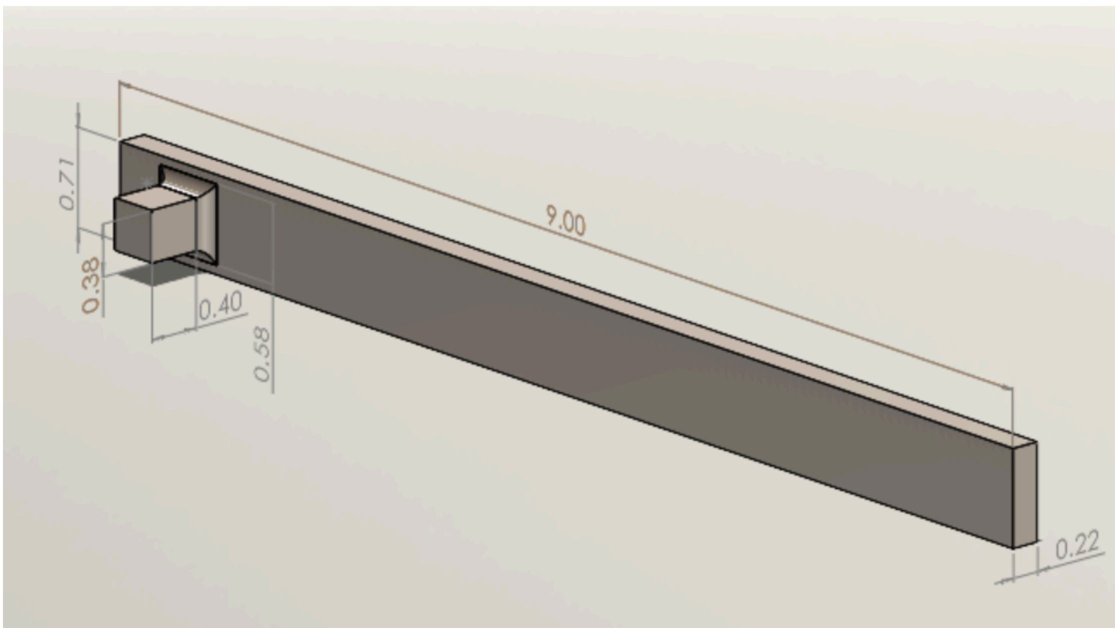
Navya Penati (ngp42) and Amanda Nicholson (agn39) Final Project Part II
5.2 Your Design, Upload to Portfolio
5.2.1 Results

1. Images of CAD model. Must show all key dimensions

   From the baseline design, our wrench has smaller handle dimensions as well as an additional fillet at the base of the drive. These dimensions are shown in Figure 1.

Figure 1: Wrench Design Dimensions



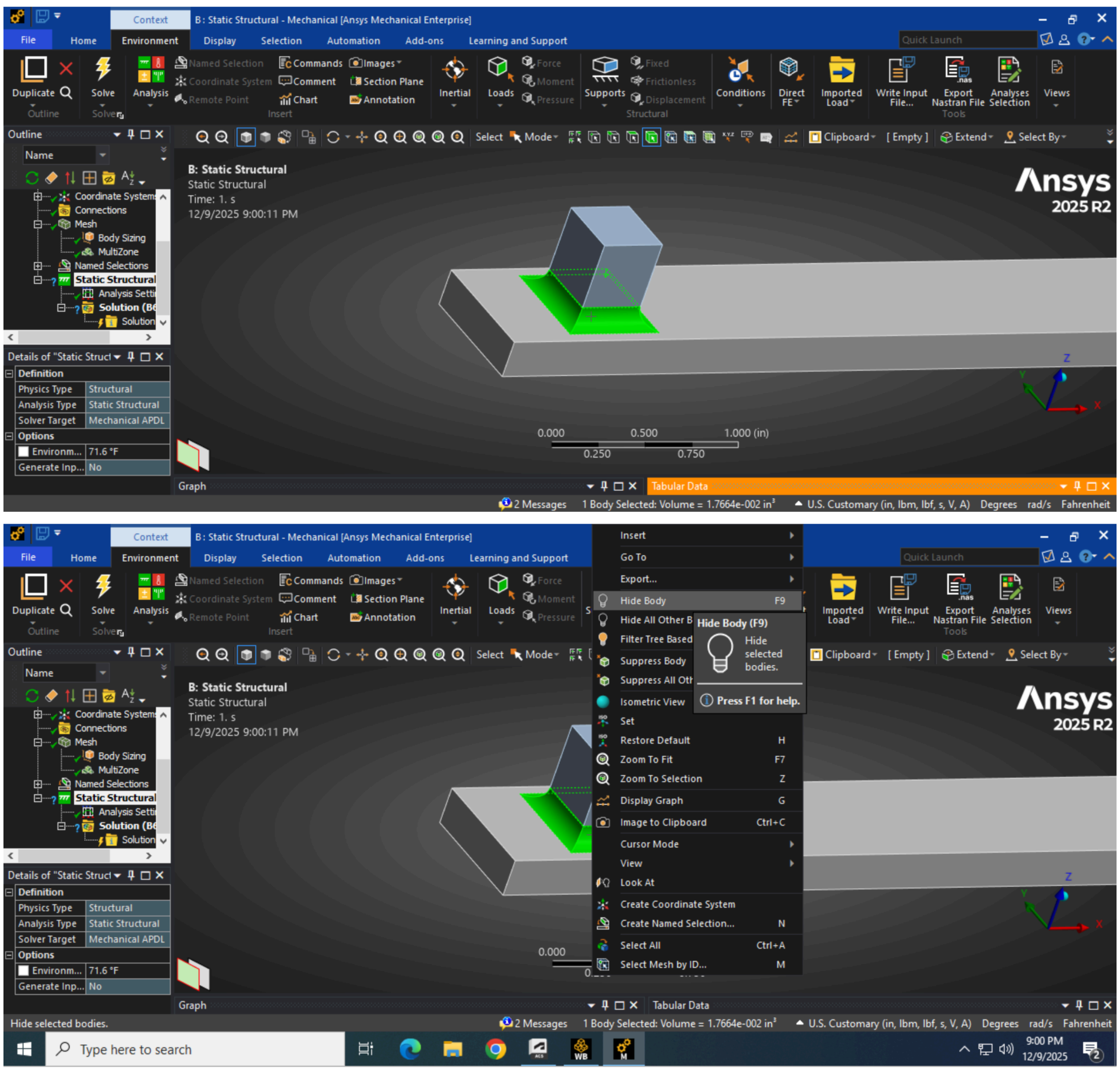
2. Describe material used and its relevant mechanical properties

   Our final design uses High Alloy Steel, AerMet 100 (solution treated and aged). This material was chosen because it is both strong and durable and allows the handle dimensions to be reduced for higher strain sensitivity while obtaining all required safety factors. Its mechanical properties are: Young's modulus = 28 x 10^6 psi, Poisson's ratio = 0.30, yield/tensile strength = 235 ksi, fracture toughness = 91,000 psi√(in), and fatigue strength at 10^6 cycles = 135 ksi. These properties ensure that the design meets and exceeds the requirements for yield, fracture, and fatigue safety margins.

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

In Ansys mechanical, the boundary conditions and loads can be applied in the static structural tab. Figure 2 shows the streamline process of adding the clamped boundary condition by hiding the fillet body, selecting the faces of the drive, and adding a displacement boundary of 0 at each face for each axial direction. Figure 3 shows the process of adding the load given 600 lb-in torque at the end of the handle. Note that the applied load is 600 lb-in/8 in = 75 lbf since the length of our handle from the center of the drive is 8in

Figure 2: Adding Clamped Boundary



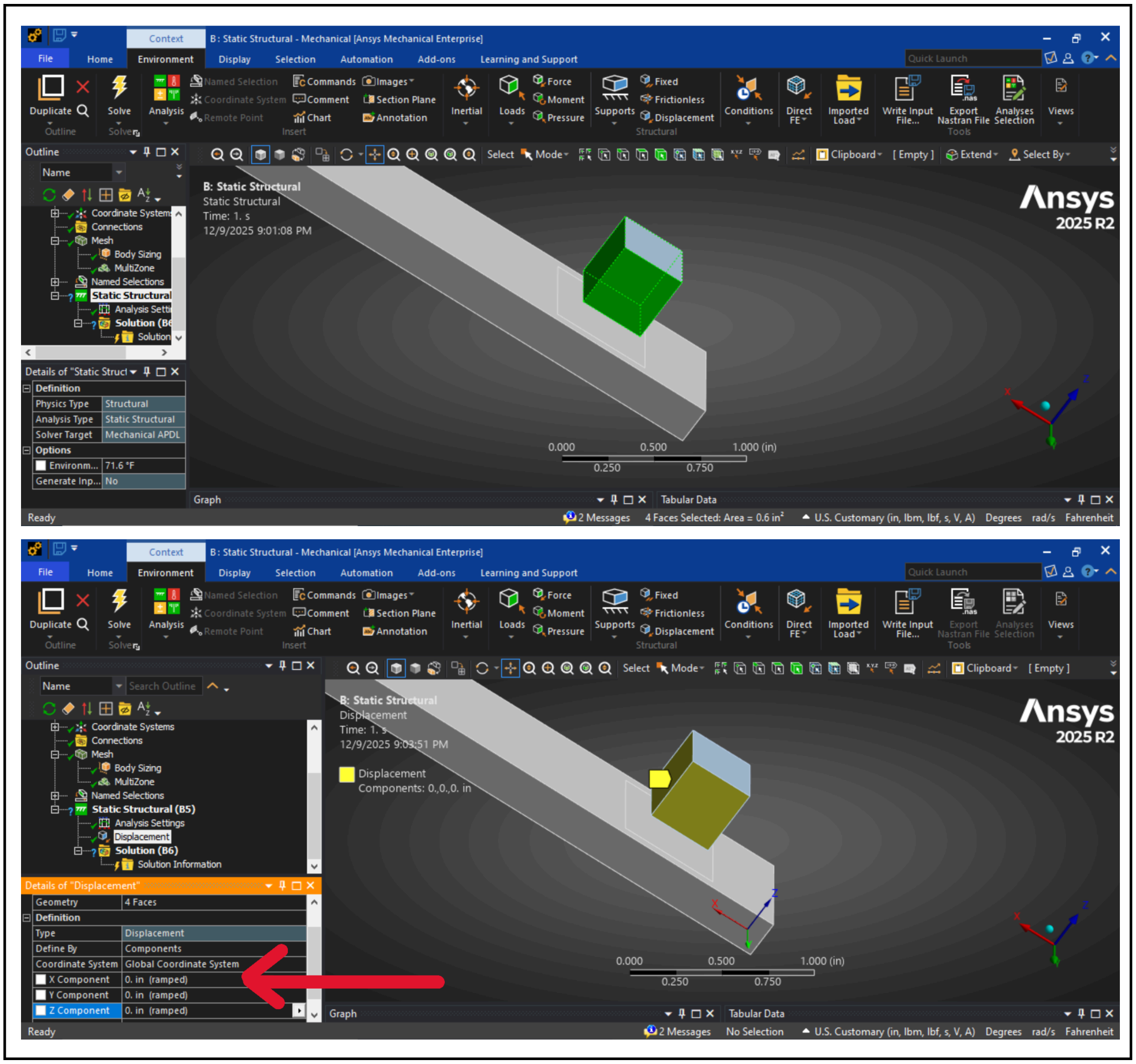
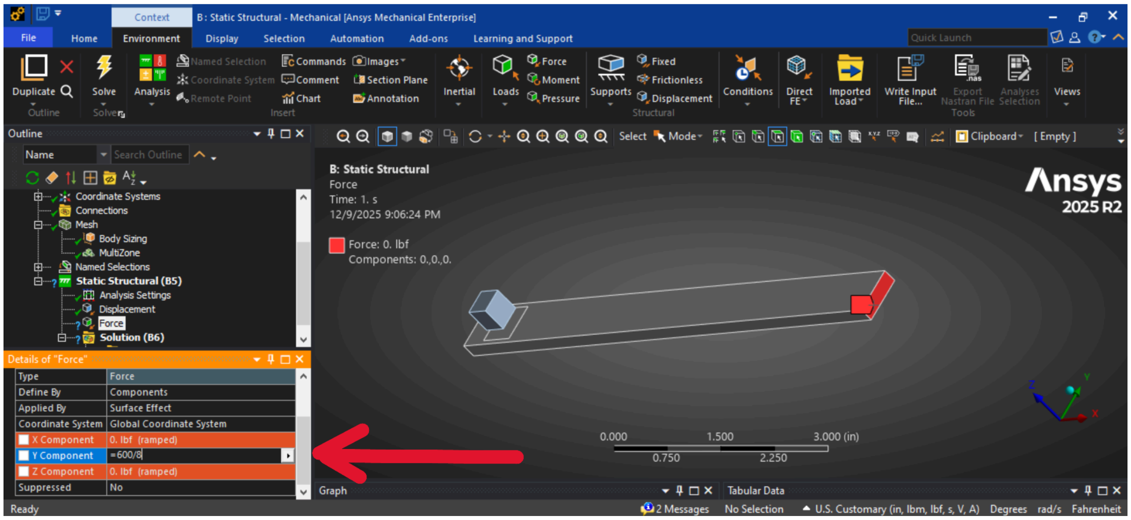

B: Static Structural - Mechanical [Ansys Mechanical Enterprise]
File
Home
Environment
Display
Selection
Automation
Add-ons
Learning and Support
Quick Launch
Duplicate
Solve
Analysis
Named Selection
Coordinate System
Remote Point
Commands
Comment
Chart
Images
Section Plane
Annotation
Inertial
Loads
Force
Moment
Pressure
Supports
Fixed
Frictionless
Displacement
Conditions
Direct FE
Imported Load
Write Input File...
Export Nastran File
Analyses Selection
Views
Outline
Solver
Insert
Structural
Tools
Name
Coordinate Systems
Connections
Mesh
Body Sizing
MultiZone
Named Selections
Static Structural
Analysis Settings
Solution
Solution Information
Details of "Static Structural"
Definition
Physics Type
Structural
Analysis Type
Static Structural
Solver Target
Mechanical APDL
Options
Environm...
71.6 °F
Generate Inp...
No
B: Static Structural
Static Structural
Time: 1. s
12/9/2025 9:01:08 PM
Ansys
2025 R2
0.000
0.500
1.000 (in)
0.250
0.750
Graph
Tabular Data
Ready
2 Messages
4 Faces Selected: Area = 0.6 in²
U.S. Customary (in, lbm, lbf, s, V, A)
Degrees
rad/s
Fahrenheit
Search Outline
Static Structural (B5)
Displacement
Solution (B6)
Details of "Displacement"
Geometry
4 Faces
Type
Define By
Components
Coordinate System
Global Coordinate System
X Component
0. in (ramped)
Y Component
0. in (ramped)
Z Component
0. in (ramped)
B: Static Structural
Displacement
Time: 1. s
12/9/2025 9:03:51 PM
Displacement
Components: 0.,0.,0. in
No Selection

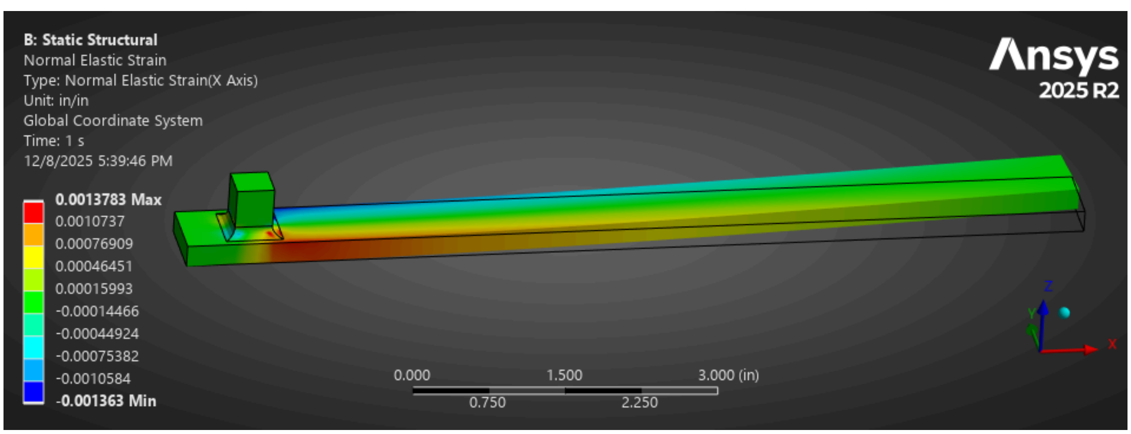
Figure 2: Adding Load to End of Handle



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

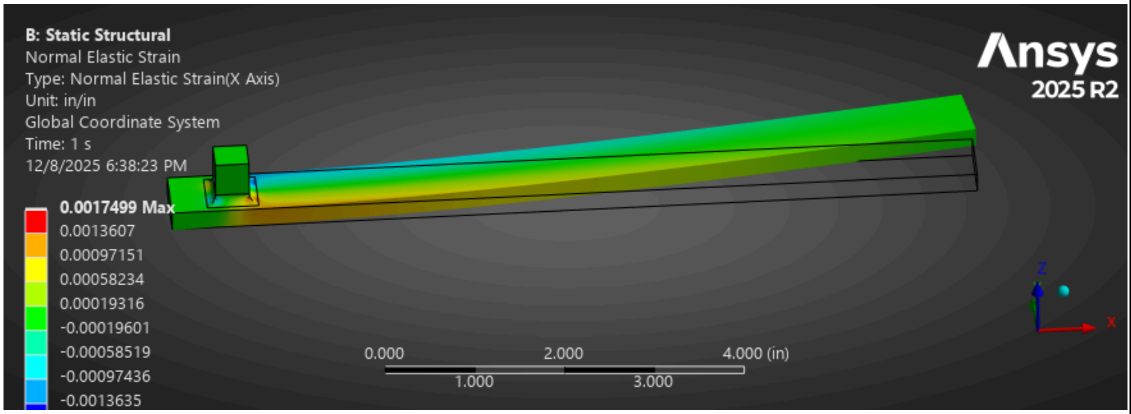
   The contour plot for normal strain shown in Figure 4 was obtained after using a mesh size of 0.25 in for the handle and 0.06 in for both of the drive components. Note that the direction of the strain gauge is along the x-axis (not the z-axis originally provided by the baseline design)

Figure 4: Normal Elastic Strain (X Axis) PRE Mesh Refinement



After refining the original mesh size to 0.1in and 0.02 in for the handle and drive components respectively, the contour plot for normal elastic strain was obtained and is shown in Figure 5.

Figure 5: Normal Elastic Strain (X Axis) POST Mesh Refinement

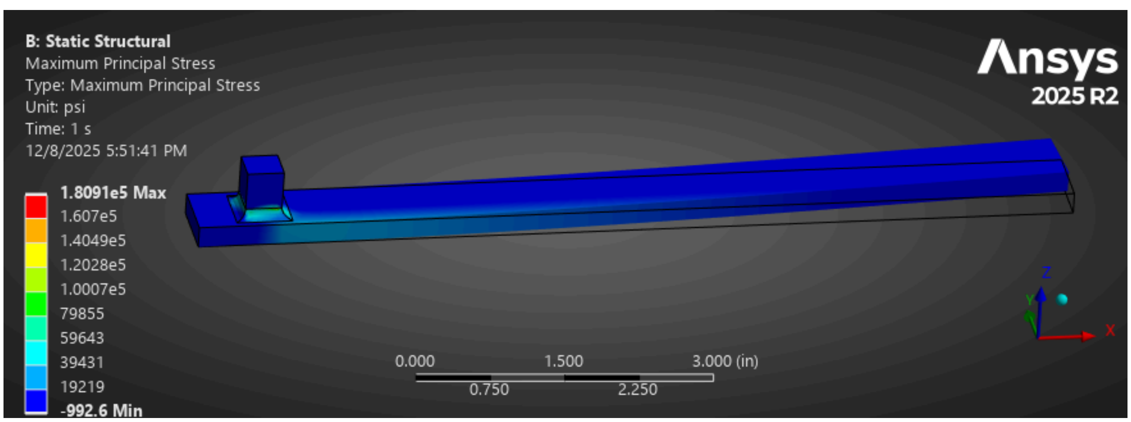


5. Contour plot of maximum principal stress for FEM

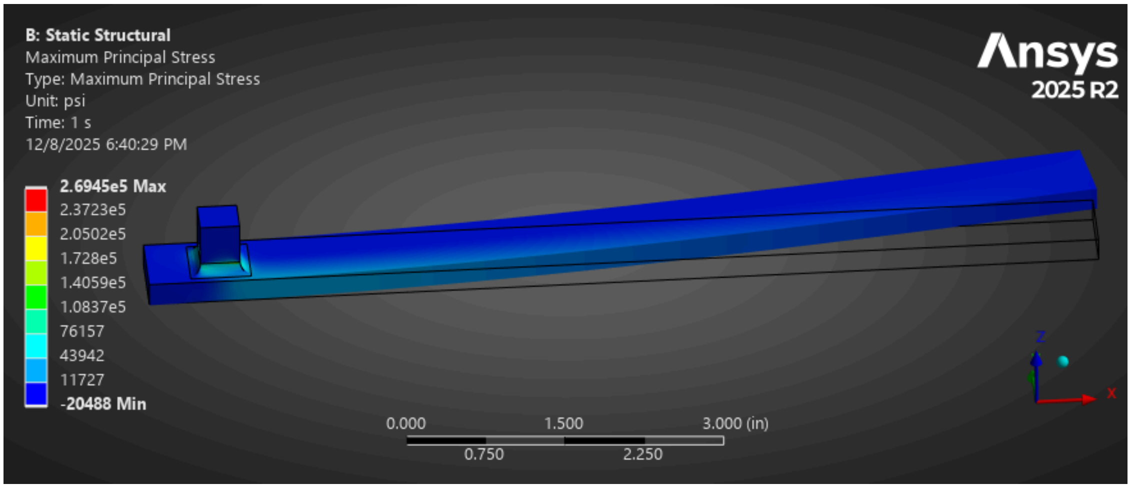
   The following are contour plots of the max principal stress, which is seen to be concentrated around the fillet of the drive as well as near the strain gauge location

Figure 6: Maximum Principal Stress

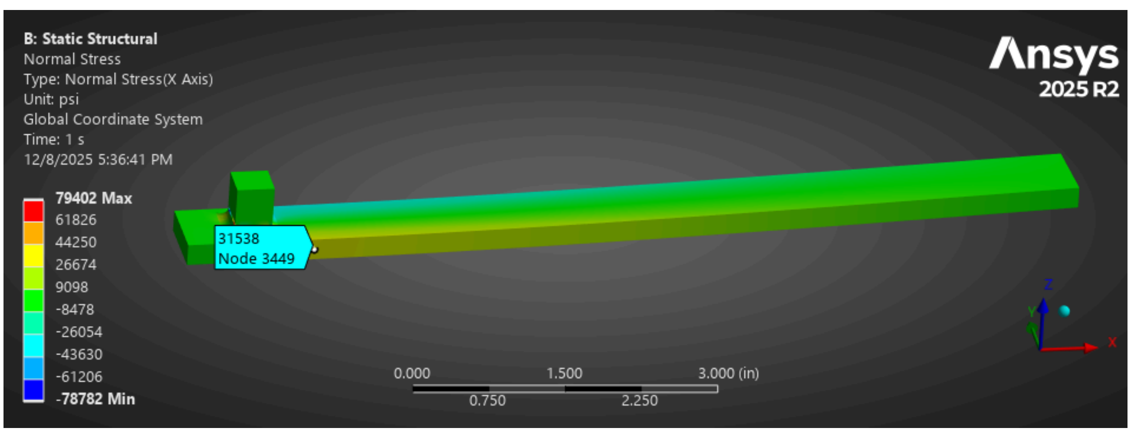
Pre Mesh Refinement



Post Mesh Refinement



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

Figure 7: Normal Stress
Pre Mesh Refinement

Post Mesh Refinement

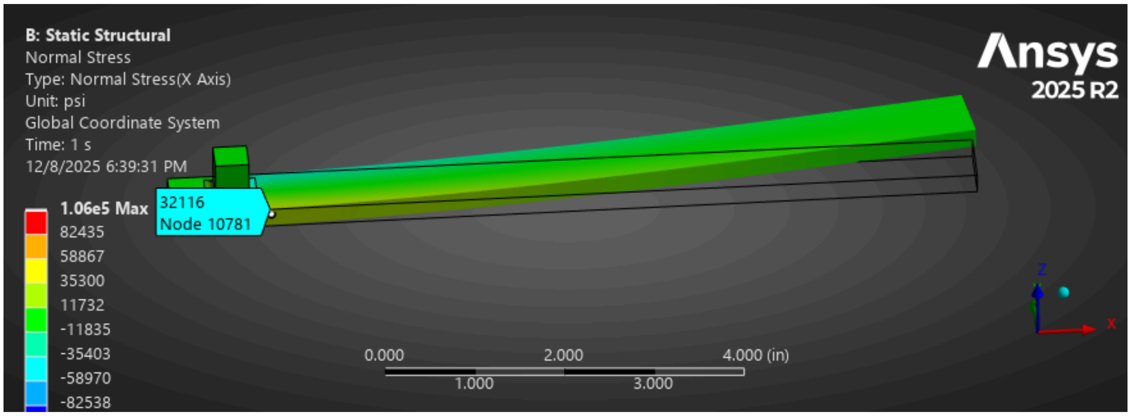


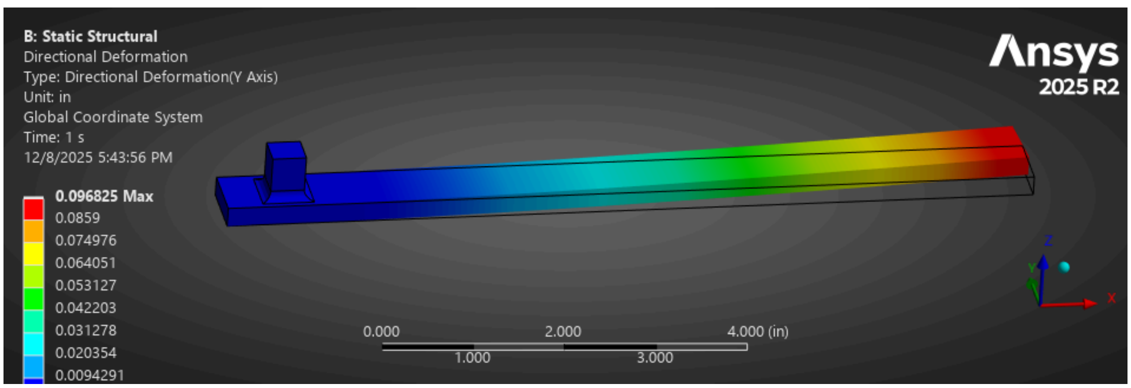
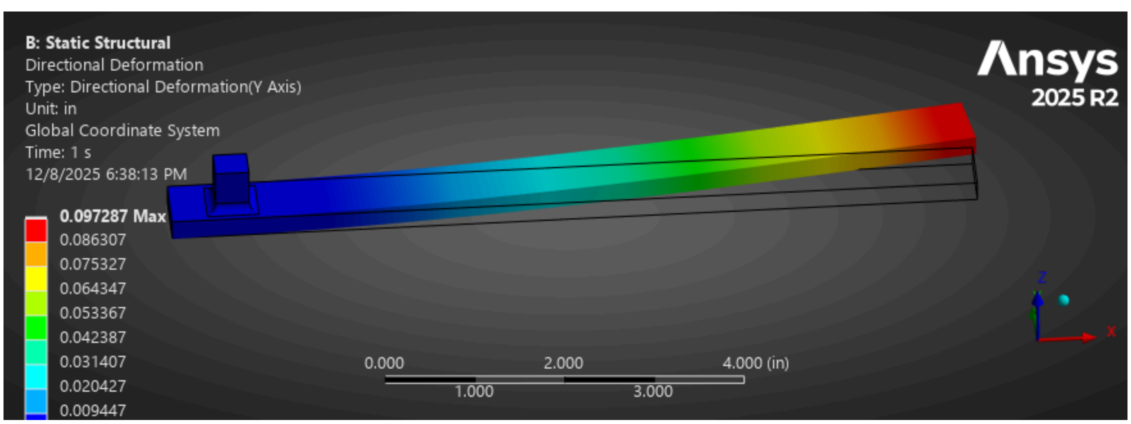
Figure 8: Load Point Deflection (Directional)
Pre Mesh Refinement

Post Mesh Refinement


Figure 9: Maximum Strain Values Over Time

Pre Mesh Refinement

Maximum Value Over Time	
Normal - X Axis	1.057e-003 in/in
Normal - Y Axis	-3.172e-004 in/in
Normal - Z Axis	-3.1707e-004 in/in
XY Shear	5.0506e-006 in/in
YZ Shear	3.6697e-007 in/in
XZ Shear	-2.8492e-007 in/in
Equivalent (von-Mises)	1.057e-003 in/in
Maximum Principal	1.057e-003 in/in
Middle Principal	-3.1694e-004 in/in
Minimum Principal	-3.1734e-004 in/in
Intensity	1.3743e-003 in/in

Post Mesh Refinement

Maximum Value Over Time	
Normal - X Axis	1.0563e-003 in/in
Normal - Y Axis	-3.1688e-004 in/in
Normal - Z Axis	-3.1689e-004 in/in
XY Shear	6.2583e-007 in/in
YZ Shear	6.1232e-008 in/in
XZ Shear	1.7109e-007 in/in
Equivalent (von-Mises)	1.0563e-003 in/in
Maximum Principal	1.0563e-003 in/in
Middle Principal	-3.1685e-004 in/in
Minimum Principal	-3.1692e-004 in/in
Intensity	1.3732e-003 in/in

The FEM analysis produced three important results for testing and evaluating the performance of the final torque wrench design. As seen in Figure 7, the maximum normal stress for the model was significantly high for both versions of mesh in comparison to the hand calculations (79,402 ksi and $1.06e^5$ ksi pre and post mesh refinement respectively). These stresses were concentrated around the drive/fillet of the drive. After probing around the handle near the gauge location, we observed a normal stress of approximately 32 ksi which was the max stress predicted by our hand calculations. The load-point deflection under the 600 in-lbf torque was 0.0973 in (Figure 8), which is slightly more than the 0.0694 in predicted by our calculations. This shows that the handle was stiff but still allowed bending strain. Probing for the strain at the center of the strain gauge yields the results shown in Figure 9. Given that our gauge length is oriented in the x-axis, the strain at the gauge location was observed as 1,057 microstrain, which is better than expected when it comes to achieving the required electrical output.

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

Further probing of strain at the point of the center of the strain gauge on the handle allows us to see the exact strain at the location of the gauge. The details for maximum values of strain at the probed location are shown in Figure 5 for both before and after mesh refinement. Note that the strain gauge is oriented along the x-axis given the wrench geometry in Ansys.

Given the values of the Normal -X Axis max strain over time, we can calculate the sensitivity of our wrench using the output of a half-bridge strain gauge:

<u>Pre mesh refinement:</u>

Gauge output $= 2k\varepsilon/4 = \varepsilon \times 10^3\ mV/V = 1.057 mV/V$

Post mesh refinement:

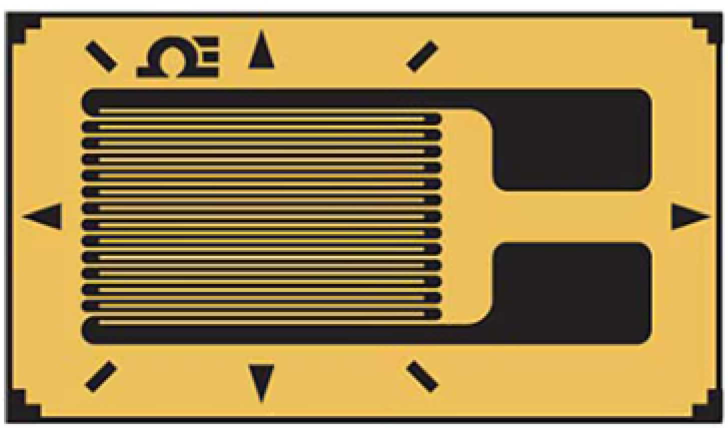
Gauge output $= 2k\varepsilon/4 = \varepsilon \times 10^{3}\, mV/V = 1.0563\, mV/V$

Both versions of the mesh produce an output/sensitivity that meets the design requirements (> 1mV/V).

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

   We decided to use two SGD-6/120-LY11 strain gauges in a half bridge configuration with one gauge on the side of the handle in tension and the other gauge on the side of the handle in compression. This gauge is 0.449 inches in total length, 0.201 inches in total width, and has an active gauge area of 0.0312 in^2. We chose this strain gauge because it is flat and fits the dimensions of our handle thickness. Figure 10 shows the layout of the gauge.

Figure 10: SGD-6/120-LY11 Strain Gauge Selection



Note: Image and dimensions were obtained from the vendor site: https://www.dwyeromega.com/en-us/linear-strain-gages/p/SGD-LINEAR1-AXIS#

Additional: Raw Manual Calculations Output

~~~~~~ MATERIAL CHOICE: High alloy steel, AerMet 100, solution treated & aged ~~~~~~

Material properties:
Young's modulus (psi): 28000000
Poisson's ratio: 0.3
tensile strength (or yield strength) (psi): 235000
fracture toughness (psi sqrt(in)): 91000
fatigue strength for 10^6 cycles (psi): 135000
price per unit volume (USD/in^3): 4.8032407407407405
density (lb/in^3): 0.286

Dimensions:
length: L = 8 in
width: h = 0.71 in
thickness: b = 0.22 in
volume: V = 1.3199125 in^3

price: 6.339857494212962 USD
mass: 0.37749497499999995 lb

~~~~~~ OUTPUTS FOR High alloy steel, AerMet 100, solution treated & aged ~~~~~~

Stress and deflection analysis:
load point deflection: 0.06966833915432871 in
max normal stress: 32.46109177472003 ksi

factor of safety results:
factor of safety for strength: 7.23943611111111
requirement satisfied.

factor of safety for crack growth: 7.060822843142121
requirement satisfied.

factor of safety for fatigue: 4.158824999999999
requirement satisfied.

strain gauge results:
strain at gauge: 1014.4091179600009 microstrain
output (600 in-lbf using half-bridge): 1.014409117960001 mV/V
requirement satisfied.