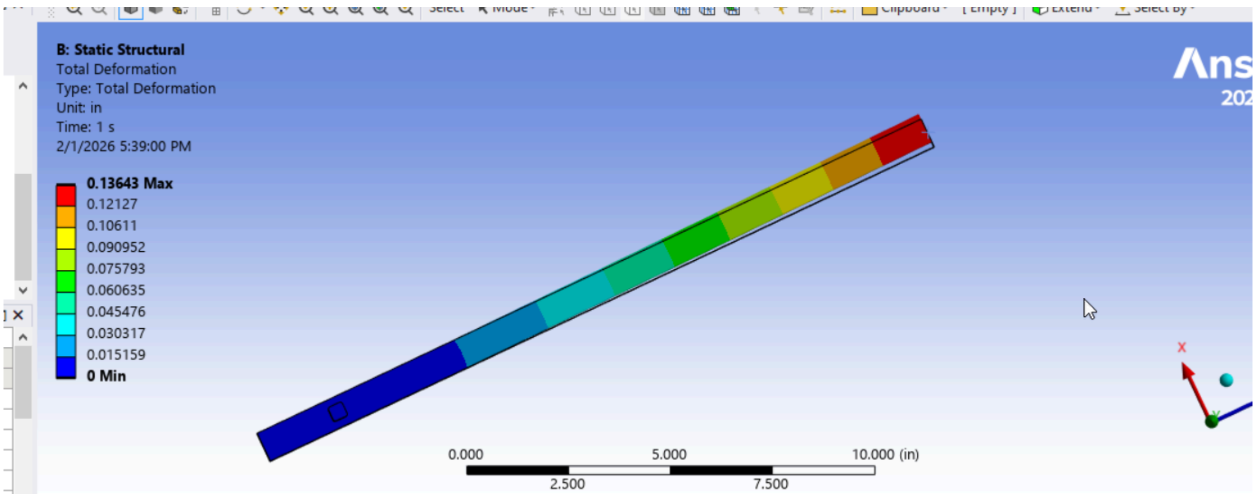
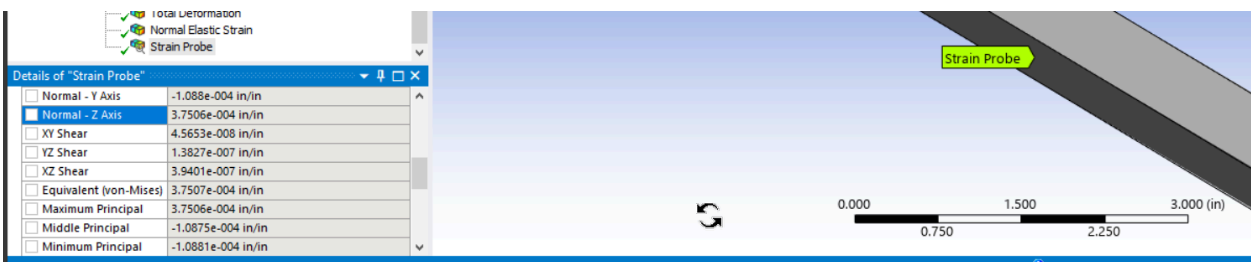
Final hw 3270

Baseline

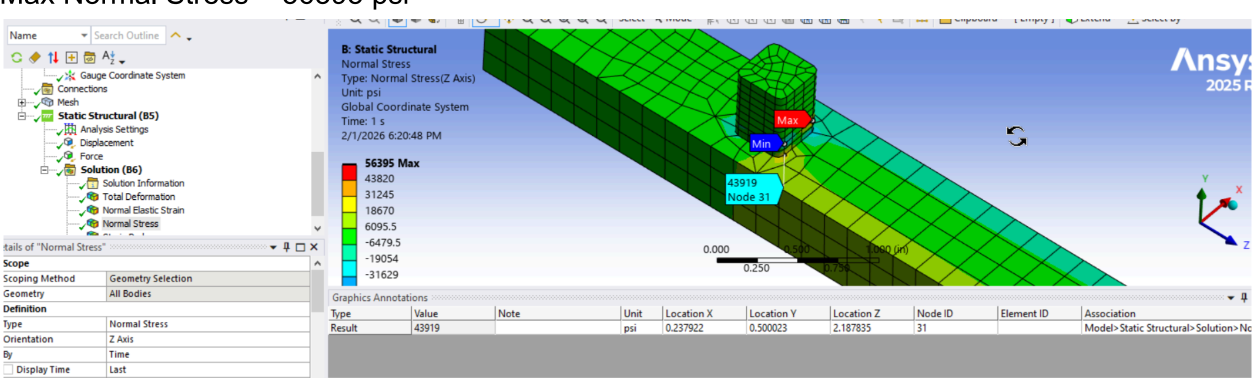
Max displacement = 0.13643



Normal strain at gauge = 375 microstrain



Max Normal Stress = 56395 psi



5.2.1 Results 1. Image(s) of CAD model. Must show all key dimensions.
2. Describe material used and its relevant mechanical properties.
The torque wrench was designed using the titanium alloy Ti-6Al-4V, which is a commonly used aerospace and structural alloy due to its high strength-to-weight ratio, good fatigue performance, and corrosion resistance. Ti-6Al-4V behaves as a ductile material, so yield strength was used as the governing failure criterion.

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

The finite element model represents a simplified non-ratcheting torque wrench subjected to an applied torque at the handle.

- Boundary conditions:
  The square drive end of the wrench was modeled as fully clamped over the upper 0.4 inches of the drive. All translational and rotational degrees of freedom were constrained to simulate engagement with a rigid socket and fastener.

- Loading:
  A point load applied at the end of the handle was used to generate the rated torque of 600 in-lbf about the drive axis. The load direction was chosen to produce bending consistent with hand-applied torque during normal wrench operation.

This loading and constraint configuration ensures that the FEM model closely matches the assumptions used in the hand calculations while capturing local stress concentrations and non-uniform strain distributions.

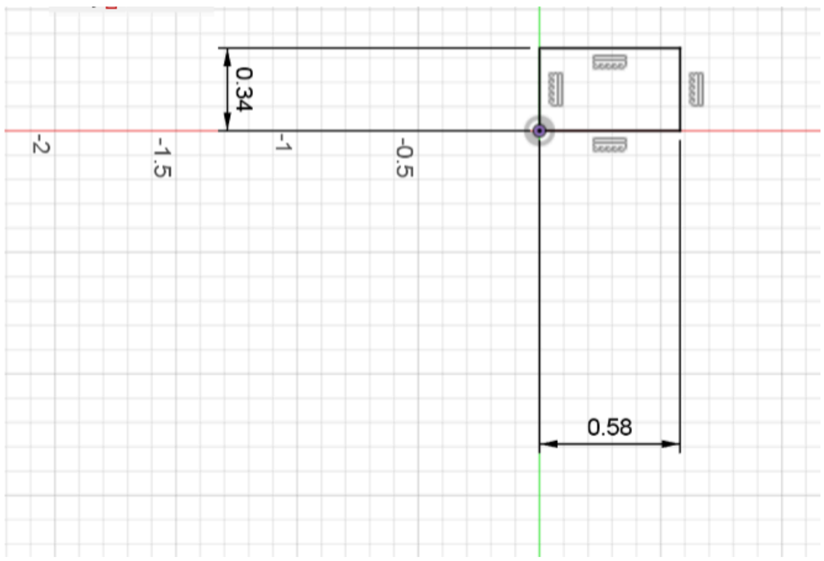
4. Normal strain contours (in the strain gauge direction) from FEM
See above
5. Contour plot of maximum principal stress from FEM
See above
6. Summarize results from FEM calculation showing maximum normal stress (anywhere), load point deflection, strains at the strain gauge locations
See above

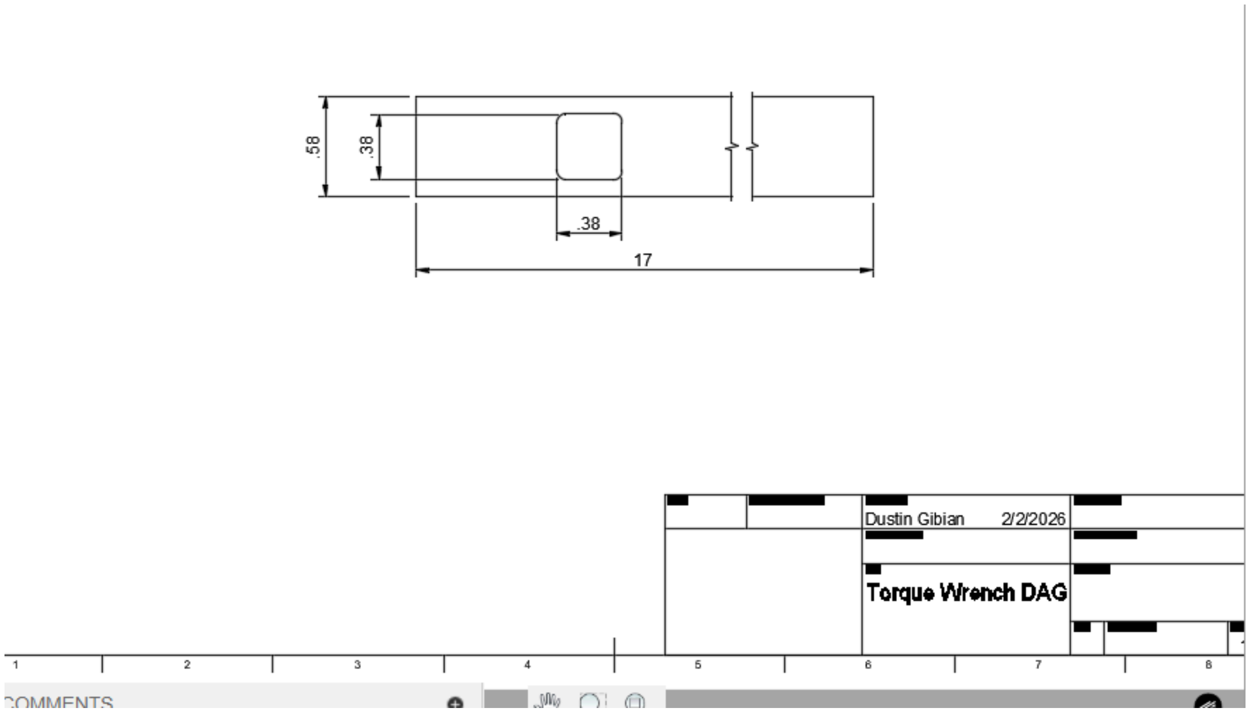
7. Torque wrench sensitivity in mV/V using strains from the FEM analysis
Sensitivity (mV/V)$=2.0 \cdot (1788\times10^{-6}) \cdot 0.5 \cdot 1000$
Sensitivity = 1.79 mV/V

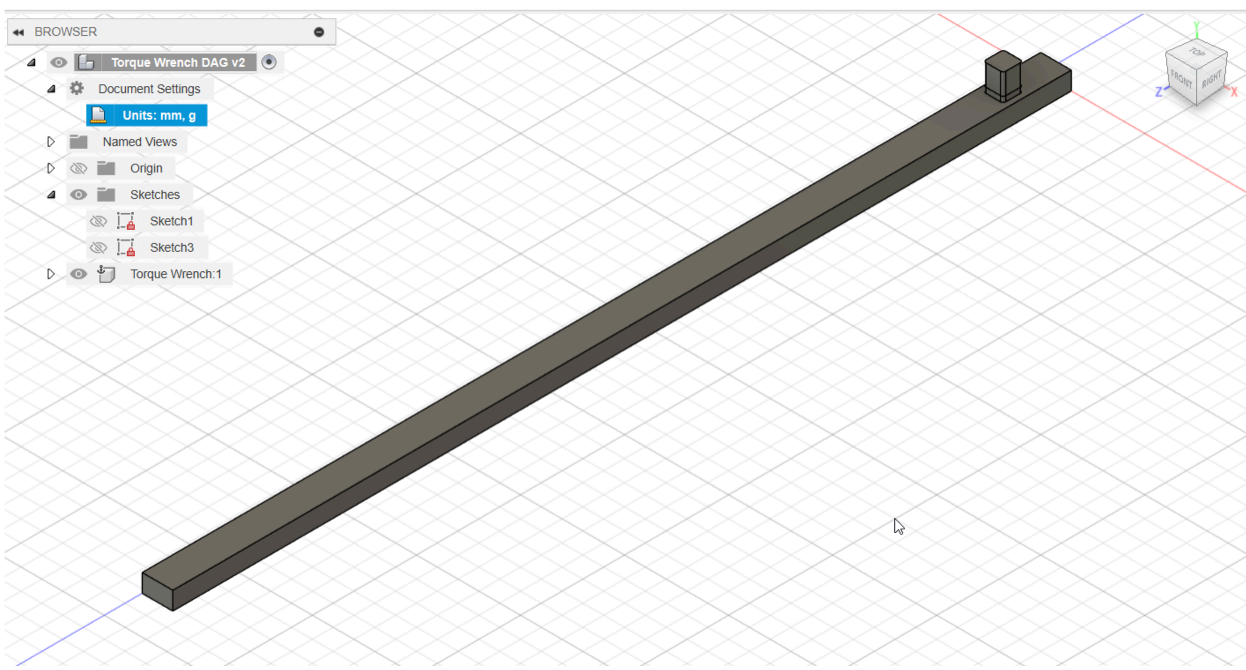
8. Strain gauge selected (give type and dimensions). Note that design must physically have enough space to bond the gauges.
A metal foil strain gauge with a gauge factor of approximately 2.0 was selected due to its linear response, durability, and suitability for metallic structures. The gauge dimensions (approximately

0.125 in grid length) are small enough to fit comfortably on the 0.58 in × 0.34 in wrench cross-section while allowing proper bonding and lead routing. A half-bridge configuration was used to increase sensitivity and provide partial temperature compensation.

My Design:



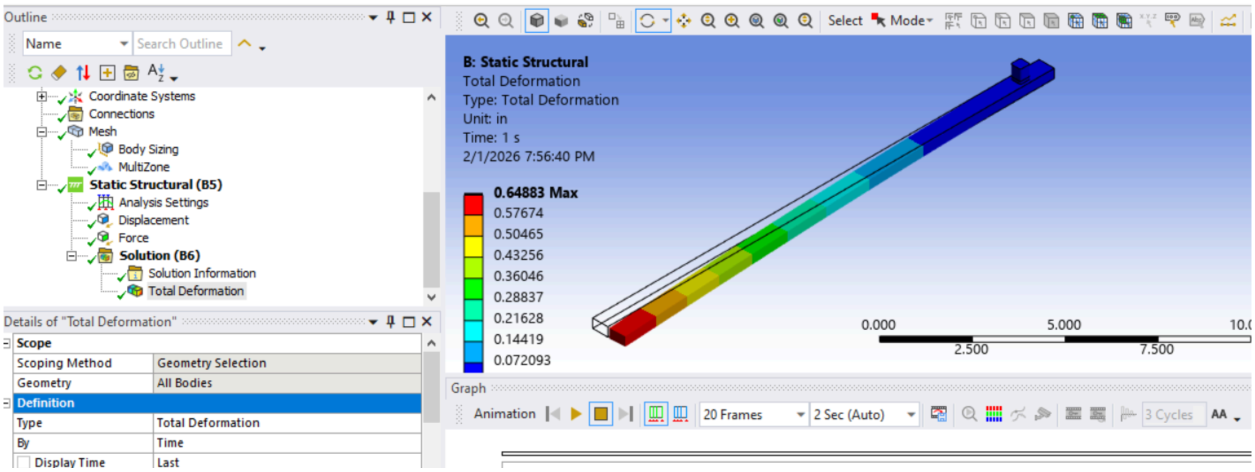




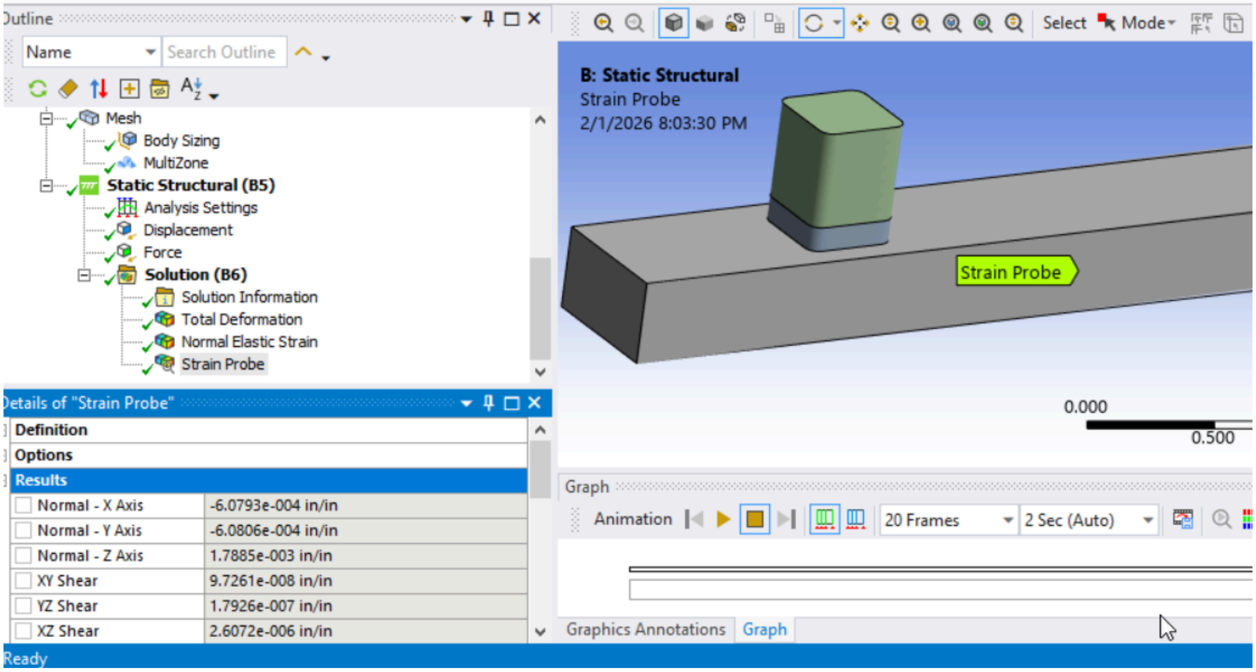
Ti-6Al-4v alloy
B = 0.340”
H = 0.580”

```
PS C:\Users\dagib\OneDrive\Desktop\MechMaterials> & C:/Python/Python313/python.exe c:/Users/dagib/OneDrive/Desktop/MechMaterials/finalhw1.py
=== TORQUE WRENCH HAND CALC RESULTS ===

Material: Ti-6Al-4V
b = 0.340 in, h = 0.580 in
Max stress = 31.48 ksi
Strain at gauge = 1687.1 microstrain
Deflection at load = 0.561 in

---- STRAIN GAUGE OUTPUT ----
Output = 1.69 mV/V

---- SAFETY FACTORS ----
Strength SF = 4.07
Fatigue SF = 1.91
Crack SF = 4.40

✅ DESIGN MEETS ALL REQUIREMENTS
PS C:\Users\dagib\OneDrive\Desktop\MechMaterials>
```

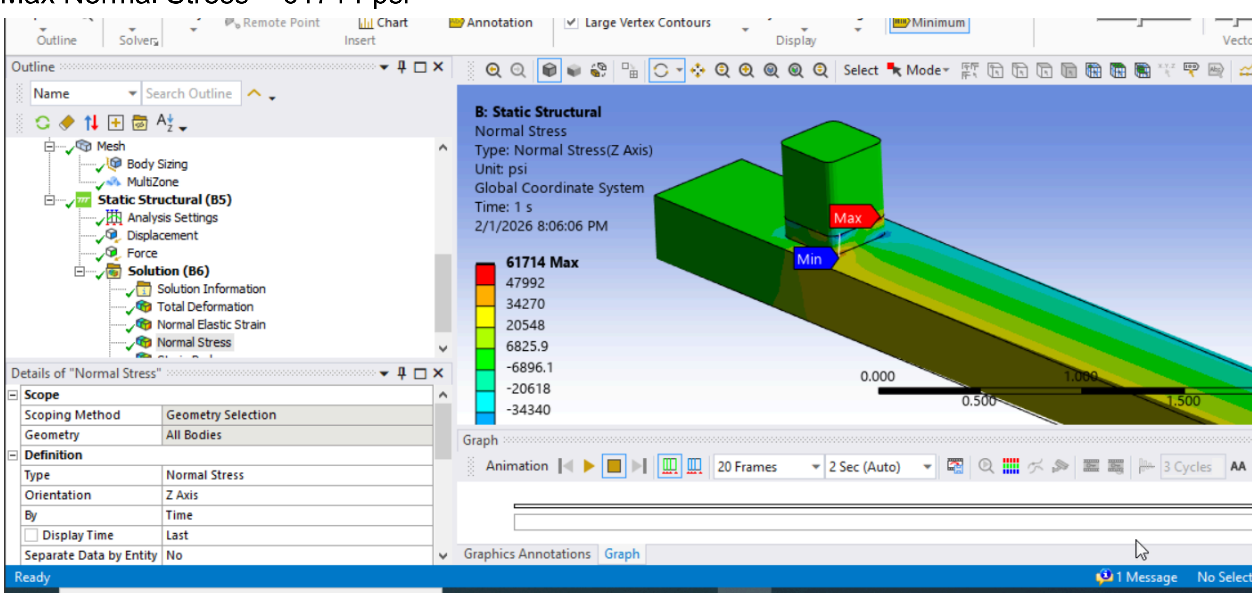
1st run mesh 0.20 overall and 0.06 at drive
Displacement Max = 0.64883”

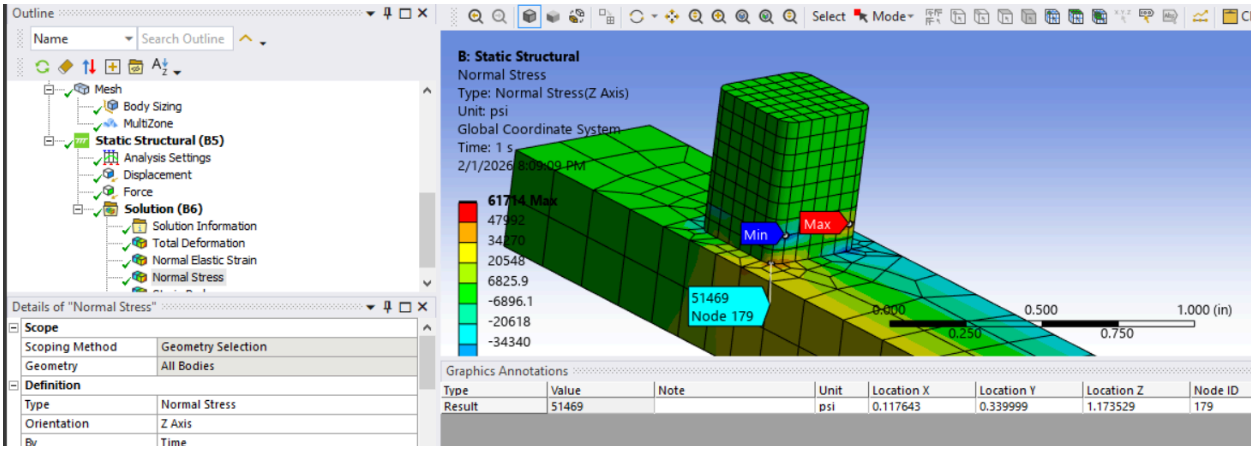


Normal Strain Z = 1788 Microstrain



Max Normal Stress = 61714 psi

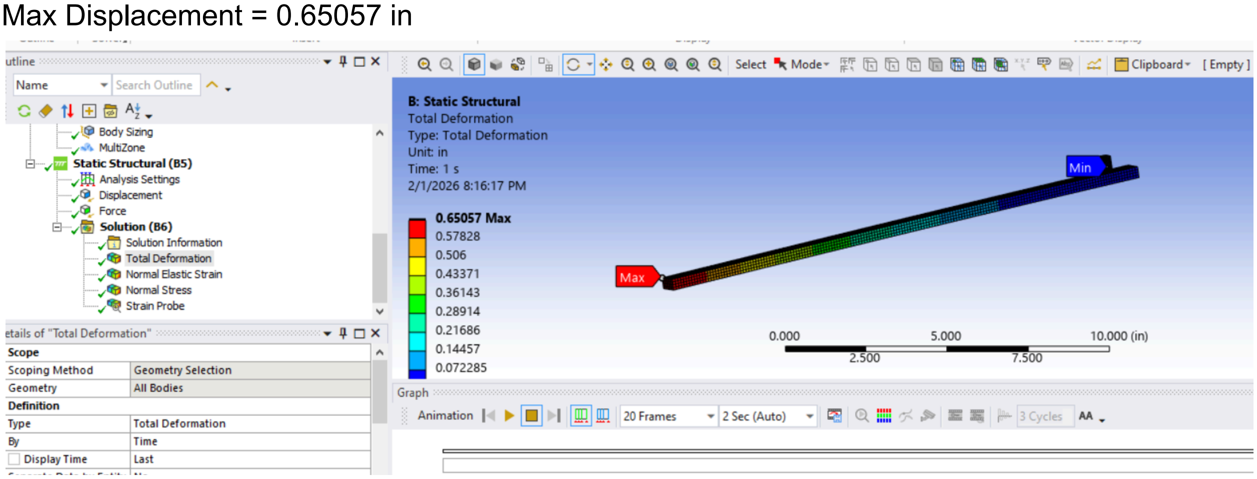




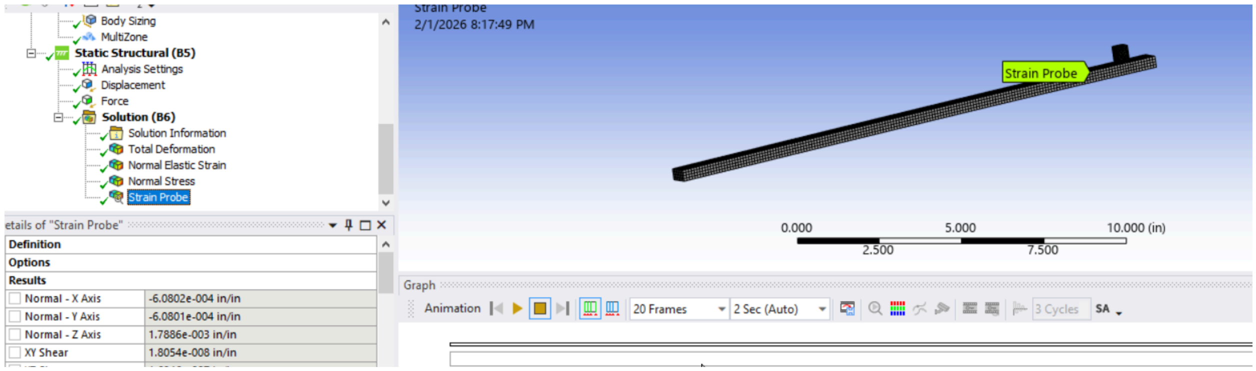
2nd run:

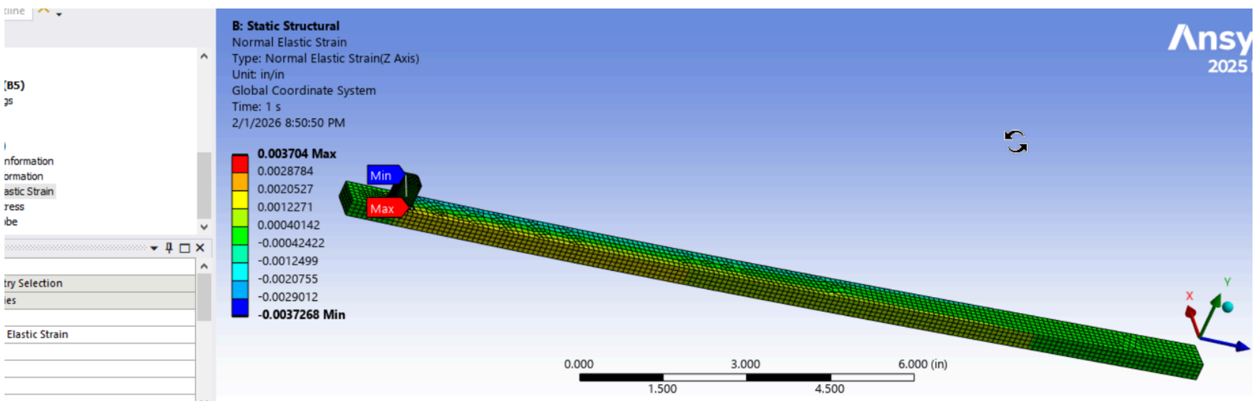
Mesh 0.1 overall and 0.04 drive

Max Displacement = 0.65057 in

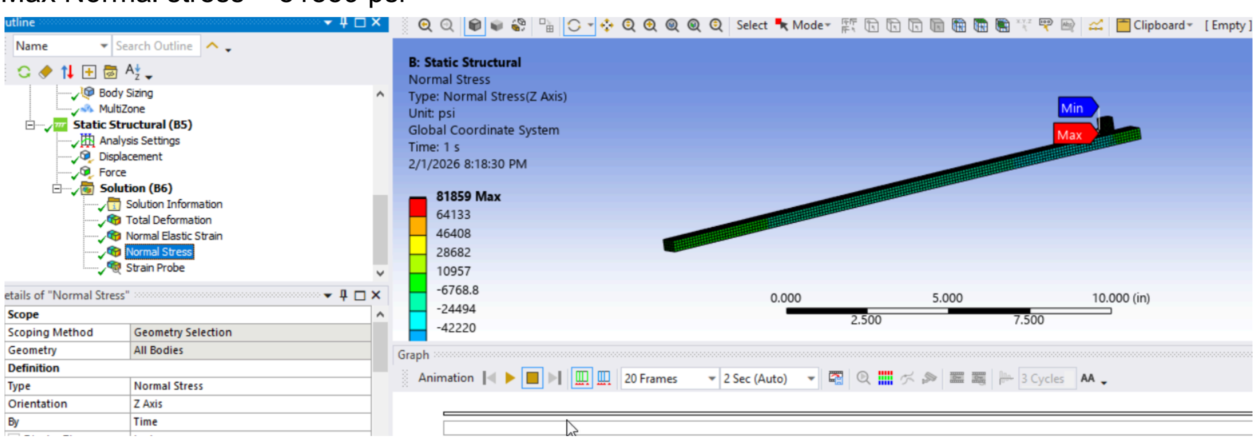


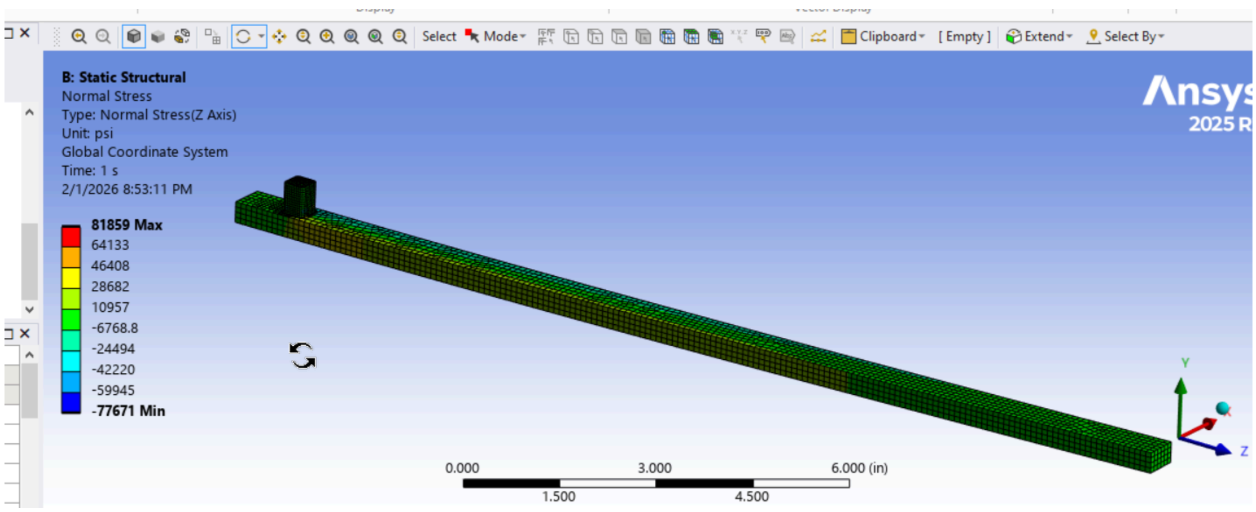
Normal Z strain = 1788 microstrain





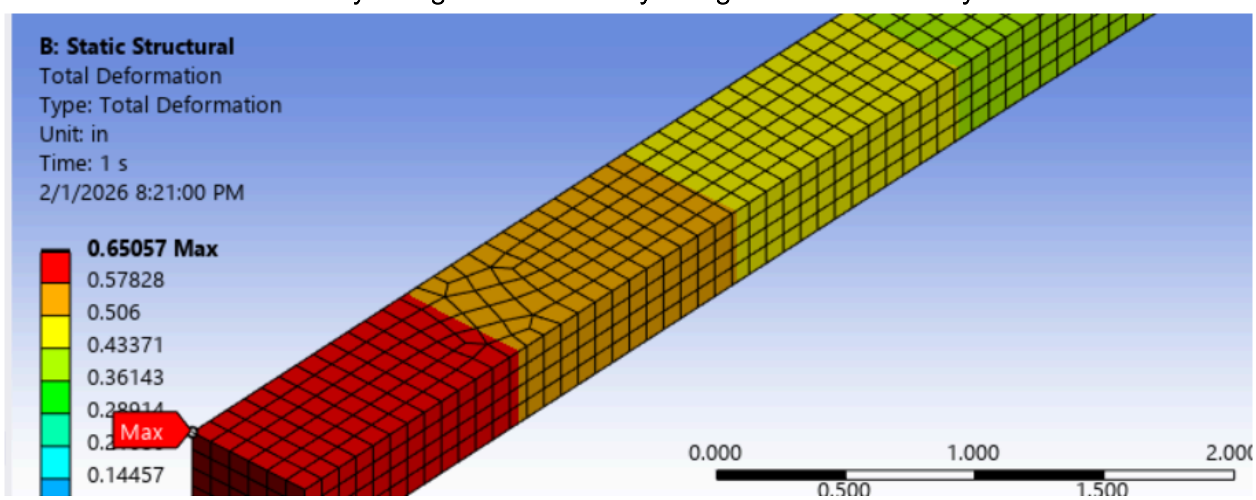
Max Normal stress = 81859 psi





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 mesh remains relatively straight. Beam theory is a good baseline analysis tool.



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

Hand calculations show a max normal stress of 31.8ksi, while the ansys analysis gives a much higher result between 61-82ksi. The hand calculation underpredicts maximum stress because it neglects stress concentrations that are captured by the finite element analysis. In the hand analysis, the wrench is idealized as a prismatic beam with uniform cross-section subjected to bending. Classical beam theory assumes uniform stress distribution across the section, smooth load transfer, and no geometric discontinuities. The finite element analysis includes geometric discontinuities such as corners and fillets and three-dimensional stress states.

The hand calculations show that the torque wrench meets all the requirements, but the ansys analysis shows that my torque wrench design fails to meet the safety factor of 4 for Strength.

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

The hand calculation predicts a load-point displacement of 0.561 in, while the ANSYS finite element analysis predicts a slightly larger displacement of 0.651 in.
The FEM displacement is larger because it includes additional deformation mechanisms not accounted for in the hand calculation. The difference between these results arises from the simplifying assumptions used in the hand calculation. In the hand analysis, the wrench is modeled using Euler–Bernoulli beam theory, which assumes small deflections, linear elastic behavior, uniform cross-section, and idealized boundary conditions.
This approach neglects additional sources of compliance such as local deformation near the drive interface, shear deformation, three-dimensional effects, geometric flexibility at fillets and transitions.

finalhw1.py

```
import numpy as np

# ========================
# CONSTANTS & REQUIREMENTS
# ========================

T = 600.0                  # applied torque (in-lbf)
L = 16.0                   # handle length (in)
GF = 2.0                   # strain gauge factor
bridge_factor = 0.5        # half-bridge
eps_cycles = 1e6           # fatigue cycles (used implicitly)

# Safety factor requirements
SF_strength_req = 4.0
SF_fatigue_req = 1.5
SF_crack_req = 2.0

# Crack assumptions
a = 0.04                   # crack depth (in)
Y = 1.12                   # geometry factor (edge crack)

# ========================
# MATERIAL DATABASE
# ========================
materials = {
    "M42 Steel": {
        "E": 32e6,           # psi
        "nu": 0.29,
        "sigma_y": 370e3,    # psi
        "KIC": 15e3,         # psi*sqrt(in)
        "sigma_f": 115e3     # fatigue strength (psi)
    },
    "7075-T6 Aluminum": {
        "E": 10.4e6,
        "nu": 0.33,
        "sigma_y": 73e3,
        "KIC": 25e3,
        "sigma_f": 23e3
    },
    "Ti-6Al-4V": {
        "E": 16.5e6,
        "nu": 0.34,
        "sigma_y": 128e3,
        "KIC": 55e3,
        "sigma_f": 60e3
    }
}

```

```python
# ========================
# DESIGN VARIABLES
# ========================
h = 0.58    # width (in)
b = 0.34     # thickness (in)


material_name = "Ti-6Al-4V"
material = materials[material_name]

# ========================
# HAND CALCULATIONS
# ========================

# Section properties
I = (b * h**3) / 12.0
c_max = h / 2.0          # extreme fiber (failure)
c_gauge = 1.0            # distance from drive center to gauge (in)

# Bending moment at flexure
M = T   # torque produces equivalent bending moment

# Stress & strain


sigma_max = M * c_max / I
sigma_gauge = M * c_gauge / I

nu = material["nu"]
strain_gauge = sigma_max * (1 - nu**2) / material["E"]



# Deflection at load point
delta = (M * L**2) / (3 * material["E"] * I)

# ========================
# STRAIN GAUGE OUTPUT
# ========================
Vout_over_Vin = GF * strain_gauge * bridge_factor
mV_per_V = Vout_over_Vin * 1000

# ========================
# FAILURE CRITERION
# ========================
# Ductile failure criterion is used.
# The material (M42 Steel / Ti-6Al-4V) is ductile, so yielding governs failure.
# Strength safety factor is based on yield strength under bending
# (maximum normal stress, equivalent to von Mises for pure bending).
# Brittle fracture is NOT the primary failure mode, but crack growth
```

```
# is checked separately using fracture mechanics (K_IC).

# =========================
# SAFETY FACTORS
# =========================

SF_strength = material["sigma_y"] / sigma_max
SF_fatigue = material["sigma_f"] / sigma_max

K = Y * sigma_max * np.sqrt(np.pi * a)
SF_crack = material["KIC"] / K

# =========================
# OUTPUT RESULTS
# =========================

print("=== TORQUE WRENCH HAND CALC RESULTS ===\n")
print(f"Material: {material_name}")
print(f"b = {b:.3f} in, h = {h:.3f} in")
print(f"Max stress = {sigma_max/1000:.2f} ksi")
print(f"Strain at gauge = {strain_gauge*1e6:.1f} microstrain")
print(f"Deflection at load = {delta:.3f} in\n")

print("---- STRAIN GAUGE OUTPUT ----")
print(f"Output = {mV_per_V:.2f} mV/V\n")

print("---- SAFETY FACTORS ----")
print(f"Strength SF = {SF_strength:.2f}")
print(f"Fatigue SF = {SF_fatigue:.2f}")
print(f"Crack SF = {SF_crack:.2f}\n")

# =========================
# REQUIREMENT CHECK
# =========================

if (mV_per_V >= 1.0 and
    SF_strength >= SF_strength_req and
    SF_fatigue >= SF_fatigue_req and
    SF_crack >= SF_crack_req):
    print("✅ DESIGN MEETS ALL REQUIREMENTS")
else:
    print("❌ DESIGN DOES NOT MEET ALL REQUIREMENTS")
```