MAE 3260

Final HW Part 1

Baseline Design Analysis

Members: Ana Badea (amb675), Alison Chavez (avc35)

Date Due: 12/8/2025

Baseline Design Analysis

1. Results

1.1) Script used for hand calculation.

```
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

a = 0.04;

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

% Calculate the moment of inertia

I = (b * h^3) / 12;

% Calculate the maximum normal stress (sigma)

% sigma =  M*(h/2)/I;

sigma_norm_ksi = sigma / 1000 % Convert to ksi

load_point_deflection = (M * L^2) / (3 * E * I)

F = M/L;

Mgage = F*(L-c);

sigma_g=  Mgage*(h/2)/I;

% Calculate the strain (epsilon1) at the strain gauge location
```

```
k = 2;

epsilon1 = sigma_g / E;

epsilon2 = -epsilon1;

vout_vin = k/4 * 2*epsilon1; % THIS IS CORRECT formula value

strain =epsilon1/10^-3

voltageOutput = vout_vin/10^-3

Ki= 1.12*sigma*sqrt(pi*a);

X_CrackGrowth = KIC/Ki;

X_Yield_strength = su / sigma;

X_Fatigue = sfatigue / abs(sigma); % Safety factor for fatigue
```

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

Results from script:

Maximum Normal Stress - 12.8000 ksi

Strain at gauge - 375 microstrain

Deflection of the load point - 0.0910 in

1.3) 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.

Our FEM setup followed the steps done in lecture, including clamping the driver (via the Displacement "A") and applying the load at the edge of the handle (via the Force "B").

Ansys Setup:

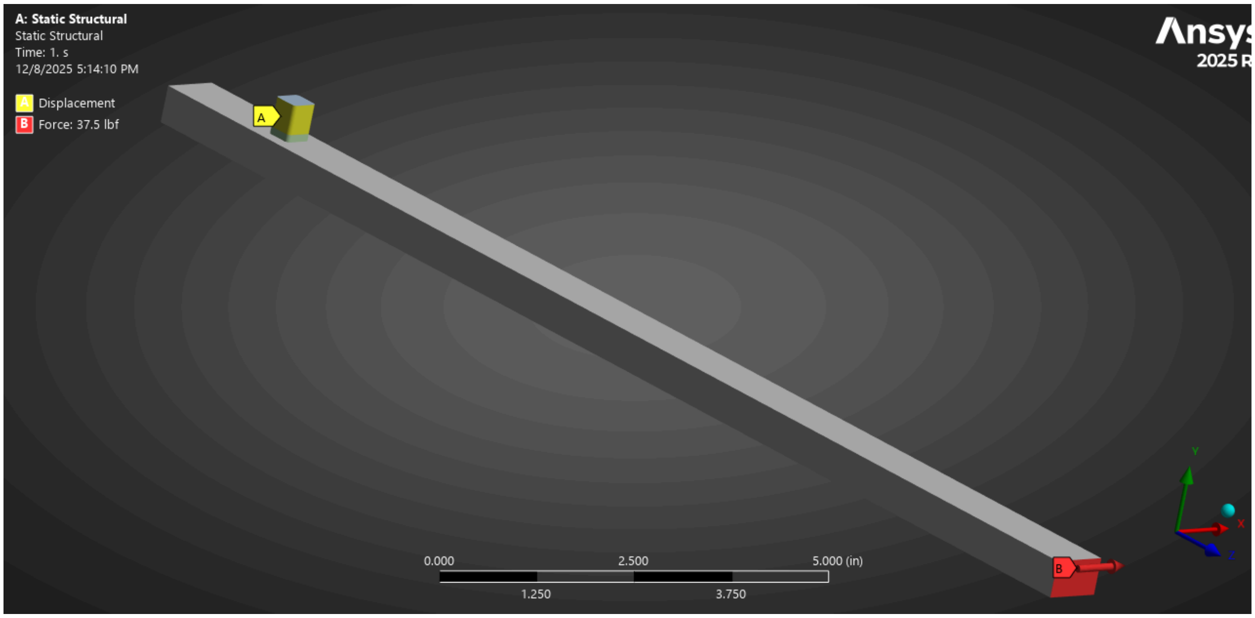
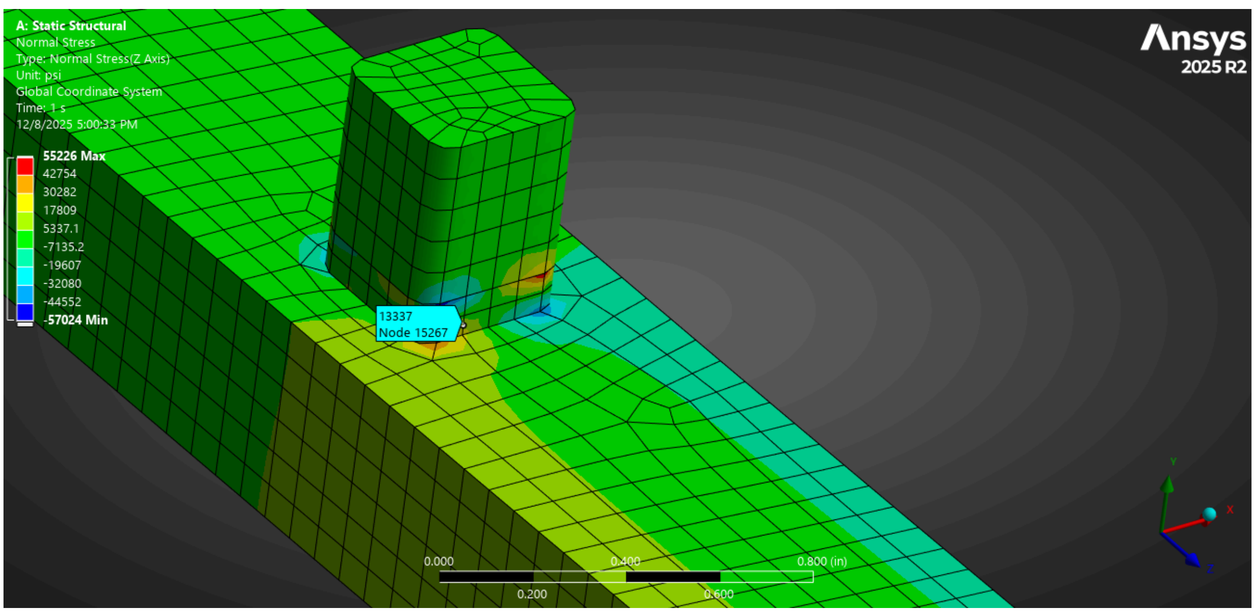


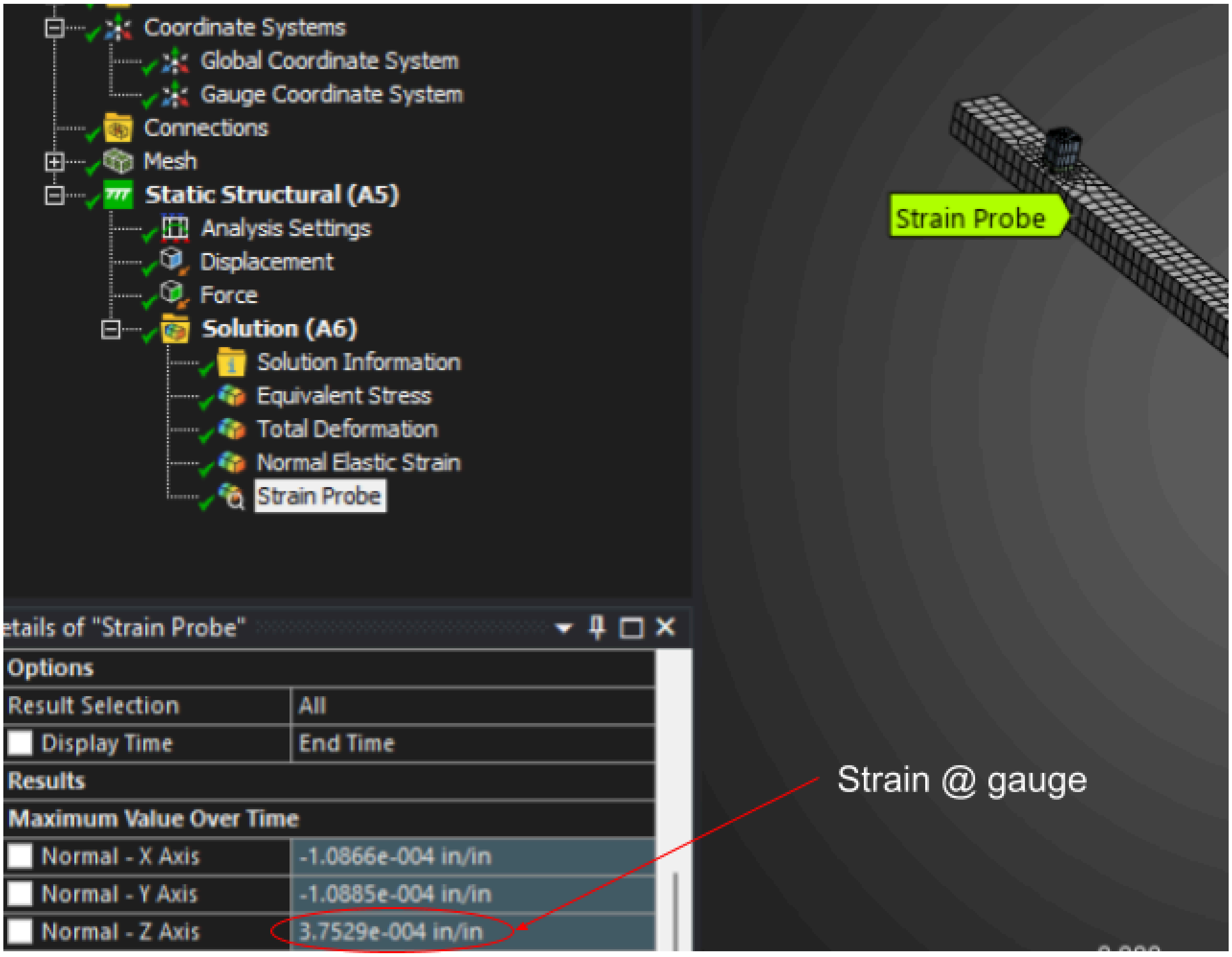
Image of FEM for Equivalent Stress results:



Maximum normal stress: 55226 psi

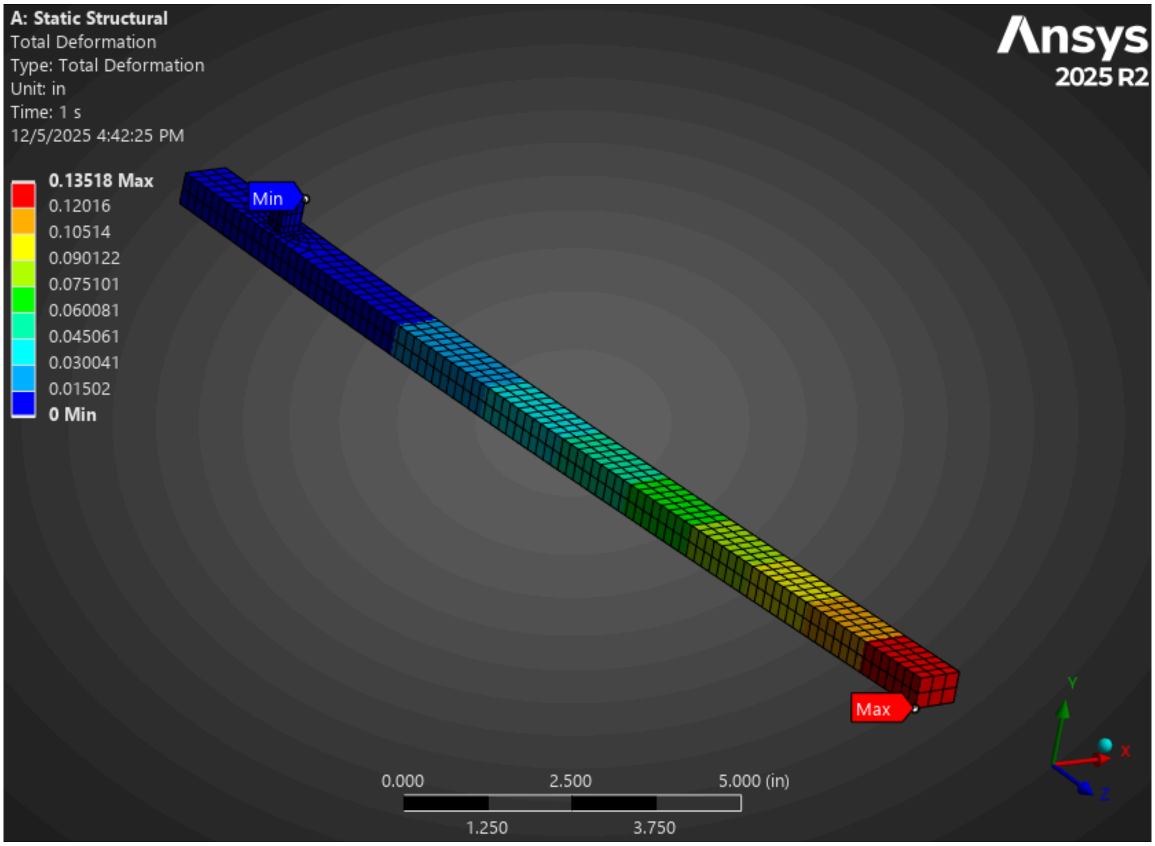
However, this normal stress was enlarged due to the boundary conditions of, thus choosing a node not along the boundary but near the expected location of maximum stress yields a sigma = 13.337 ksi.

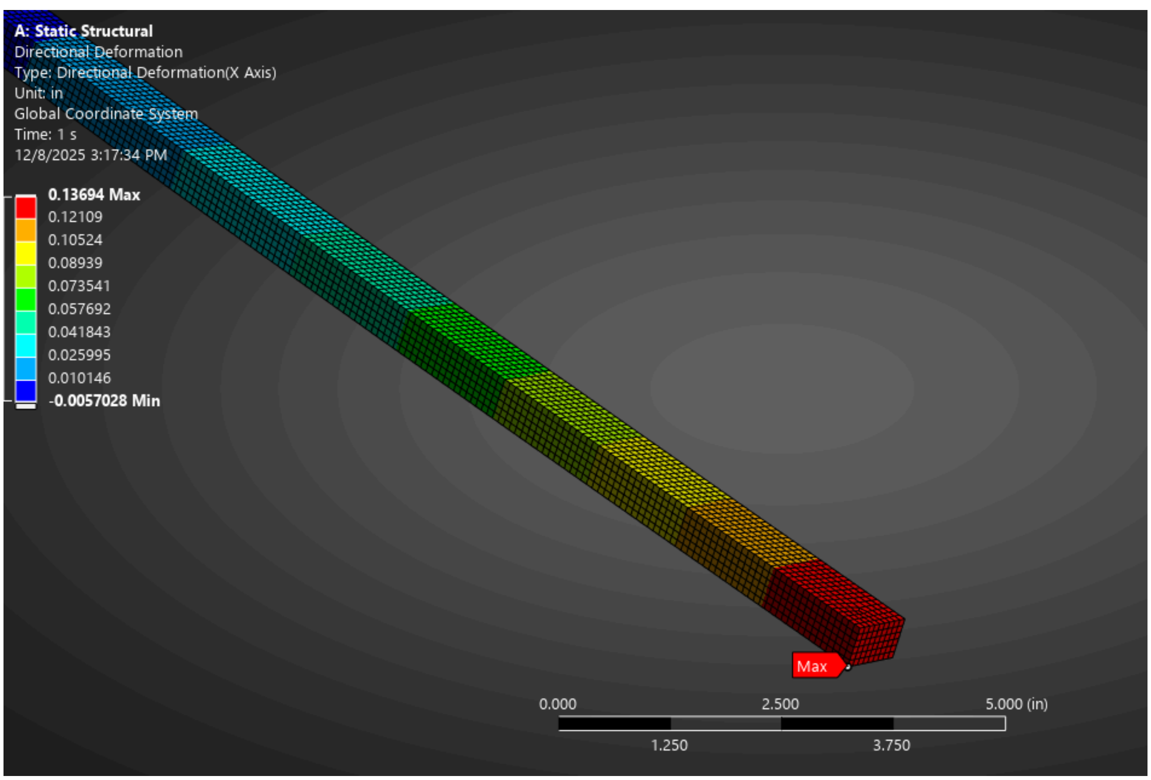
Image of FEM for Strain results:



Strain @ gauge location: 375.29 microstrain

Image of FEM for Deformation results: Total Deformation (Top), Directional (Bottom)





Deflection @ load point: 0.13518 in

2. Reflection

2.1) 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?

The plane sections didn't remain plane in our deformed mesh due to our mesh lines not remaining straight along the handle. In particular, the mesh lines warped near the driver connection and where the force is applied. We believe that beam theory does not completely hold as it does not account for shear and stress concentrations stemming from torsional effects and the way we defined our boundary and loading conditions.

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

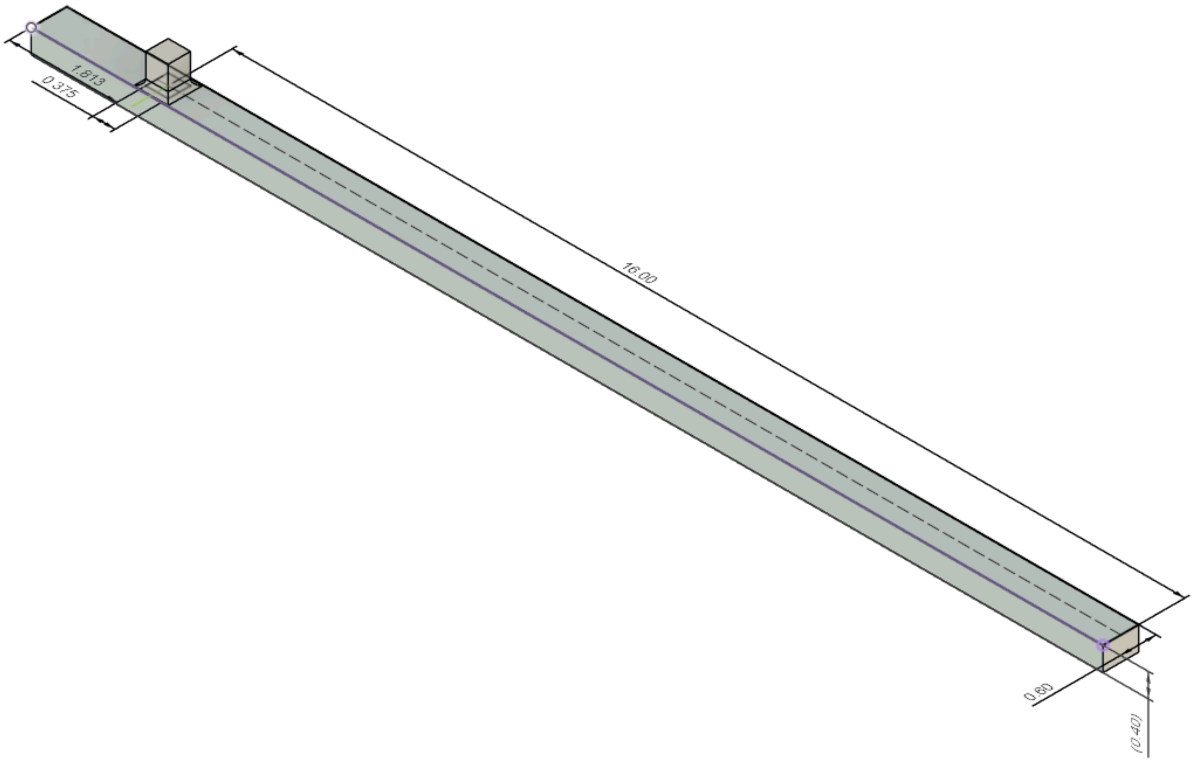
The FEM results do differ from the hand calculations for maximum normal stress. Our measured FEM max normal stress is 55.226 ksi. However, this normal stress was enlarged due to the boundary conditions of the driver and , thus choosing a node not along the boundary but near the expected location of maximum stress yields a sigma = 13.337 ksi. Our calculated maximum normal stress is 12.8 ksi. This is a pretty significant difference which we can attribute to stress concentration between the driver and the wrench of the beam, not filletting the design gave us a strong stress concentration at these boundary points that are not accounted for in our hand calculations.

2.3) How do the FEM and hand calculated displacements compare? If they differ, why?

They do differ in that the FEM displacement is .13518 in whereas the hand calculated displacement is .091 in, that is a percent difference of 33%. We can attribute this difference to the setup boundary condition creating a stress singularity along the driver connector and handle, affecting displacement along the x-axis.

Our Design Results

4.1 CAD



Hand Calculations and Outputs

```
%TIME TO DESIGN MY OWN

%1mV/V output at 600 inlbf

%X_Yield_strength = 4

%X_CrackGrowth = 2 is a = 0.04

%X_fatigue = 1.5

%steel, al, or titanium
```

```
a = 0.04; % crack length

M = 600; % max torque

c = 1.0; % distance from center of drive to center of strain gauge
```

```
% material properties
```

```
E = 17.E6; % Young's modulus psi

nu = 0.37; % Poisson's ratio

su = 132.E3; % tensile strength use yield or ultimate depending on material psi

KIC = 97.E3; % fracture toughness psi sqrtin

sfatigue = 69.e3; % fatigue strength from Granta for 10^6 cycles

name = 'Ti-6Al-4V' % Ti-6Al-4V aalpha beta annealed
```

```
%dimensional properties

L = 16; % length from drive to where load applied (inches)

h = .6; % width

b = .4; % thickness
```

```
%Stress and deflection analysis

I = (b * h^3) / 12;

sigma =  M*(h/2)/I;

sigma_norm_ksi = sigma / 1000 % Convert to ksi

load_point_deflection = (M * L^2) / (3 * E * I)

F = M/L;

Mgage = F*(L-c);

sigma_g=  Mgage*(h/2)/I;

k = 2;

epsilon1 = sigma_g / E;

epsilon2 = -epsilon1;

vout_vin = k/4 * 2*epsilon1; % THIS IS CORRECT formula value

strain = epsilon1/10^-3

voltageOutput = vout_vin/10^-3
```

```
Ki= 1.12*sigma*sqrt(pi*a);

X_y_strength = su / sigma

X_CrackGrowth = KIC/Ki

X_Fatigue = sfatigue / abs(sigma) % Safety factor for fatigue
```

Max normal stress: 25 ksi

Deflection at Load Point: 0.4183 in

Strain @ gauge: 1378.7 microstrain

Voltage Output = 1.3787 mV/V using halfbridge

Yield strength FOS: 5.28

Crack Growth FOS: 9.77

Fatigue FOS: 2.76

Results

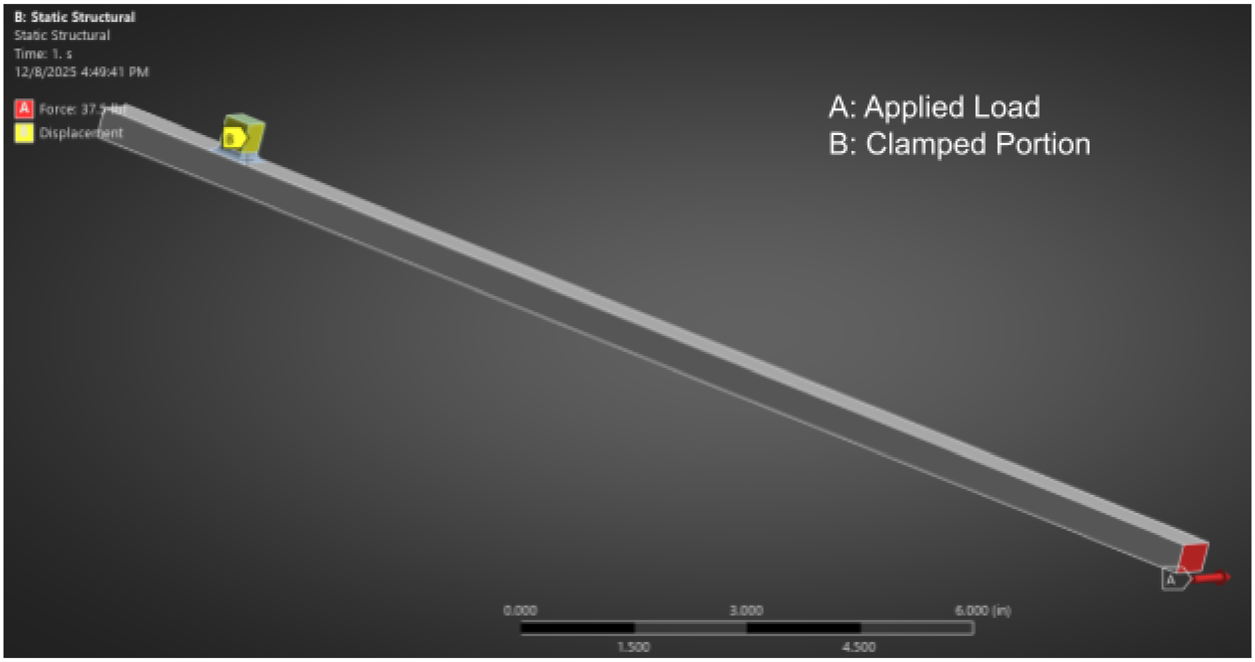
1. Image(s) of CAD model. Must show all key dimensions.

See 4.1 CAD above

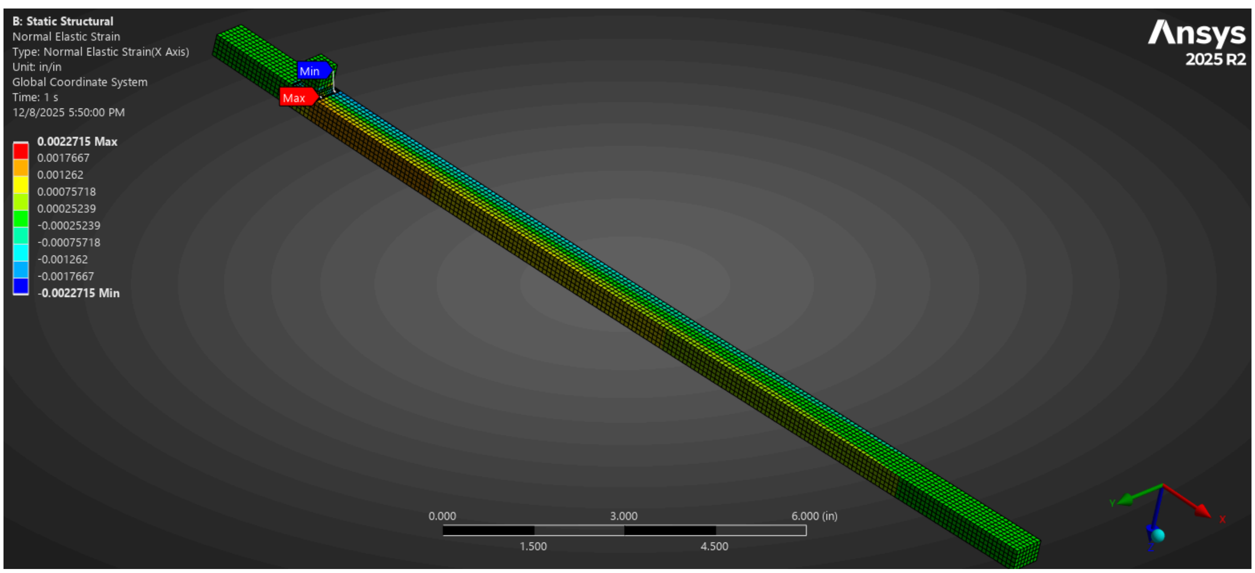
2. Describe material used and its relevant mechanical properties.

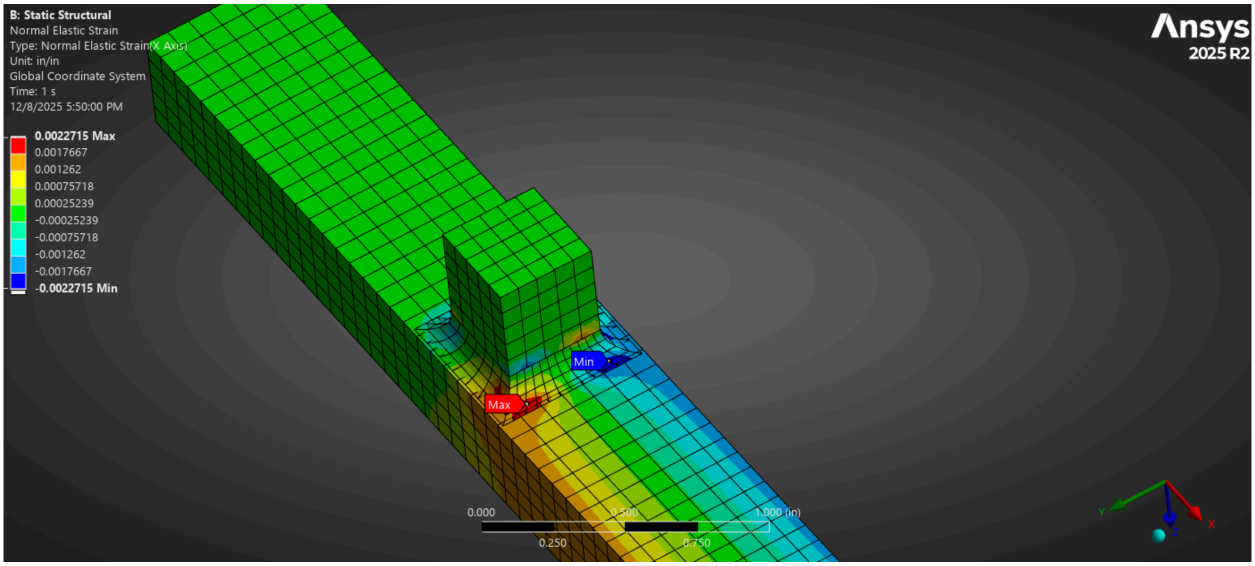
The material we chose was Ti-6Al-4V, a titanium alloy. We choose this material due to the strong material properties, such as a high tensile strength and high ductility.

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

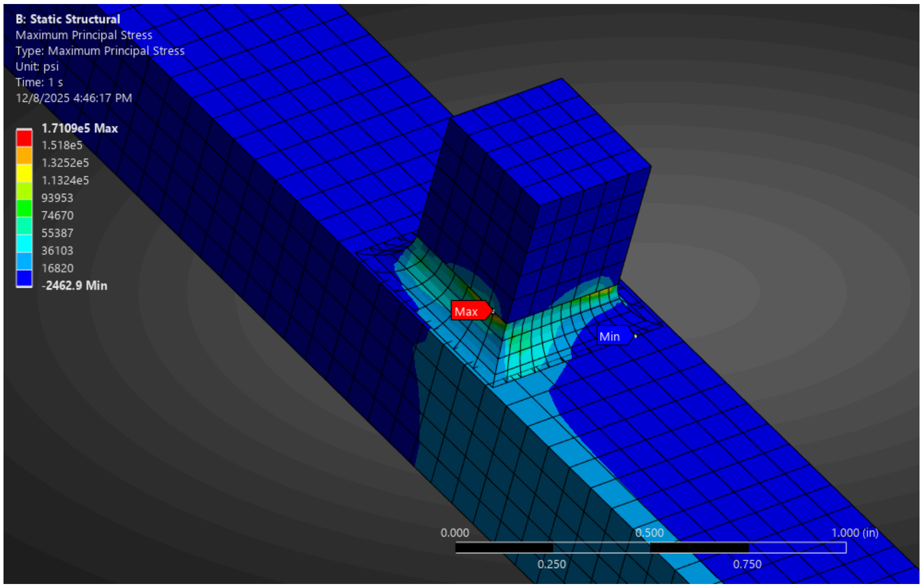


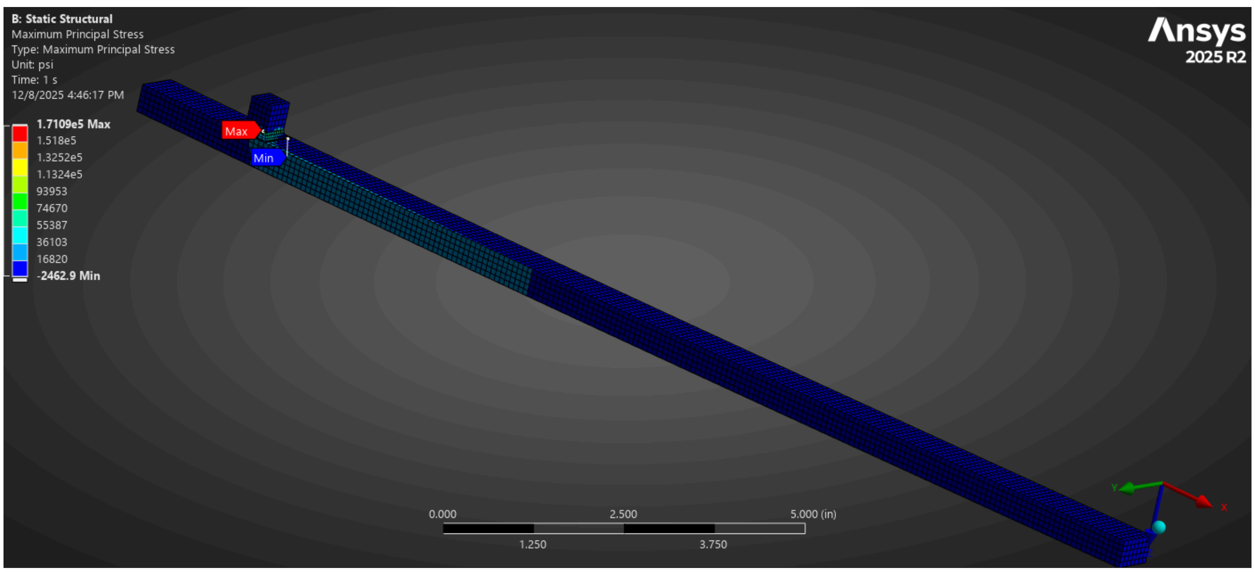
4. Normal strain contours (in the strain gauge direction) from FEM





5. Contour plot of maximum principal stress from FEM



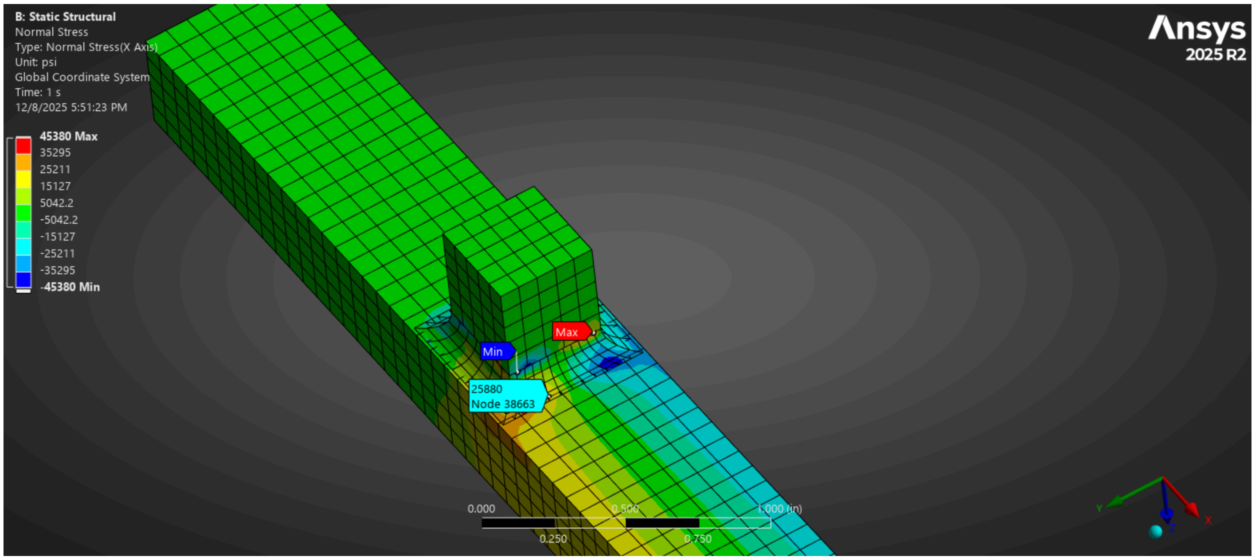


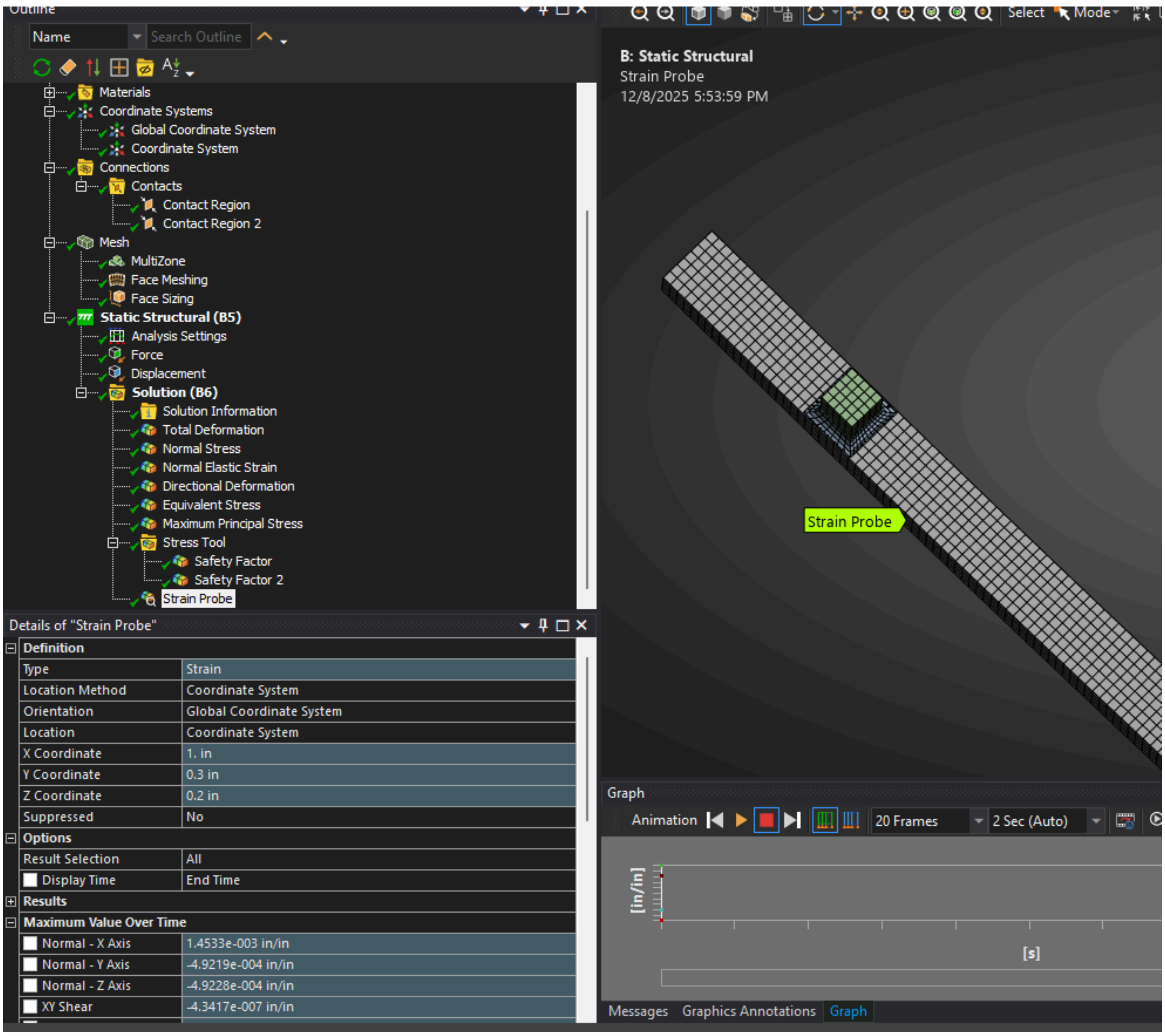
6. Summarize results from FEM calculation showing maximum normal stress (anywhere), load point deflection, strains at the strain gauge locations

Load Point Deflection: 0.503 in

Strain @ gauge location: 1.4533e-003 in/in

Max Normal Stress: 45.38 ksi


B: Static Structural
Normal Stress
Type: Normal Stress(X Axis)
Unit: psi
Global Coordinate System
Time: 1 s
12/8/2025 5:51:23 PM
45380 Max
35295
25211
15127
5042.2
-5042.2
-15127
-25211
-35295
-45380 Min
Ansys
2025 R2
Max
Min
25880
Node 38663
0.000
0.500
1.000 (in)
0.250
0.750


Outline
Name
Search Outline
Materials
Coordinate Systems
Global Coordinate System
Coordinate System
Connections
Contacts
Contact Region
Contact Region 2
Mesh
MultiZone
Face Meshing
Face Sizing
Static Structural (B5)
Analysis Settings
Force
Displacement
Solution (B6)
Solution Information
Total Deformation
Normal Stress
Normal Elastic Strain
Directional Deformation
Equivalent Stress
Maximum Principal Stress
Stress Tool
Safety Factor
Safety Factor 2
Strain Probe
Details of "Strain Probe"
Definition
Type | Strain
Location Method | Coordinate System
Orientation | Global Coordinate System
Location | Coordinate System
X Coordinate | 1. in
Y Coordinate | 0.3 in
Z Coordinate | 0.2 in
Suppressed | No
Options
Result Selection | All
Display Time | End Time
Results
Maximum Value Over Time
Normal - X Axis | 1.4533e-003 in/in
Normal - Y Axis | -4.9219e-004 in/in
Normal - Z Axis | -4.9228e-004 in/in
XY Shear | -4.3417e-007 in/in
Select
Mode
B: Static Structural
Strain Probe
12/8/2025 5:53:59 PM
Strain Probe
Graph
Animation
20 Frames
2 Sec (Auto)
[in/in]
[s]
Messages
Graphics Annotations
Graph

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

Sensitivity = (strain @ gauge)* 1000 = 1.4533 mV/V

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

Strain Gauge Selected: SGT-1LH/350-TY11
1.8 mm Grid Length, 5 mm Grid Width 350 Ω Resistance

Link:
https://www.dwyeromega.com/en-us/uniaxial-half-bridge-strain-gauges-with-transducer-quality/SGT-Half-Bridge-Uniaxial/p/SGT-1LH-350-TY11

It is sensitive enough that our strain max is 30,000 µm and has a carrier length 9.2 mm and a carrier width 4 mm, this fits in our spot as that is 0.36 in x 0.16 in.