
- Designate Obstacle Geometry
- Apply structural loads and structural constraints to the model
- Refine results until a suitable model can be exported from Fusion to ANSYS.

Fusion Generative Parameters



Later modified to have a larger force bounding box (highlighted in green with blue arrows)

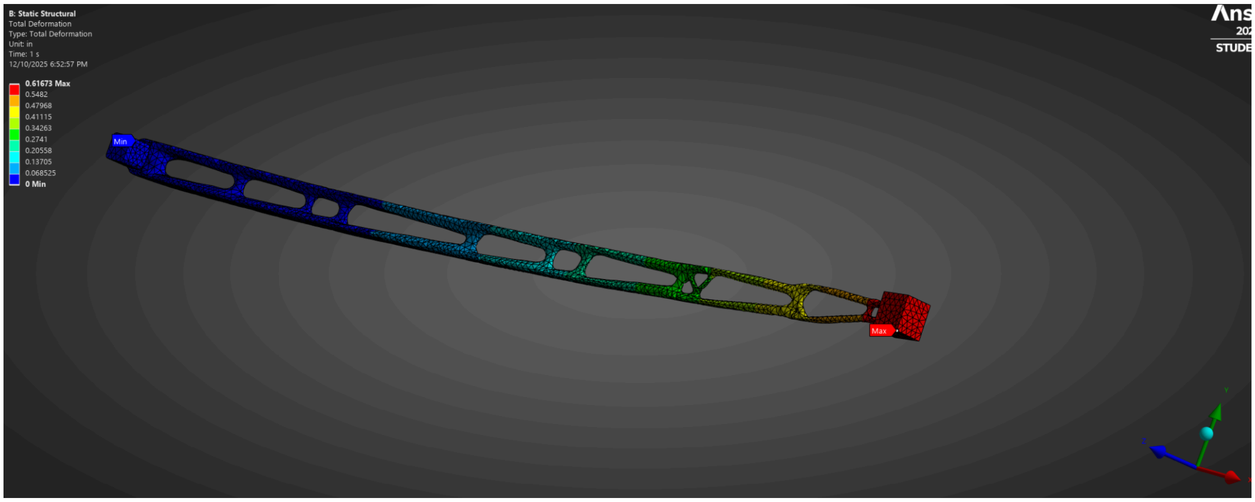
Results

Trial 1

Trial 2

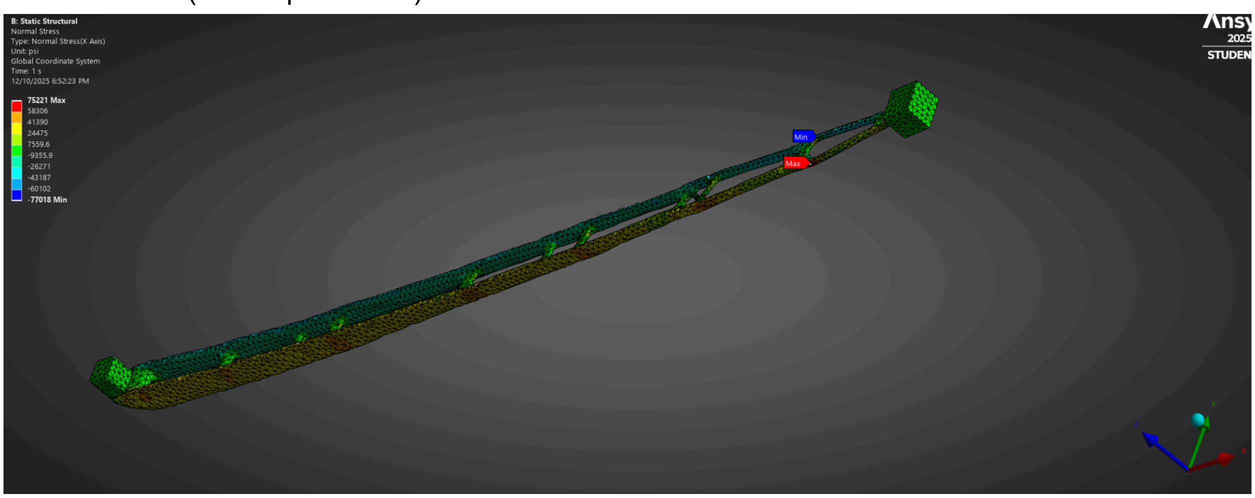
Trial 3 Final

Deformation - .61 in



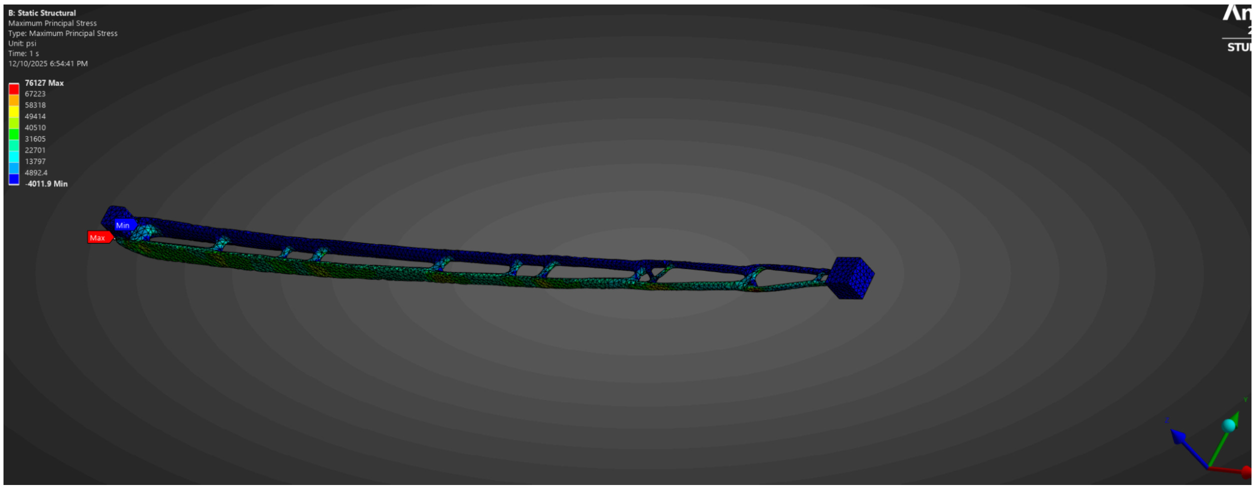
The deformation of the generative-designed wrench is noticeably larger than that of the redesigned prismatic model, reaching approximately **0.61 in** at the loading point. This increased flexibility is expected because the generative geometry removes significant material along the handle to reduce weight and follow load paths more efficiently, resulting in a more compliant structure. In contrast, the redesigned solid model used in the main report produced a deformation of roughly **0.37 in**, indicating a stiffer and more conventional beam-like response. While the generative design still carries the applied torque, its higher deflection suggests a trade-off between lightweight efficiency and overall stiffness, which is an important consideration when evaluating tool feel, accuracy, and long-term durability.

Normal Strain (75221 psi - max)



The elevated maximum normal stress shown in the FEA is likely conservative and still acceptable because of how the load was modeled. In the simulation, the entire 600 in-lbf torque was applied as a concentrated force at a single point at the extreme end of the wrench, creating an artificially severe stress gradient. In real use, however, a user applies force over a finite grip region along the length of the handle, distributing the load rather than concentrating it. This distributed loading reduces localized bending moments and lowers peak stresses compared to the point-load idealization used in the model. As a result, the simulated maximum stress represents a worst-case scenario and provides a built-in safety margin relative to actual operating conditions.

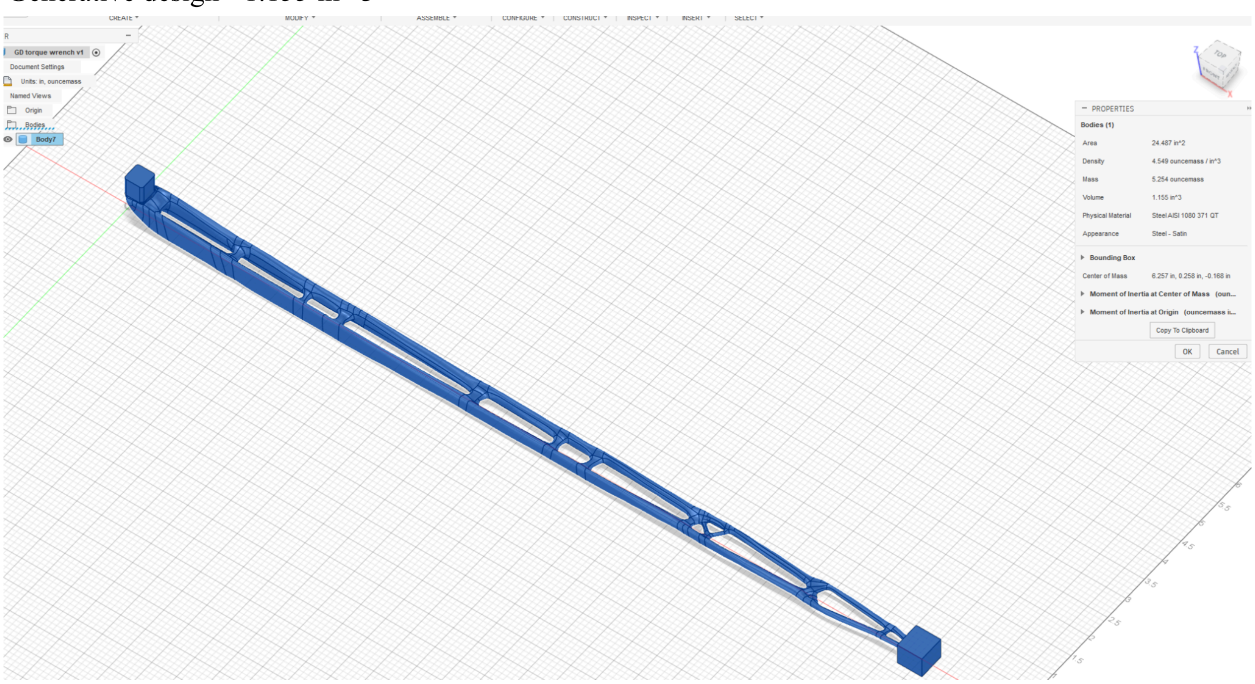
Maximum principal stresses



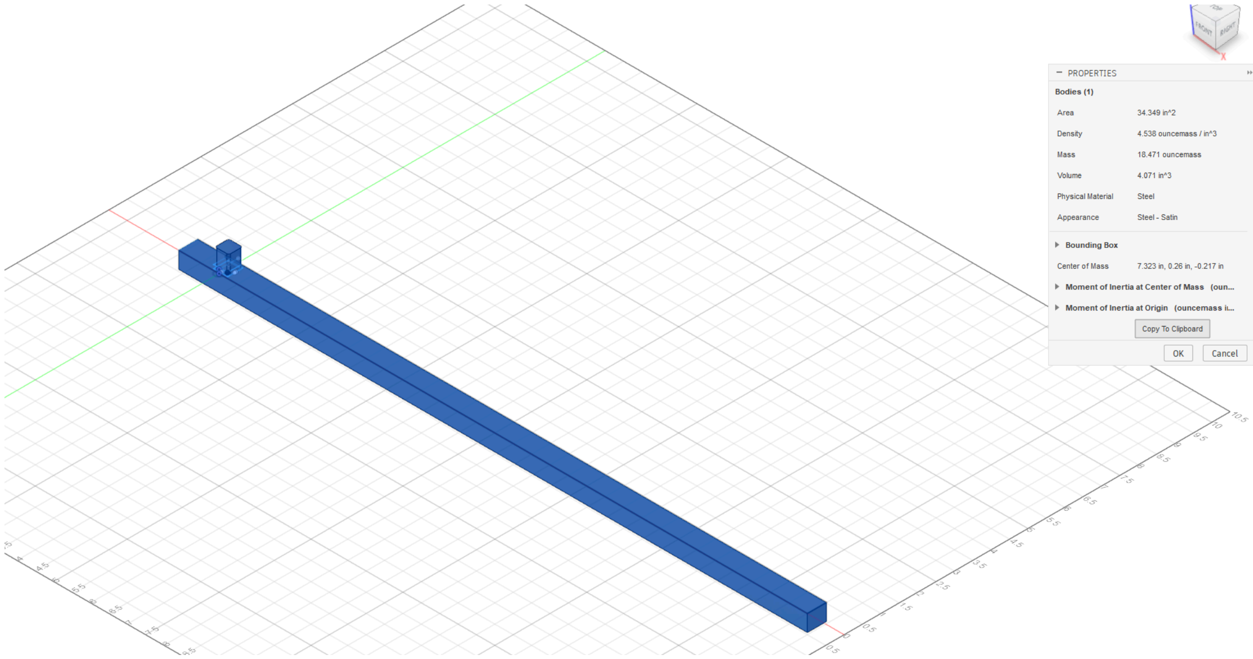
For principal stresses here, the same argument could be applied that these point cases of maximum stress could be a point of failure, they also could be a misrepresentation of compounding rounding errors in the model, and rounded corners on the socket, creating point stresses overly high.

Difference in Volume/Mass

Generative design - 1.155 in ^3



Original Design -



4.071 in^3

A significant distinction between the original wrench geometry and the generative design is the dramatic reduction in both volume and mass. The original prismatic beam design has a volume of approximately **4.071 in³**, while the generative design reduces this to just **1.155 in³**—a decrease of over **70%**. This reduction is a direct result of the generative algorithm removing non-critical material and creating a highly optimized internal structure that follows the primary load paths. The resulting geometry is far lighter and materially efficient, improving manufacturability for additive processes and decreasing overall tool mass. However, the substantial mass savings come with trade-offs in stiffness and deformation, highlighting the balance between structural efficiency and mechanical performance when employing generative design techniques.