MAE 3270 Final HW Part 1
Kat Volkova and Erik Salamanca
Thursday Dec 4 2025

MATLAB Code used for hand calculations

```
clear;clc; close all;
```

Enter Max Load Applied:

```
M = 600; % Applied Load in lbf-in
```

Enter Wrench Dimensions:

```
L = 15; % Length of wrench in inches
h = 0.8; % Height of wrench in inches
b = 0.5; % Width of wrench in inches
a = 0.04; % Crack depth
c = 1; % Strain gauge location
```

Enter Material Properties:

```
E = 10000000; % Young's Modulus psi
nu = 0.325; % Poissons Ratio
su = 72300; % Tensile or ultimate strength psi
KIC = 24200; % Fracture Toughness psi sqrt(in)
sfatigue = 26000; % Fatigue strength for 10^6 cycles in Granta
Material_Name = "Al 7075 T6"; % Material being tested
```

```
% Calculate commonly used variables based on properties
I = (b * h^3) / 12;
sigma_T = M * (h / 2) / I;
load_point_deflection = M * L^2 / (3 * E * I);
display("Deflection at load point is " + load_point_deflection + "
inches.");
disp("Max Stress is " + sigma_T * 10^-3 + " ksi.")
```

```
disp("Constraints for " + Material_Name + ".");
```

Calculate Strength Factor of Safety:

```
X_S = su / sigma_T;

display("Strength Factor of Safety is " + X_S + ".");

if X_S > 4

   disp("Strength Factor of Safety is adequate.");

else

   error("Strength Factor of Safety is not adequate, further analysis required.");

end
```

Calculate Facture Factor of Safety:

```
KI = 1.12 * sigma_T * sqrt(pi * a);

X_Frac = KIC / KI;

display("Fracture Factor of Safety is " + X_Frac + ".");

if X_Frac > 2

   disp("Fracture Factor of Safety is adequate.");

else

   error("Fracture Factor of Safety is not adequate, further analysis required.");

end
```

Calculate Fatigue Factor of Safety:

```
X_Fat = sfatigue / sigma_T;

display("Fatigue Factor of Safety is " + X_Fat + ".");

if X_Fat > 1.5

   disp("Fatigue Factor of Safety is adequate.");
```

```
else
   error("Fatigue Factor of Safety is not adequate, further analysis
required.");
end
```

Calculate Voltage Change:

```
epsilon = sigma_T / E;
voltage = epsilon * 10^3;
display("Strain Gauge value is " + epsilon * 10^6 + " microstrain.")
display("Voltage Output to input is " + voltage + "mV/V.")
if voltage > 1
   disp("Voltage is adequate.");
else
   error("Voltage is not adequate, further analysis required.");
end
```

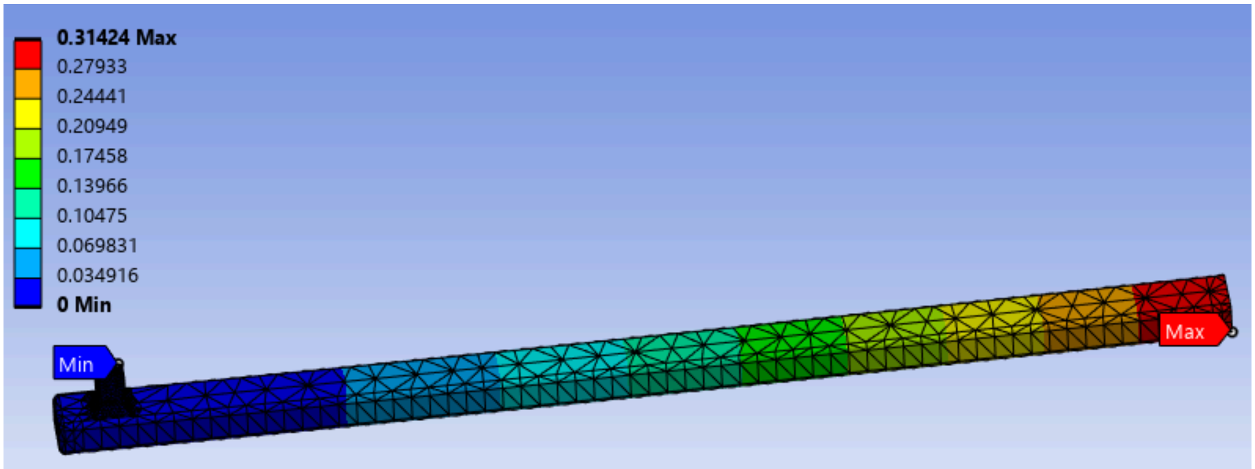
2. Results from hand calculation of base design showing maximum normal stress (anywhere), strains at the strain gauge locations and deflection of the load point.

Deflection at the load point is 0.21094 inches.
Max Stress is 11.25 ksi.
Strength Factor of Safety is 6.4267.
Fracture Factor of Safety is 5.418.
Fatigue Factor of Safety is 2.3111.
Strain Gauge value is 1125 microstrain.
Voltage Output to input is 1.125mV/V.

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.

<u>Deflection:</u>
Max deflection at the end is 0.31424 in

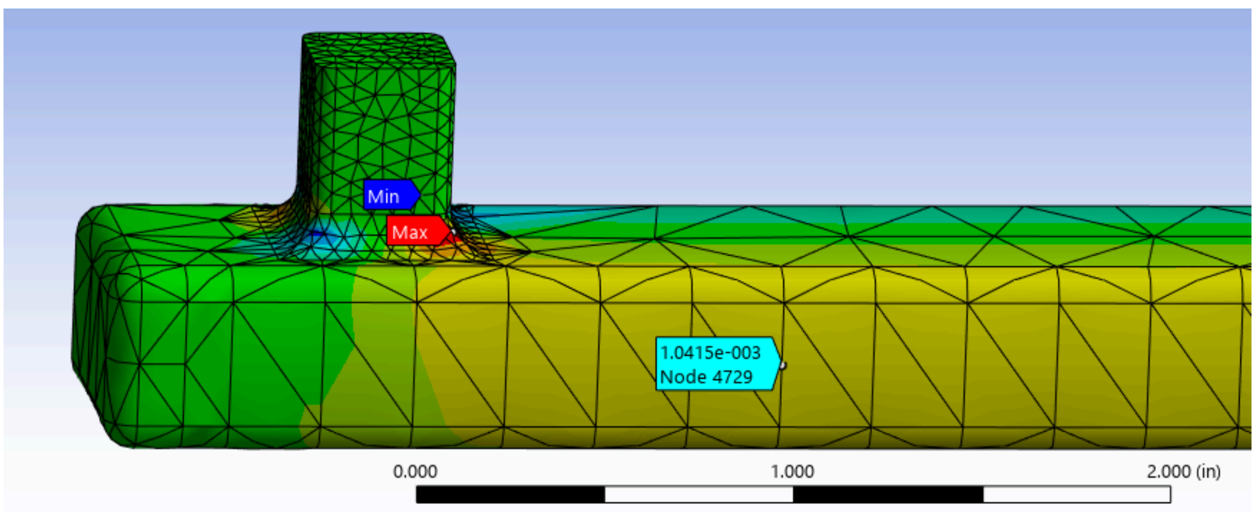
MAE 3270 Final HW Part 1
Kat Volkova and Erik Salamanca
Thursday Dec 4 2025

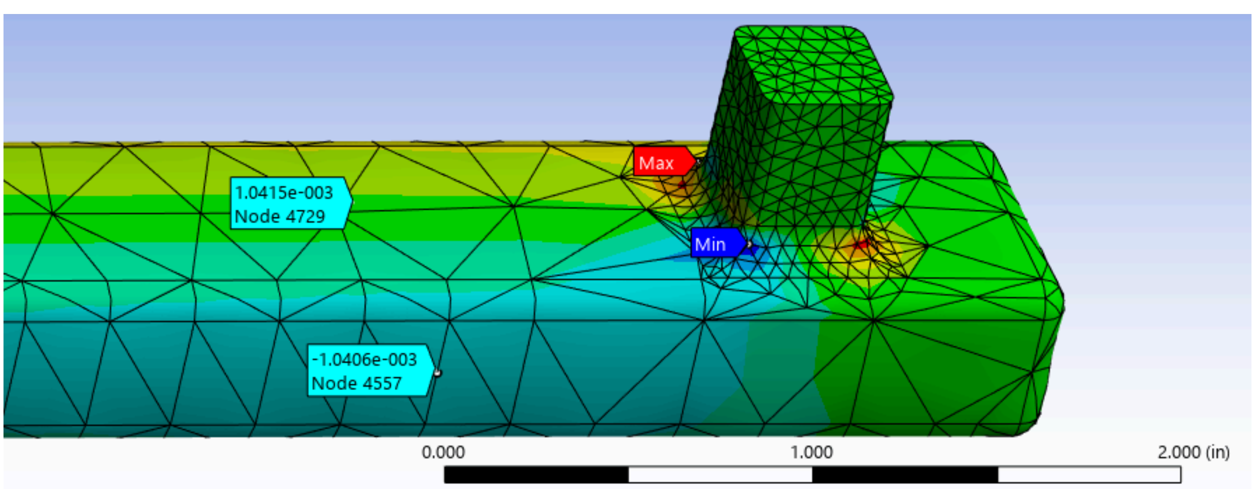


Percent difference to MATLAB calculations:

$$\frac{|0.21094 - 0.31424|}{\frac{0.21094 + 0.31424}{2}} * 100 = 32.87\%$$ Difference

<u>Strain at the strain gauge location</u>





MAE 3270 Final HW Part 1
Kat Volkova and Erik Salamanca
Thursday Dec 4 2025

Using the scale within ANSYS itself, we spot-checked the strain values at 1 inch away from the drive and saw that they matched the hand-calculated strain values very closely. As seen in the marked locations (cyan) in the pictures above, the strain gauges would measure strains of 1041.5 $\mu\varepsilon$ and -1040.6 $\mu\varepsilon$ respectively, which give percent differences of:

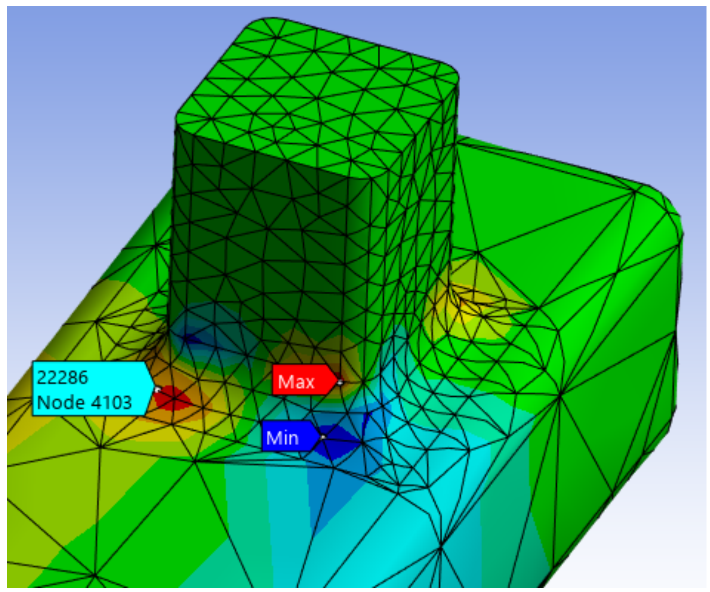
$$\frac{|1041.5 - 1125|}{\frac{1041.5 + 1125}{2}} * 100 = 7.71\% \text{ Difference}$$

$$\frac{|\,1040.6 - 1125|}{\frac{1040.6 + 1125}{2}} * 100 = 7.78\% \text{ Difference}$$

to the hand-calculated value of 1125 $\mu\varepsilon$. These percent differences are fairly low!

<u>Iterative Design for Max Normal Stress:</u>

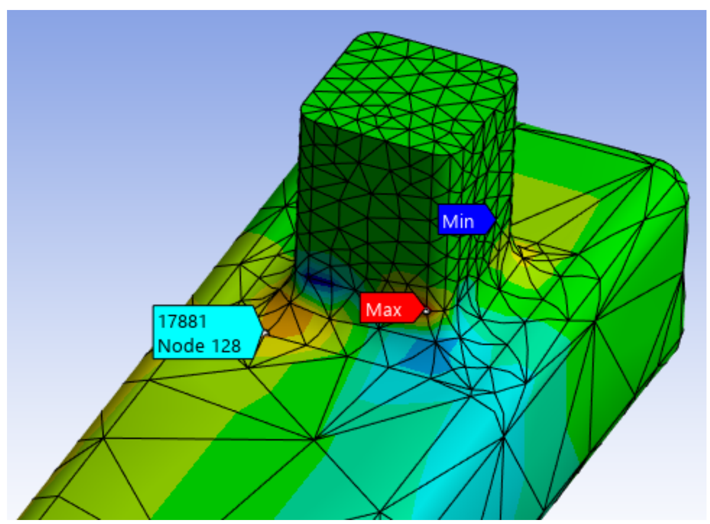
With a mesh size of 0.06 at the filleted, stress-concentration area:



This makes the factor of safety against yield, X_0 = 79000 psi/22286 psi = 3.54, which fails the design specifications.

Although for this project we did not need to meet the factor of safety for yield for this filleted area, we chose to change our mesh sizing to attempt to meet it anyway.

When reducing the mesh sizing to 0.1:



We can see that the max normal stress is about 17900 psi, which gives us a safety factor X_0 = 4.03, which just passes the design specs.

Note: The stress used here is the stress in the z direction. In reality, it would be better to use the equivalent stress (von Mises) to calculate the factor of safety.

Therefore, the final FEM calculation for the maximum stress is 17881 psi at the fillet due to the stress concentration.

MAE 3270 Final HW Part 1
Kat Volkova and Erik Salamanca
Thursday Dec 4 2025

Reflections

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

Beam theory is reasonably accurate because when looking at true scale deformation, all the mesh lines remain relatively straight. We believe that at a small enough scale, the deformed mesh sections are still plane, but as you “zoom out”, the deformation of each plane section adds up to bent lines.

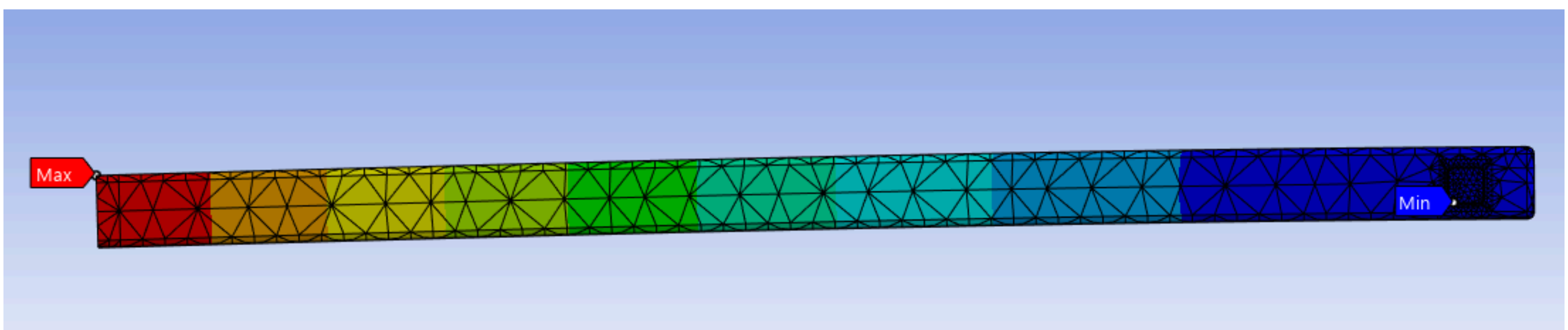
2. How do the FEM and hand-calculated maximum normal stresses compare? If they differ significantly, why?

The FEM maximum normal stress is higher than the hand-calculated value because it takes into account the fillet, which creates a stress concentration. This stress concentration significantly increases the maximum stress in the wrench, changing where the point of failure is located.

The advantage of using FEM is that it takes stress concentrations into account around the joint of the drive, which the hand calculations don't. This allows us to quantify a rudimentary stress concentration factor of this joint and fillet:

$$K \simeq \frac{17881\,psi}{11250\,psi} = 1.6$$

3. How do the FEM and hand-calculated displacements compare? If they differ, why?



As mentioned in the previous question, the percent difference between the hand-calculated and FEM-calculated displacements is 32.7%, which is a pretty significant difference. These values differ because the hand calculations do not take into account energy from shear stress. ANSYS does take into account this energy when calculating the deflection and gets a greater value due to the added energy.