MAE 3270, Fall 2025, Last HW, Design Project, Due Dec 8

This assignment pulls together several topics such as materials selection, fracture, fatigue, strain gauges, stress analysis and finite element analysis **This assignment counts as 3% of the course grade and cannot count as one your drops**. This will also be your portfolio assignment for 3270; do a good job so that you'll have something you can be proud of. We will integrate use of CAD and FEM tools along with hand calculations.

We will work on this together in lecture during the last three lecture meetings. You are encouraged, but not required to work in pairs. Everyone should submit their own assignment for their portfolio, noting who you worked with.

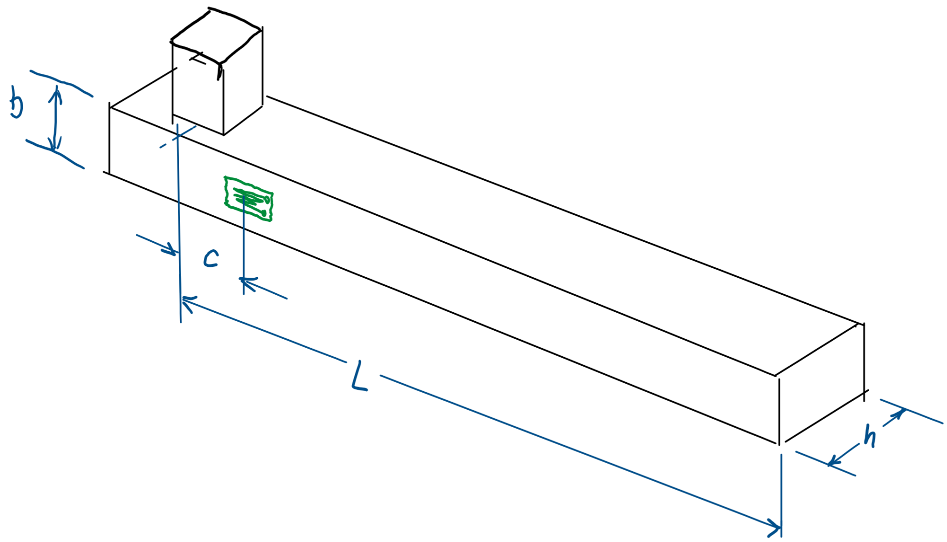

Figure 1: Schematic of torque wrench. Details of the drive dimensions are given in Figure 2. For first cut design you can assume that the load is applied at distance L from the drive. c is the distance from the drive to the strain gauge. The wrench handle itself is shown as rectangular with thickness b and width h. Your design need not be rectangular nor does it have to be of uniform shape and cross section. Note that actual wrench would be longer than L so that you there is room for the user to grip the wrench handle and apply the load.

1 Overview and Learning Goals

Your assignment is to design a non-ratcheting, 3/8 inch drive instrumented torque wrench rated for 600 in-lbf. A bare bones geometry is shown in the figure. Torque will be transduced using strain gauges bonded to the outer surfaces of the wrench at high strain locations. The basic analyses needed for the design can be performed by hand using your results from HWs 11, 12 and 13. I am also asking you to perform a finite element analysis of your final design.

The overall idea is to

- Learn to use ANSYS via an FEM analysis of a provided "baseline" design.

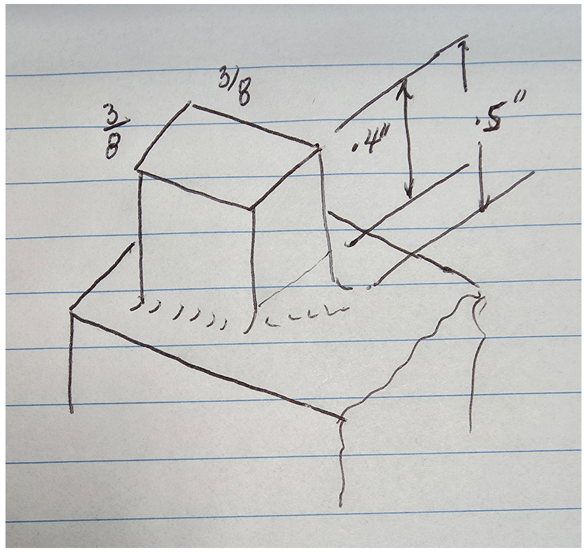

Figure 2: Details of drive dimensions. You can assume clamped boundary conditions in the upper 0.4 inches of the drive.

- Use hand, i.e. analytical calculations, to iterate the dimensions and material to come up with an improved design that meets all requirements
- Build a CAD model of your design. Your design need not be exactly what you came up with by hand calculations but the basic dimensions of the design should be guided by the hand calculations.
- Import the CAD model to ANSYS, perform the stress analysis, check that the design meets the requirements and determine the torque wrench sensitivity.

The learning goals are to

- Be able to perform first cut design considering multiple constraints.
- Be able to compare and understand results from FEM and hand calculations.

2 Design Using Hand Calculations

The torque will be transduced by strain gauges on the sides of the torque wrench. The design goal is to maximize the voltage output of the wrench (mV/V) at the rated torque. The design is required to attain at least 1.0 mV/V output at the rated torque of 600 in-lbf. Higher output will lead to more sensitivity and improved signal to noise ratio. The constraints are that the wrench must not fail due to static loading, crack growth or fatigue.

The wrench must sustain a fully reversed torque of $T = \pm 600$ in-lbf for 10^6 cycles. Design will include selecting an appropriate material and dimensions to meet or exceed the following requirements:

- attain at least 1.0 mV/V output at the rated torque of 600 in-lbf.

- safety factor of $X_o = 4$ for yield or brittle failure (you pick which criterion based on whether you are using a brittle or ductile material)
- safety factor of $X_K = 2$ for crack growth from an assumed crack of depth 0.04 inches (1 mm).
- fatigue stress safety factor of $X_S = 1.5$.
- material must be a steel, aluminum or titanium alloy.

I strongly suggest you write a Matlab or Python script to perform your hand calculation design iterations.

Below is *one* design. You can use these results to check your numerical calculations. Note that the torque wrench output is low and that the design does *not* meet the sensitivity specification. You can, and are required, to do better.

Load, dimensions and mechanical properties:

```
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
```

Stress and deflection analysis

```
load point deflection = 0.091 in
max normal stress = 12.80 ksi
```

Safety factor results:

```
safety factor for strength = 28.9
safety factor for crack growth = 2.95
safety factor for fatigue = 8.98
```

Strain gauge results:

```
strain at gauge = 375 microstrain
output = 0.38 mV/V at 600 in-lbf using half bridge
```

3 Analysis of Baseline Design

Following the example in lecture, import a model of the baseline design in Ansys and perform an FEM analysis under a torque of T = 600 in-lbf (lateral force of F = T/L). Use the results from this model to address the questions below which ask you to compare hand and FEM calculations.

4 Your Design

Using the code you developed for the hand calculations, iterate the design, including material and dimensions to come up with an improved design that meets all design requirements.

4.1 CAD

Build a CAD model based on your improved design. We recommend Autodesk Fusion for ease of integration with ANSYS.

4.2 FEM Analysis

Import your CAD model into ANSYS. (Instructions on how to do this will be provided.) Load with a torque of T = 600 in-lbf and solve the model. Perform at least one mesh refinement study, i.e. reduce the mesh size and re-run the analysis.

5 What to include in your HW and portfolio

Your submission will consist of two parts. A PDF file with the items below, submitted to Gradescope and a link to your portfolio submission, submitted via Canvas and the portfolio submission itself.

5.1 Baseline Design - PDF File Submitted to Gradescope

5.1.1 Results

1. Script used for hand calculation.
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.
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.

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

5.2 Your Design, Upload to portfolio

5.2.1 Results

1. Image(s) of CAD model. Must show all key dimensions.
2. Describe material used and its relevant mechanical properties.
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
7. Torque wrench sensitivity in mV/V using strains from the FEM analysis
8. Strain gauge selected (give type and dimensions). Note that design must physically have enough space to bond the gauges.

6 Notes and things to think about

- Your final design should mitigate stress concentrations to the extent you can
- You could consider making the handle narrower at the locations of the gauges. That may give higher sensitivity but will also reduce safety factors. You will need to make some tradeoffs.