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
```

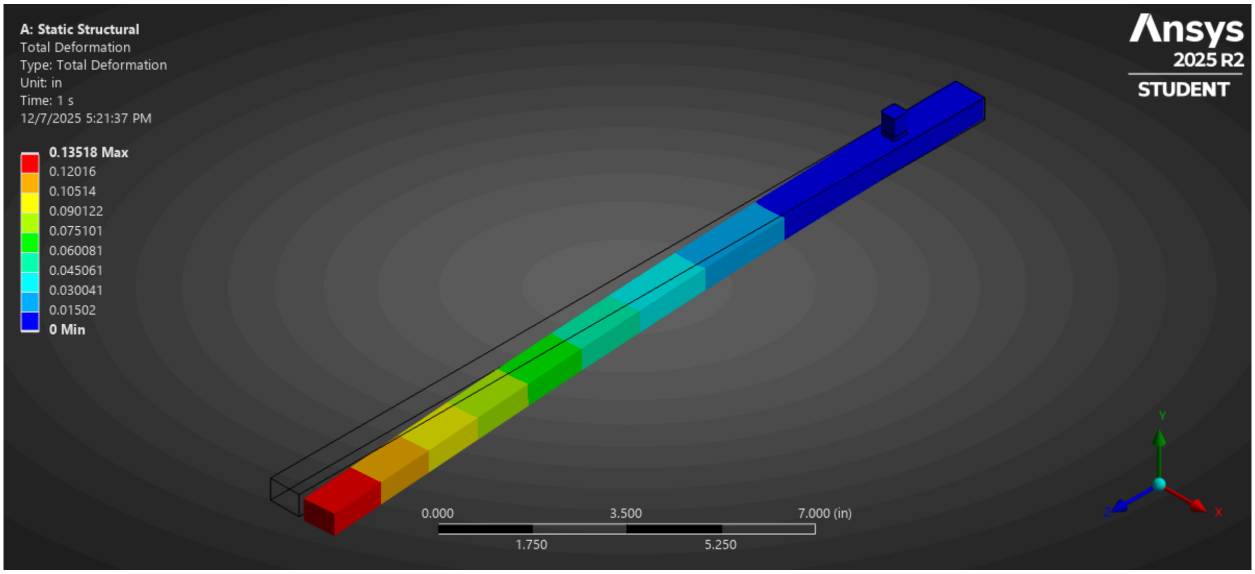
```
strainatgauge = 3.7500e-04

output = 0.3750
```

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

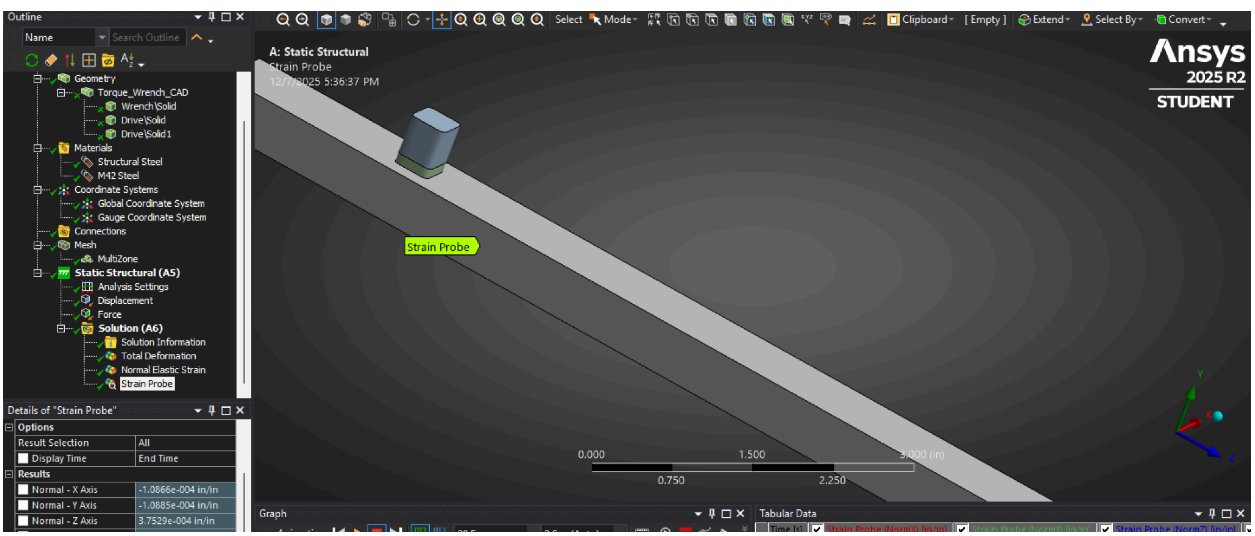
FEM Results:

Deflection of the load point:



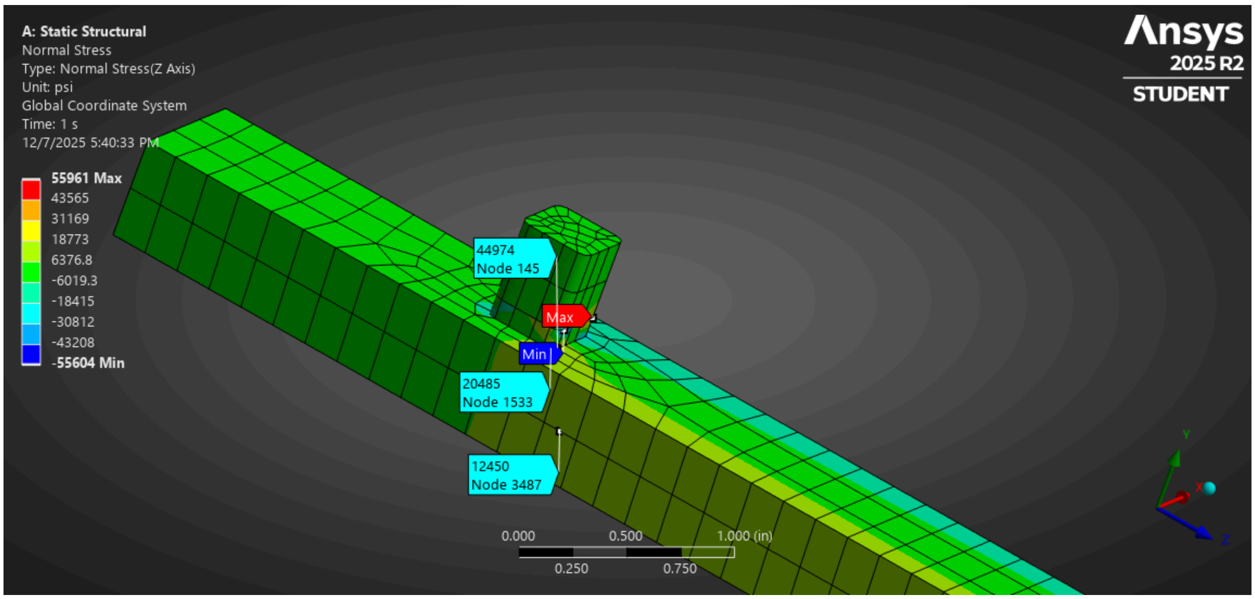
As can be seen in the screenshot above, the deflection of the load point calculated by Ansys is 0.13518 in. The hand calculations produced a value of 0.0910 in.

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>

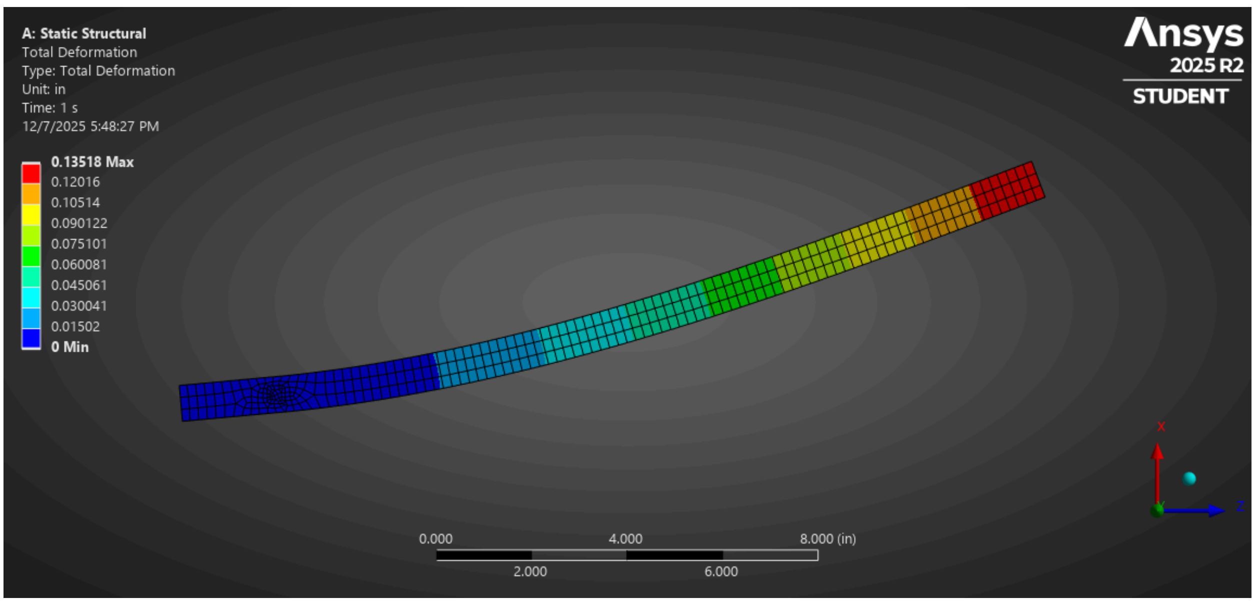


We see a maximum normal stress of around 55.961 ksi (from the legend on the right hand side) at the location that the clamped section of the drive meets the free section. We expect high stress concentrations here due to the sharp corners, however, our hand calculations did not take these stress concentrations 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.

Reflection Questions:

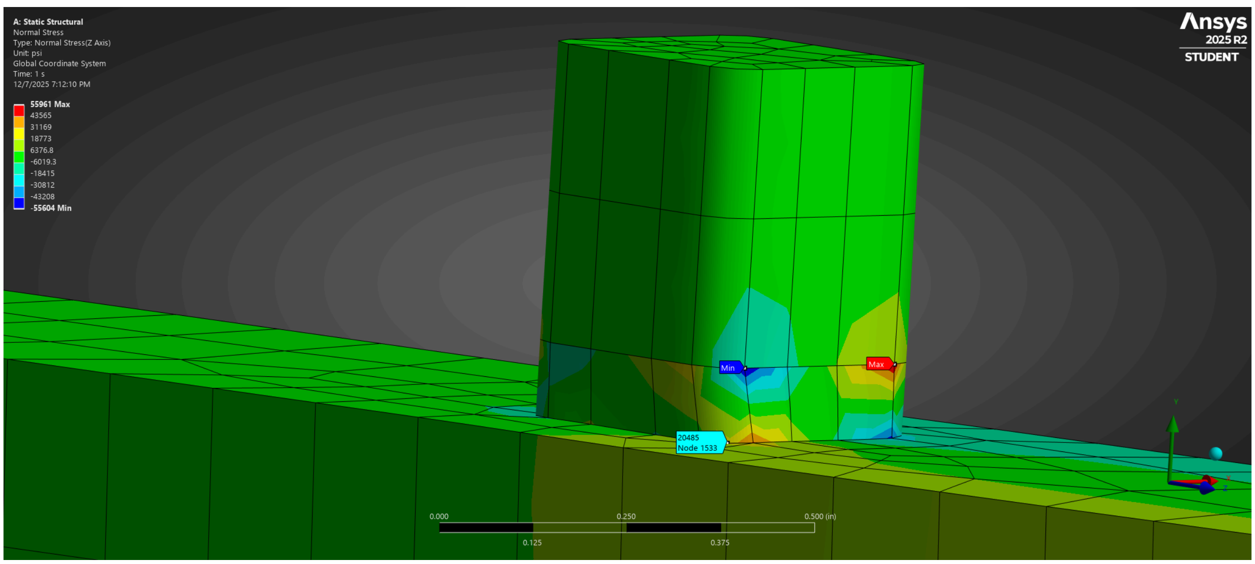
Validity of plane sections remaining plane:

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 is 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.



Comparison of the FEM and Hand Calculations for Maximum Normal Stress:

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.