5.1 Baseline Design

5.1.1 Results

1. Script used for hand calculations

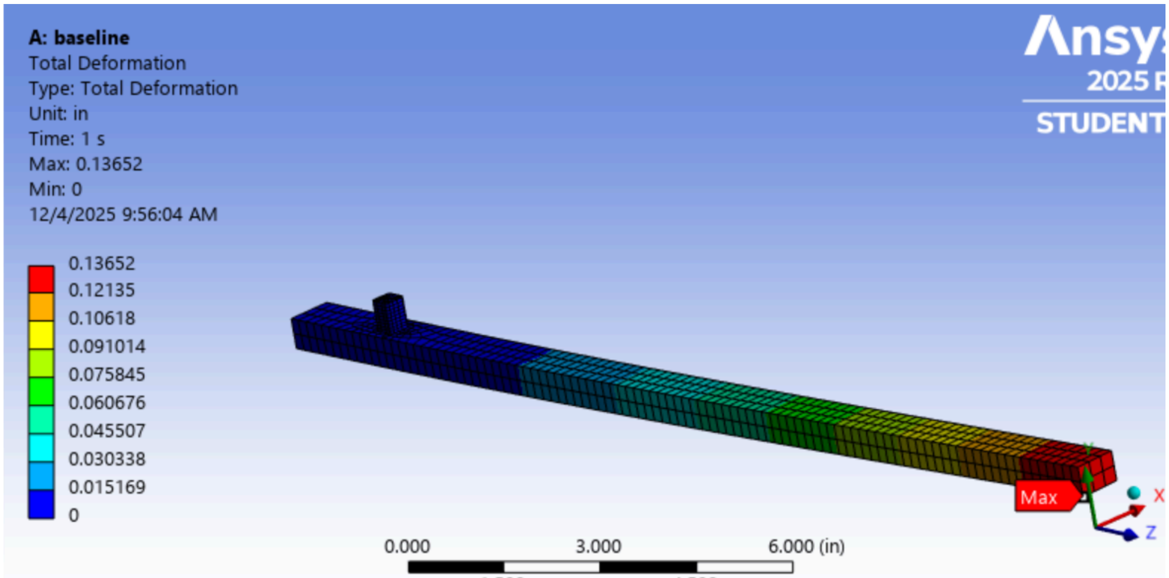
```
%MAE 3270 final HW torque wrench
M = 600; % max torque (in-lbf)
L = 16; % length from drive to where load applied (inches)
h = 0.75; % width
b = 0.5; % thickness
c = 1.0; % distance from center of drive to center of strain gauge
E = 32.E6; % Young's modulus (psi)
nu = 0.29; % Poisson's ratio
su = 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
a = 0.04; %crack length (in)
I = (b*h^3)/12; %moment of inertia (in^4)
%deflection
umax = M*L^2/(3*E*I)
% maximum stress
maxStress = M*h/(2*I)
% crack length resistance
KI = 1.12*sqrt(pi*a)*6*M/(b*h^2);
%Fatigue Failure
sallow = M*6/(b*h^2);
%safety factors
Xo = su/maxStress
Xk = KIC/KI
Xs = sfatigue/sallow
%stress at c
stressAtC = (M/L)*(L-c)*(h/2)/I;
%strain at c (in microstrain)
strainAtC = (stressAtC/E * 10^6)
voltAtC = strainAtC*10^(-3) %in mV/V
```

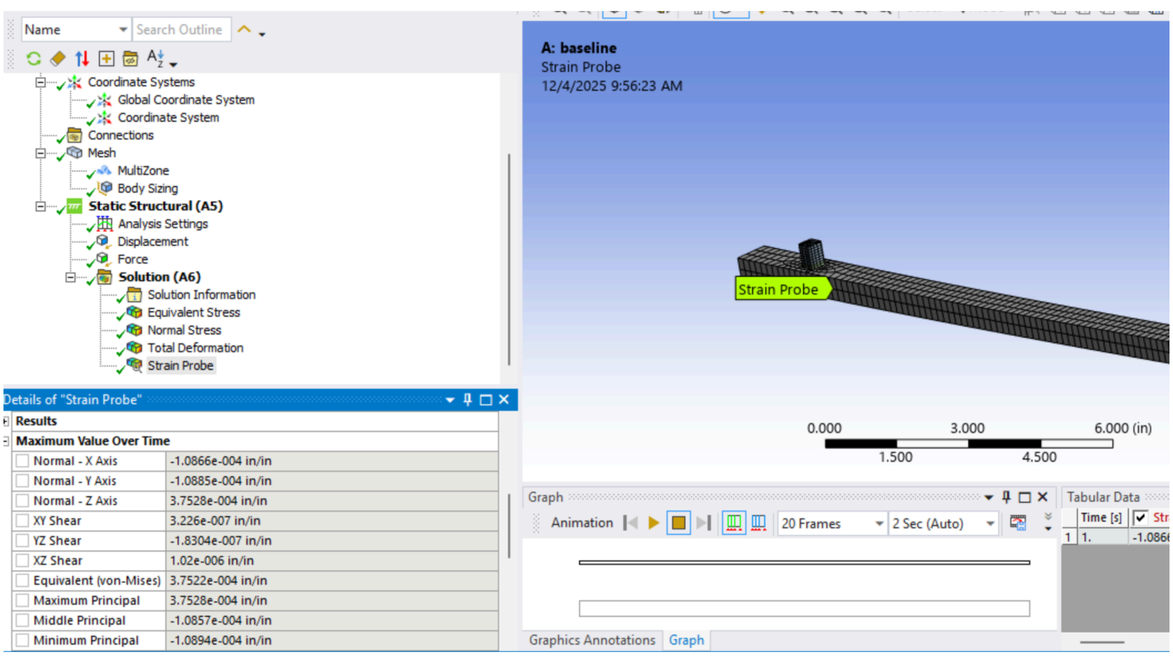
2. Results from hand calculation of base design showing maximum normal stress (anywhere), strains at the strain gauge locations and deflection of the load point

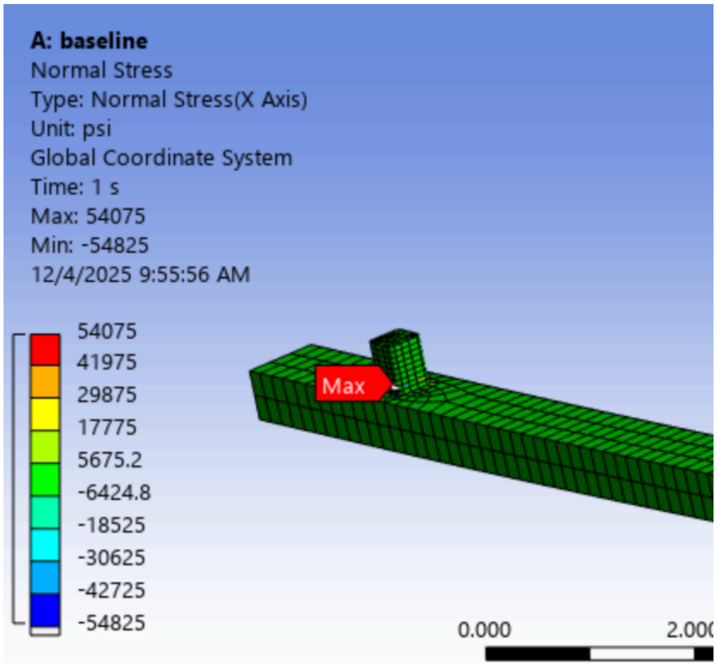
Maximum normal stress: **12.8 ksi**
Strain at strain gauge location: **375 microstrain**
Deflection of load point: **0.091 in**

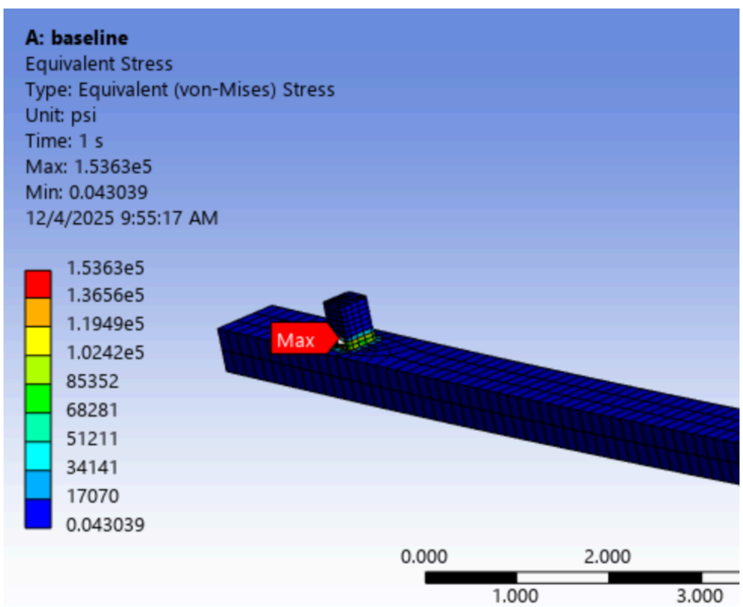
3. Results from FEM calculation of base design. From the FEM find the maximum normal stress (anywhere), strains at the strain gauge locations and deflection of the load point.

Maximum normal stress: 1.54e5 psi which is **15.4 ksi** (Equivalent Von-Mises normal stress)
Strain at strain gauge location: **375 microstrain**
Deflection of load point: **0.137 in**









5.1.2 Reflections

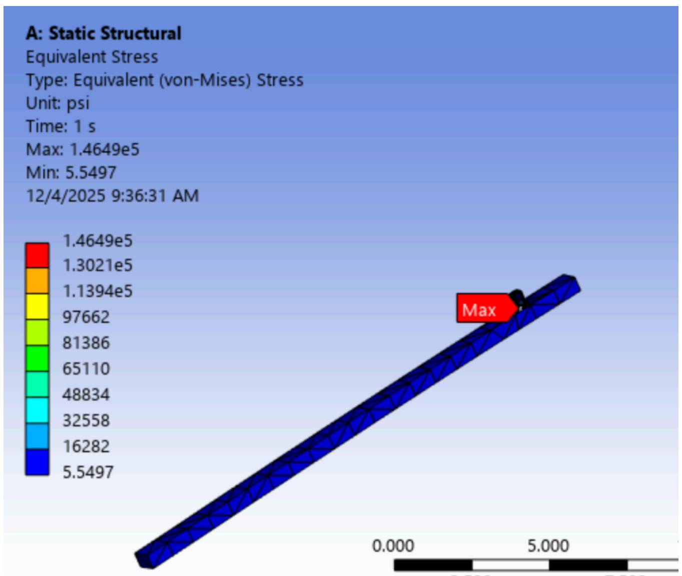
1. Beam theory assumes that plane sections remain plane. View the deformed mesh and check if mesh lines that cut across the beam handle remain as straight lines. Do you think that beam theory is reasonably accurate?

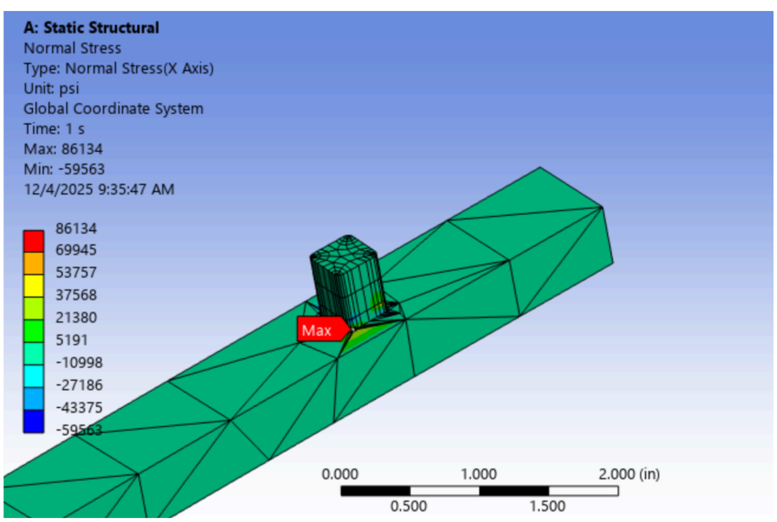
I think beam theory is reasonably accurate because the mesh lines that cut across the beam handle do remain in straight lines. This means that the deformation is small enough to approximate the plane sections as a plane.

2. How do the FEM and hand calculated maximum normal stresses compare? If they differ significantly, why?
The FEM maximum normal stress is larger than the hand calculated maximum stress. The FEM max normal stress was 15.4 ksi while the hand calculated value was 12.8 ksi. This could be because there are stress concentrations in the model that the hand calculations did not account for, especially around the base of the drive.

3. How do the FEM and hand calculated displacements compare? If they differ, why?
The FEM maximum displacement from the FEM was 0.13652 in. while the hand calculated value was 0.091 in. These are similar, but not quite identical. This may be because the plane sections may not be exactly plane the way beam theory assumes them to be.

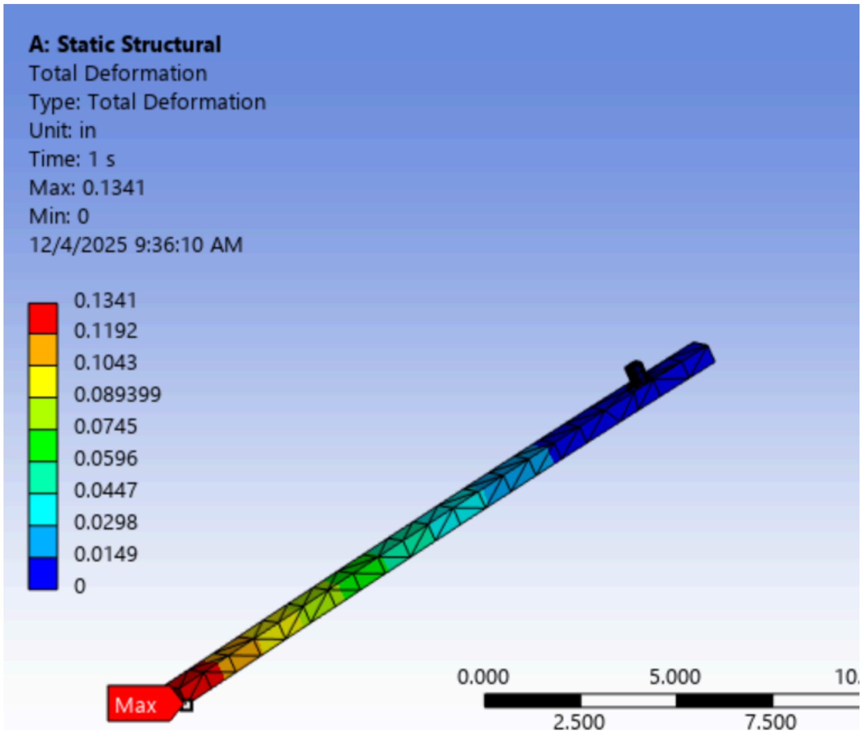
ANSYS OLD MODEL:





At strain gauge: 372 microstrain while the hand calcs get 375 microstrain

Refined the mesh and then when we looked at the strain at the strain gauge the strain was the same as our hand calculations!

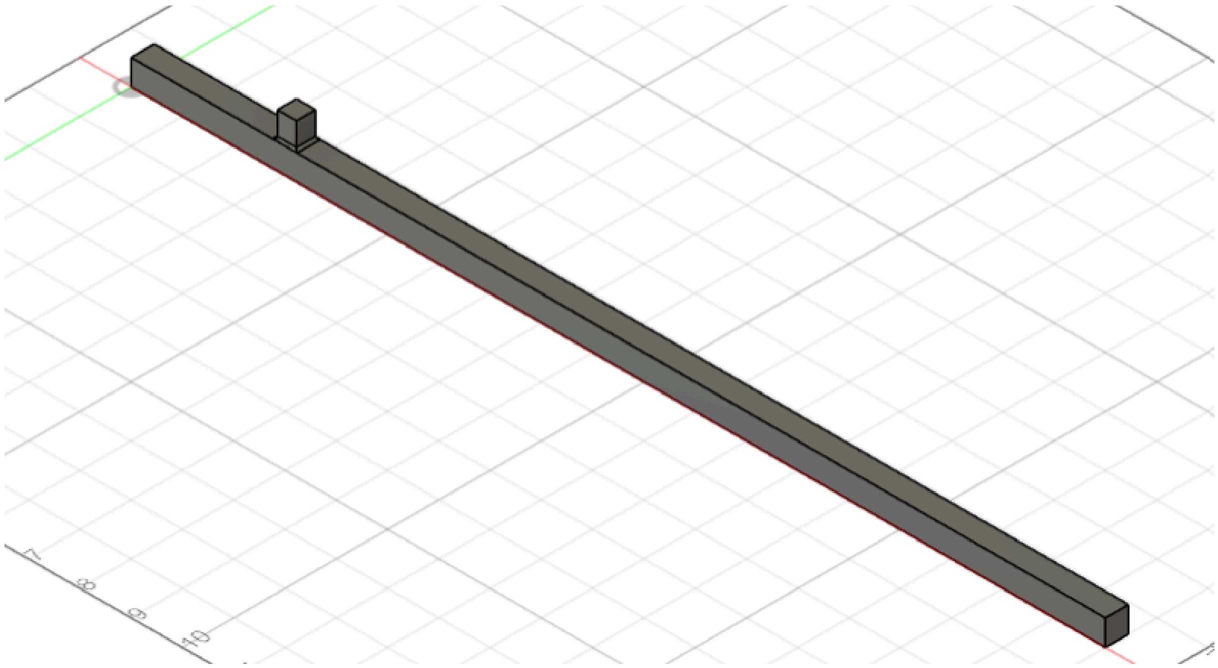


5.2 Your Design

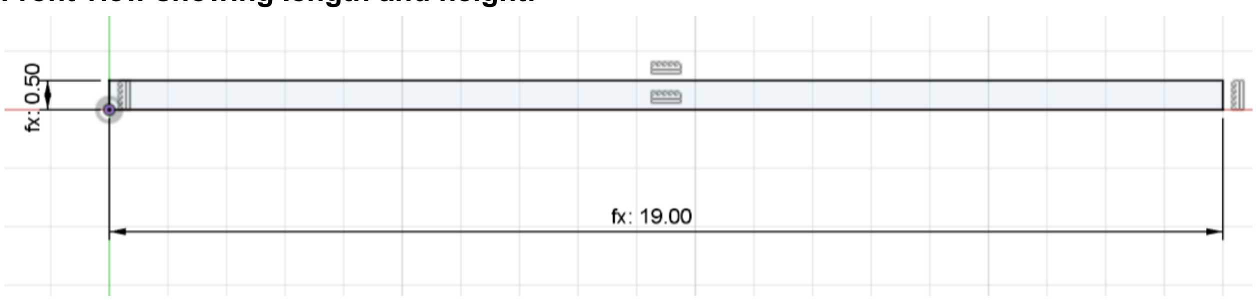
5.2.1 Results

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

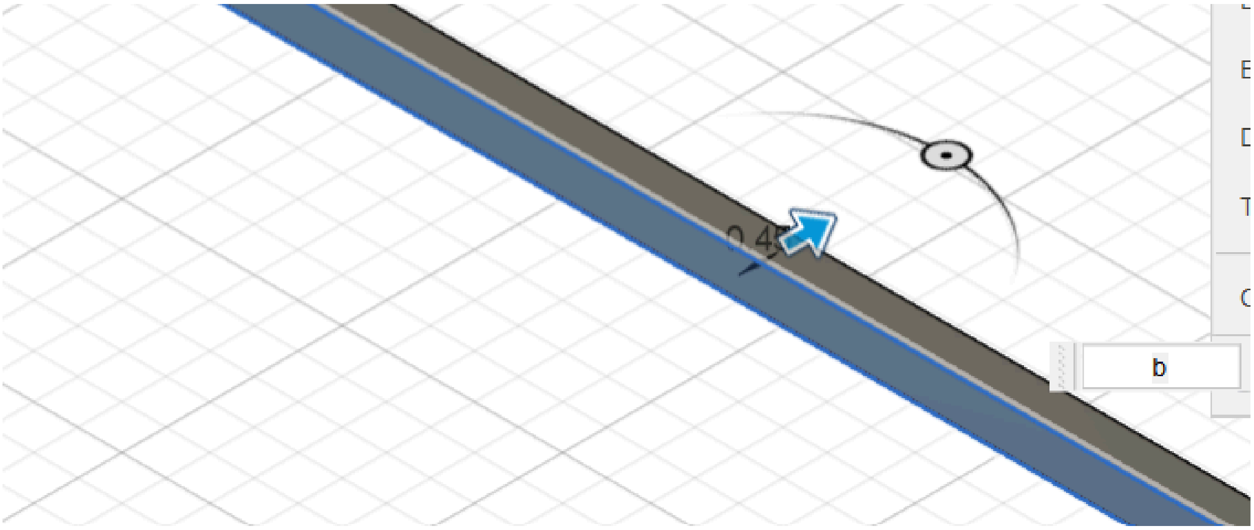
Isotropic view (no dimensions):



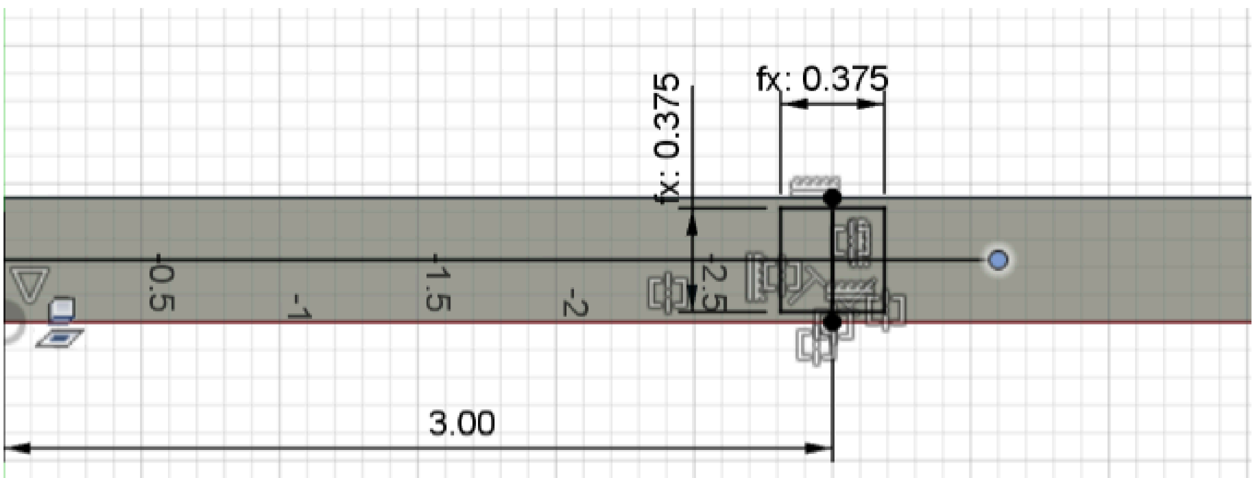
Front view showing length and height:



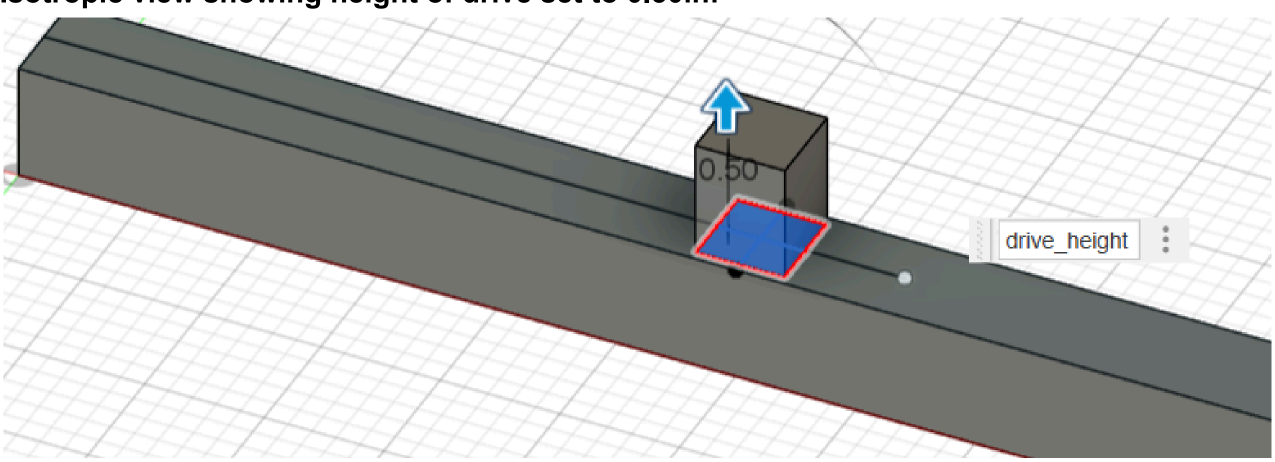
Isometric view showing depth of b = 0.45in:



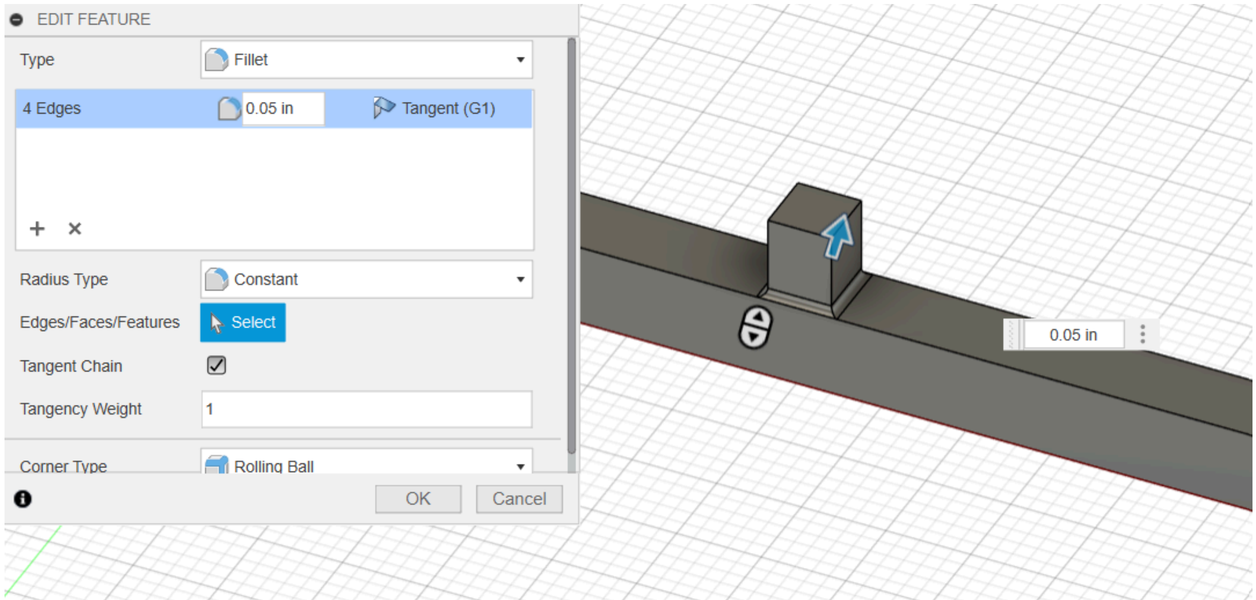
Top view showing distance of drive from end of torque wrench and width of drive:



Isotropic view showing height of drive set to 0.50in:

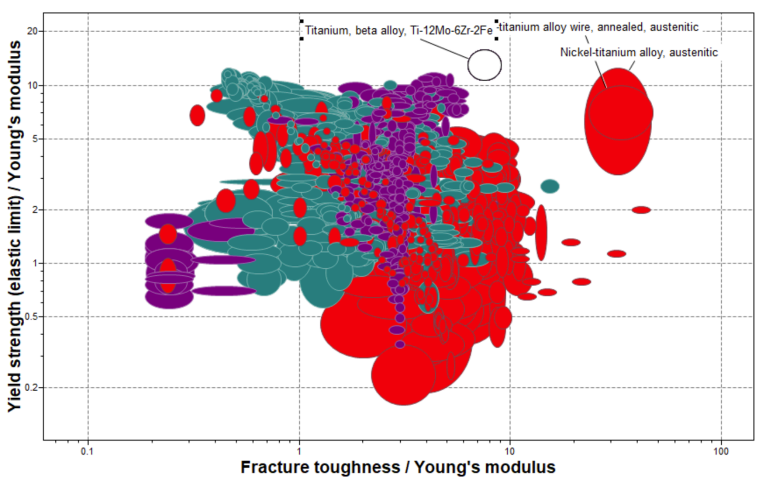


Isotropic view showing fillet radius of 0.05in:



2. Describe material used and its relevant mechanical properties.

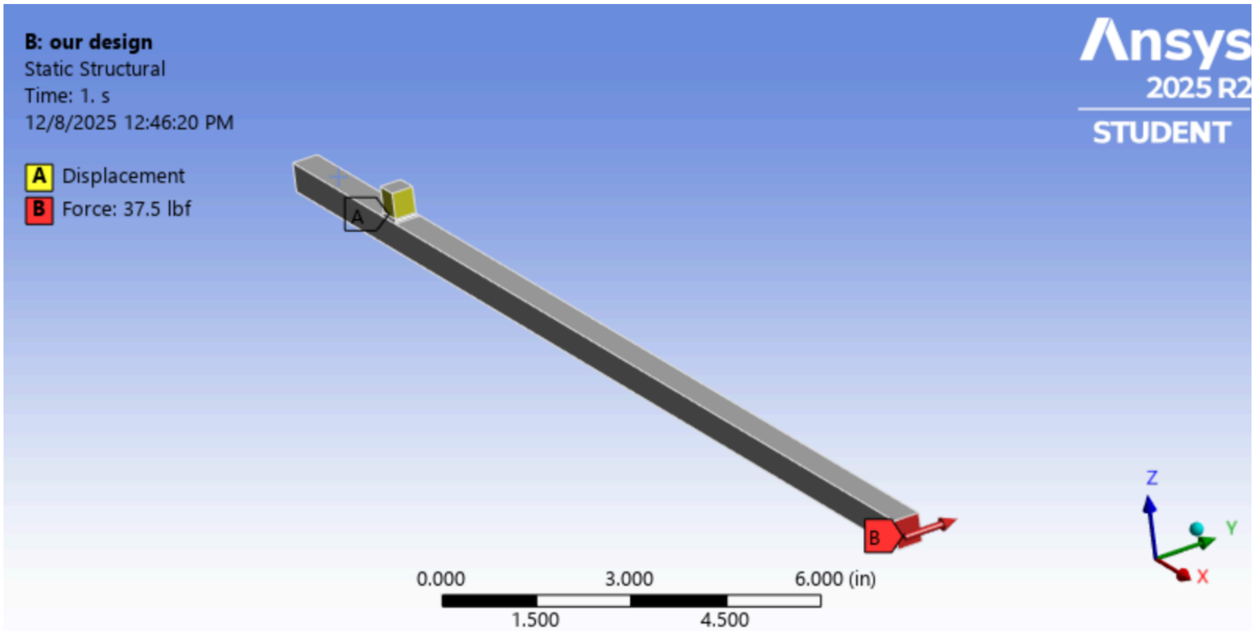
We chose to use Titanium beta alloy Ti-12Mo-6Zr-2Fe. This material is known for its high strength and ease of being formed. It also has a pretty good strength to weight ratio, which may not be as critical for this application. We chose this material by creating a plot of fracture toughness/Young's Modulus vs. yield strength/Young's Modulus for different steel, titanium, and aluminum alloys and choosing a material that had high values for both of these variables. This graph is pictured below.



Material properties from Granta (bolded is value chosen to be used in calculations)
E = **9.16** to 13.1 10^6 psi (Young's modulus)
nu = **0.31** to 0.35 (Poisson's ratio)
su = **135** to 164 ksi (tensile strength)

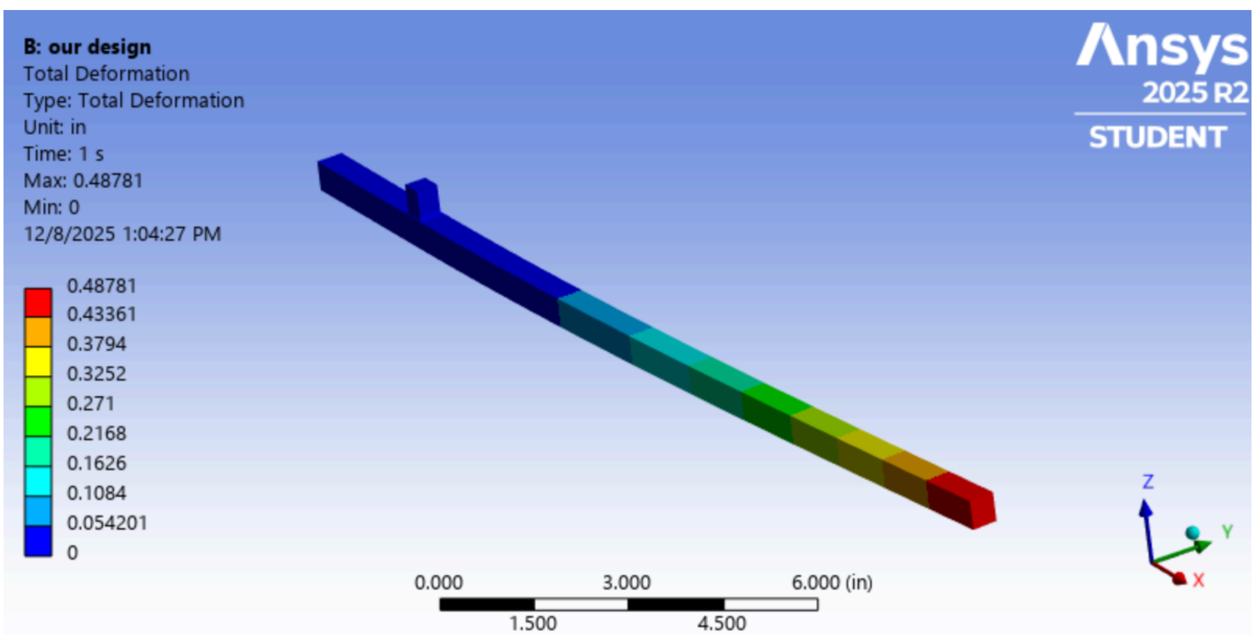
KIC = **80.1** to 83.7 ksi.in^0.5 (Fracture toughness)
sfatigue = **100** psi (fatigue strength for 10^6 cycle)

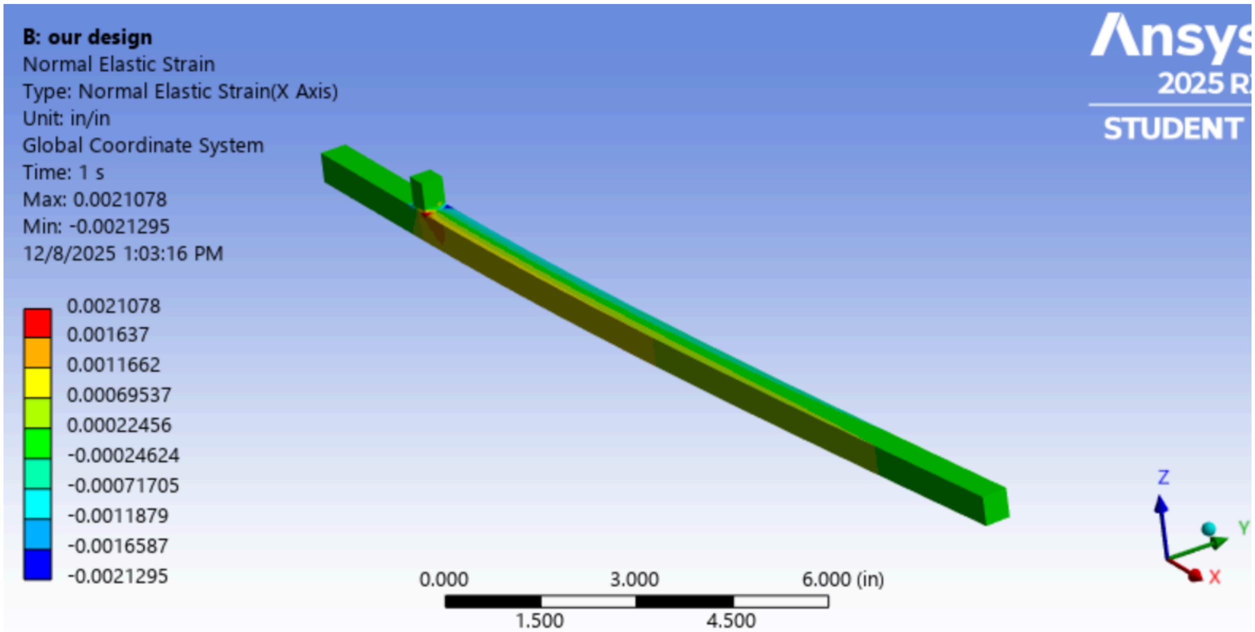
3. Diagram communicating how loads and boundary conditions were applied to your FEM Model.

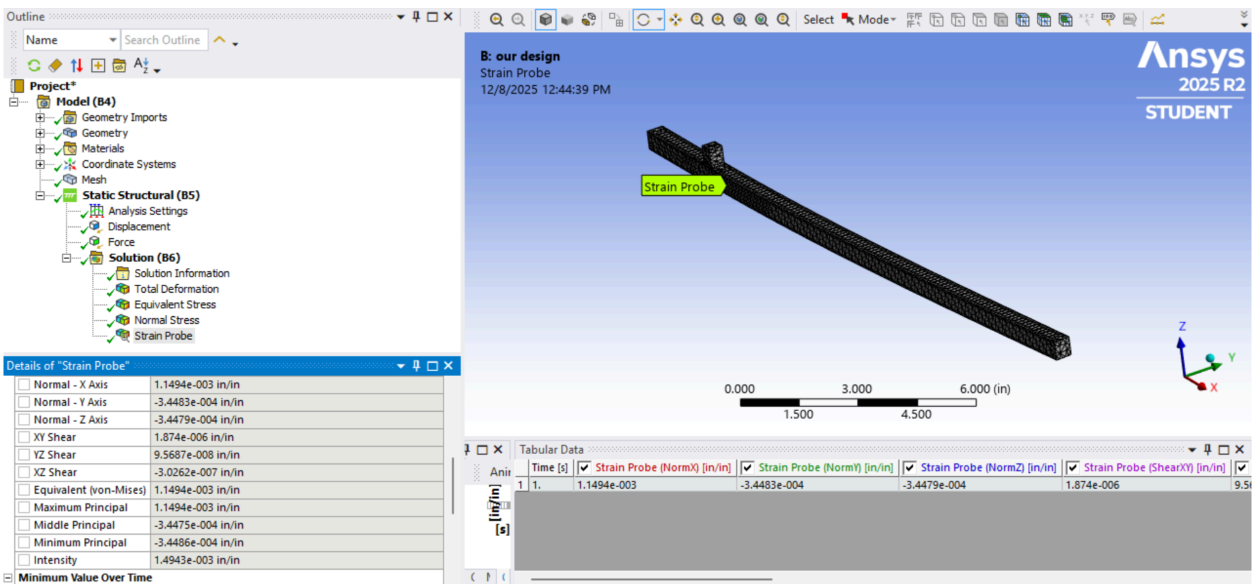


Yellow areas set at 0 displacement. Red force value of 37.5 lbf.

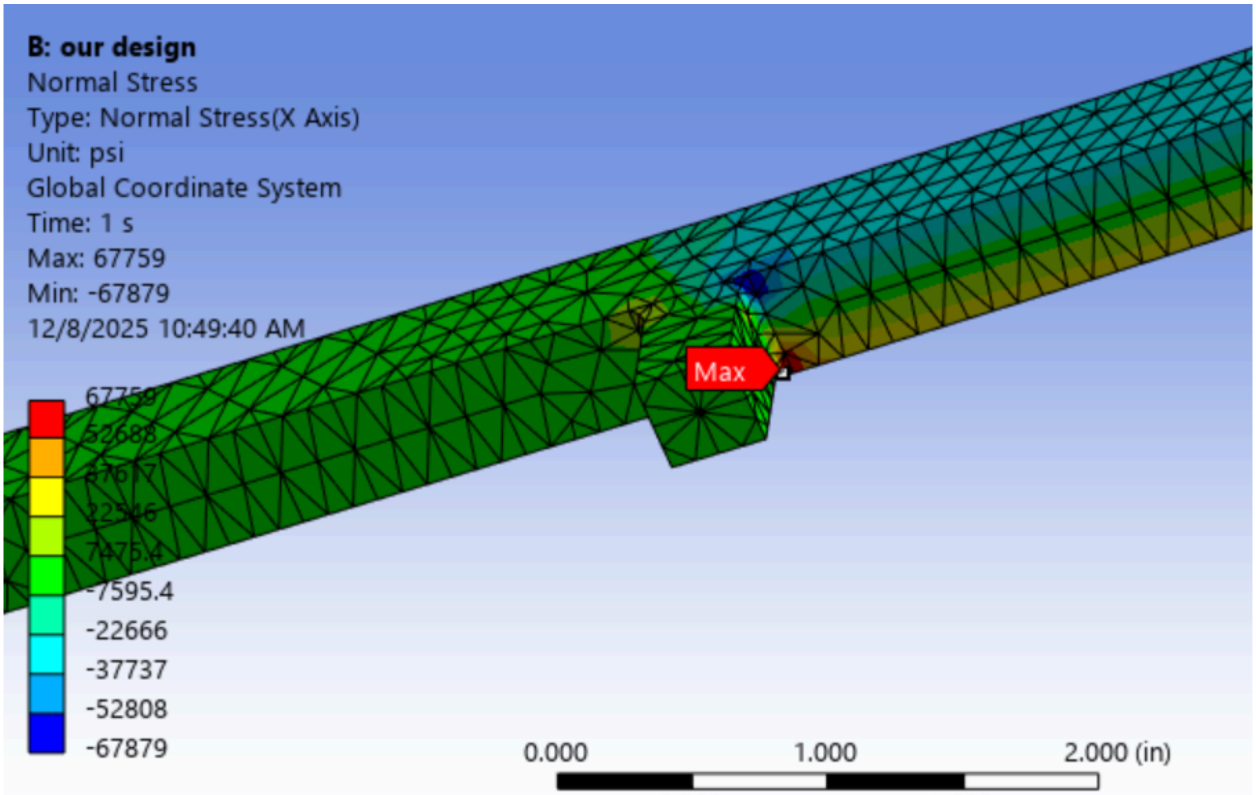
4. Normal strain contours (in the strain gauge direction) from FEM

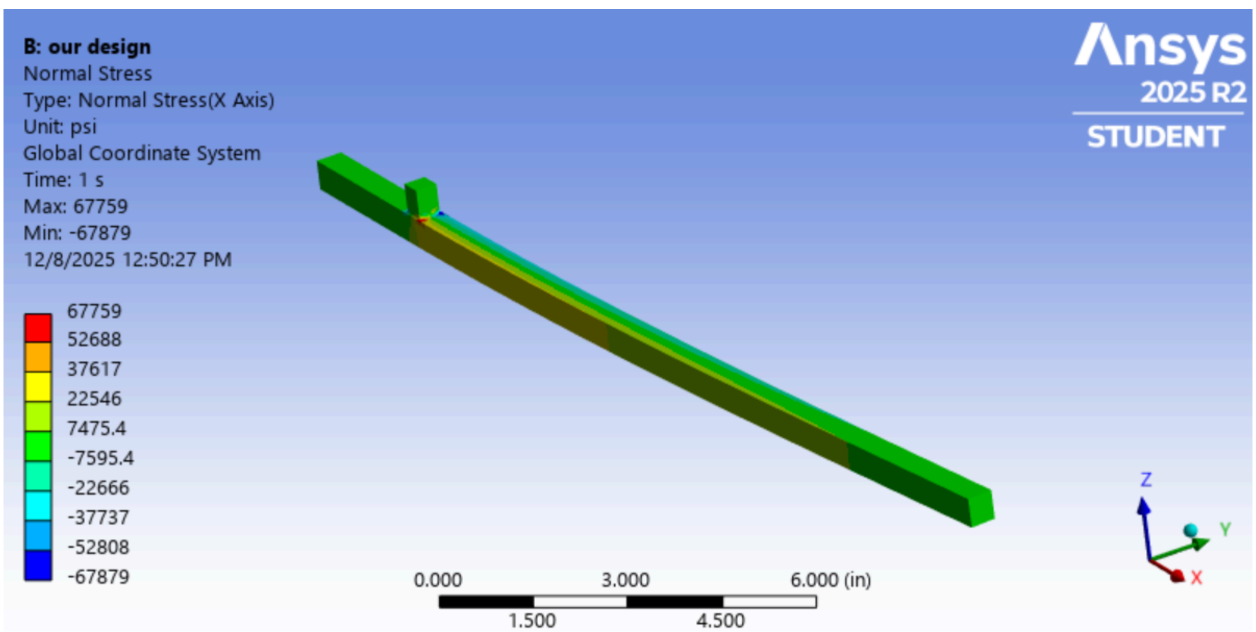






5. Contour plot of maximum principal stress from FEM





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

Maximum normal stress: **67.7 ksi**
Strain at strain gauge location: **1.15 microstrain**
Deflection of load point: **0.48 in**

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

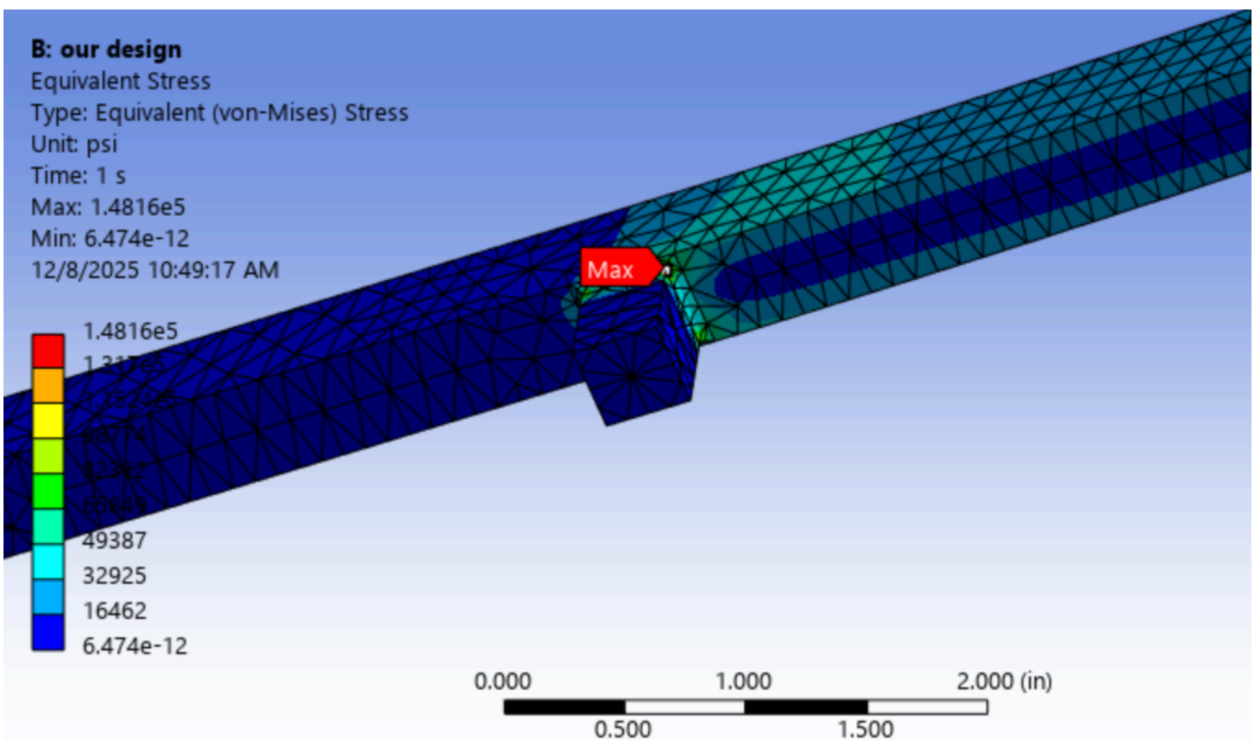
At the strain gauge, the strain is expected to be 1.15e-3 in/in, which is equivalent to 1.15 mV/V.

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

Strain gauge choice: SGD-3/120-LY11
Gauge Type: Linear pattern, grid width narrow
Carrier Length: 7.8mm = 0.31in ← Less than both the width and height (0.45 in.) of our design so it will fit
Carrier Width: 3.8mm = 0.15in
Grid Length: 3mm = 0.12in
Grid Width: 1.5mm = 0.06in

The largest dimension of the strain gauge is smaller than the smallest dimension of where the strain gauge will go, so we are confident that the strain gauge will fit.

Additional FEM Outputs:



Outputs from MATLAB:

```
Factor of Safety Requirements:
Xo > 4
Xk > 2
Xs > 1.5
Voltage > 1
```

<u>Iteration 1: Using original dimensions</u>

```
Dimensions:
L = 16; % length from drive to where load applied (inches)
h = 0.75; % width
b = 0.5; % thickness

Output:
umax =   0.3180
maxStress = 12800

Xo = 10.5469
Xk =  15.7616
Xs =  7.8125

strainAtC = 1.3100e+03
voltAtC =  1.3100
```

<u>Iteration 2: Change h from 0.75in to 0.5in</u>

```
L = 16; % length from drive to where load applied (inches)
h = 0.5; % width
b = 0.5; % thickness

Output:
umax =  1.0732
maxStress =  28800

Xo =  4.6875
Xk =   7.0051
Xs =   3.4722

strainAtC =  2.9476e+03
voltAtC =  2.9476
```

<u>Iteration 3: Change b from 0.5in to 0.3in</u>

```
L = 16; % length from drive to where load applied (inches)
h = 0.5; % width
b = 0.3; % thickness

Output:
umax =  1.7886
```

maxStress = 4.8000e+04

Xo = 2.8125 ← failed 🙁
Xk = 4.2031
Xs = 2.0833

strainAtC = 4.9127e+03
voltAtC = 4.9127

Iteration 4: Change b from 0.3in to 0.4in
L = 16; % length from drive to where load applied (inches)
h = 0.5; % width
b = 0.4; % thickness

Output:
umax = 1.3415
maxStress = 36000

Xo = 3.7500 ← failed 🙁
Xk = 5.6041
Xs = 2.7778

strainAtC = 3.6845e+03
voltAtC = 3.6845

Iteration 5: Change b from 0.4in to 0.45in
L = 16; % length from drive to where load applied (inches)
h = 0.5; % width
b = 0.45; % thickness

Output:
umax = 1.1924
maxStress = 32000

Xo = 4.2188
Xk = 6.3046
Xs = 3.1250

strainAtC = 3.2751e+03
voltAtC = 3.2751

Iteration 6: Change L from 16 in to 17in
L = 17; % length from drive to where load applied (inches)
h = 0.5; % width
b = 0.45; % thickness

Output:
umax = 1.3461

```
maxStress = 32000

Xo = 4.2188
Xk = 6.3046
Xs = 3.1250

strainAtC = 3.2880e+03
voltAtC = 3.2880
```

Iteration 7: Change L from 17in to 16in and change h from 0.5 in to 0.3in

```
L = 16; % length from drive to where load applied (inches)
h = 0.3; % width
b = 0.45; % thickness

Output:
umax = 5.5205
maxStress = 8.8889e+04

Xo =  1.5188 ← failed 🙁
Xk =  2.2697
Xs =   1.1250 ← failed 🙁

strainAtC =  9.0975e+03
voltAtC = 9.0975
```

Iteration 8: Change h from 0.3 in to 0.4in

```
L = 16; % length from drive to where load applied (inches)
h = 0.4; % width
b = 0.45; % thickness

Output:
umax = 2.3290
maxStress =  5.0000e+04

Xo = 2.7000 ← failed 🙁
Xk = 4.0350
Xs = 2.0000

strainAtC = 5.1174e+03
voltAtC = 5.1174
```