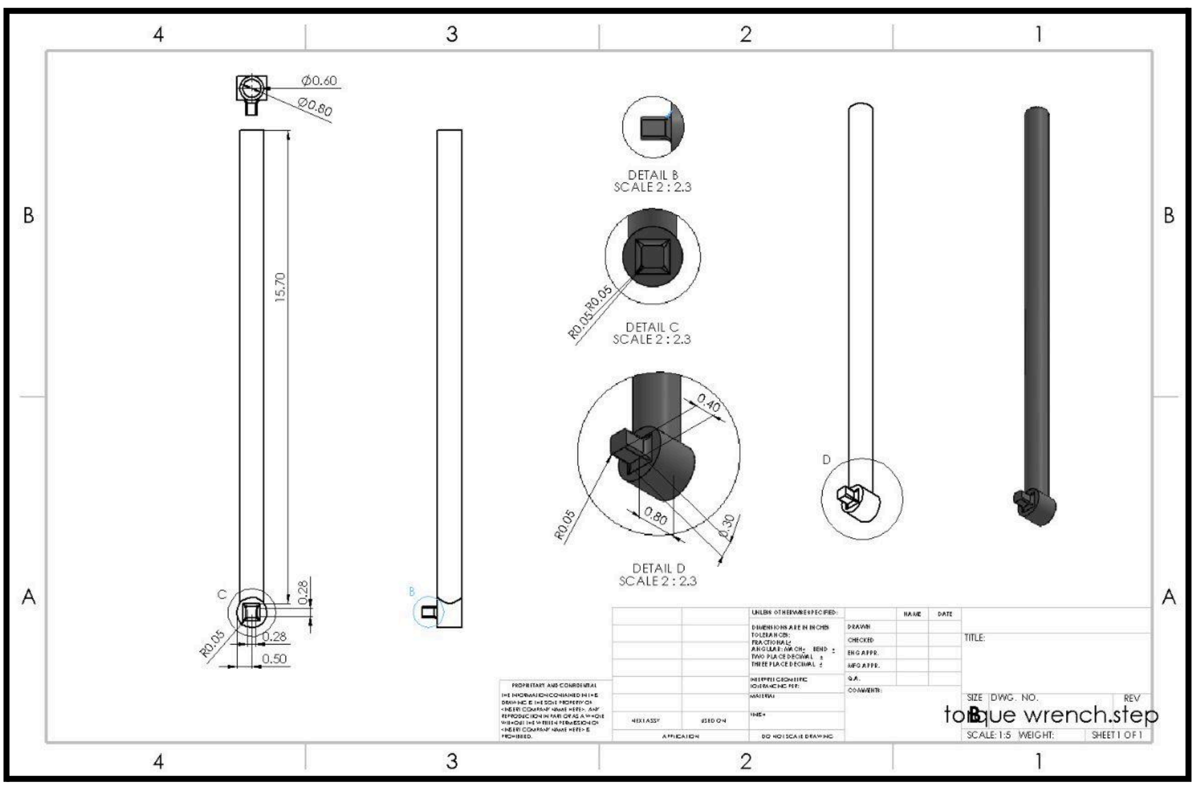
```matlab
function [] = finalProj()
    %Loads
    M = 600;
    L = 16;
    F = M / L;

    %Wrench dimensions
    ro=.4;
    ri=.3;
    c = ro;        % Distance from center of drive to center of strain gauge (inches)

    %Material Properties
    name = 'Ti-6Al-4V';
    E = 16.5E6;     % Young's modulus (psi)
    ou = 138.0E3;   % Ultimate Tensile Strength (psi) - Use for safety factor
    KIC = 68.3E3;   % Fracture toughness (psi * sqrt(in))
    S = 74.0E3; % Fatigue strength for 10^6 cycles (psi)

    I=pi/4*(ro^4-ri^4);
    X0req=4;
    Xkreq=2;
    Xsreq=1.5;

    omax=M*ro/I; %also ogauge
    ex = omax/E;
    deflection = (F*L^3)/(3*E*I);

    %finding o1,o2
    oav=omax/2;
    R=omax/2;
    o1=oav+R;
    o2=oav-R;

    disp(["Max Stress: ", omax]);
    disp(["Max Deflection: ", deflection]);

    X0=ou/o1;
    disp(["Strength safety factor: ", X0]);

    a = 0.04; % crack depth (in)
    K_I = abs(omax) * sqrt(pi * a);
    Xk = KIC / K_I;
    disp(["Fracture safety factor: ", Xk]);

    Xs = S / abs(omax);
    disp(["Fatigue safety factor: ", Xs]);

    if X0>X0req && Xk>Xkreq && Xs>Xsreq
        disp('All safety factors are satisfied.');
    end

    GF=2;
    mVperV = (GF*ex/2)*1000; %half bridge strain
    disp(["mV/V: ", mVperV]);
end
```

Results

```
>> Materials_Final_Project
    "Max Stress: "    "17461.5709"

    "Max Deflection: "    "0.22577"

    "Strength safety factor: "    "7.9031"

    "Fracture safety factor: "    "11.034"

    "Fatigue safety factor: "    "4.2379"

All safety factors are satisfied.
    "mV/V: "    "1.0583"
```

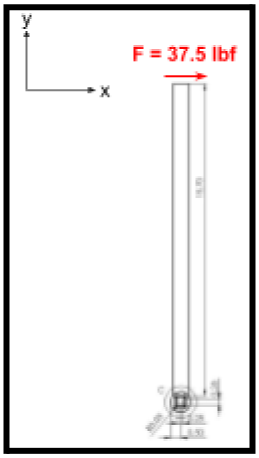
CAD Diagram

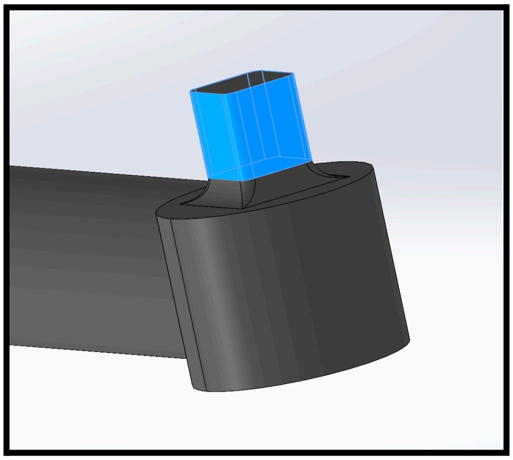


Material used and its relevant mechanical properties.

```
Name = 'Ti-6Al-4V'
E = 16.5e6;      % Young's modulus (psi)
σu = 138.0e3;    % Ultimate Tensile Strength (psi) - Use for safety factor
KIC = 68.3e3;    % Fracture toughness (psi * sqrt(in))
S = 74.0e3; % Fatigue strength for 10^6 cycles (psi)
```

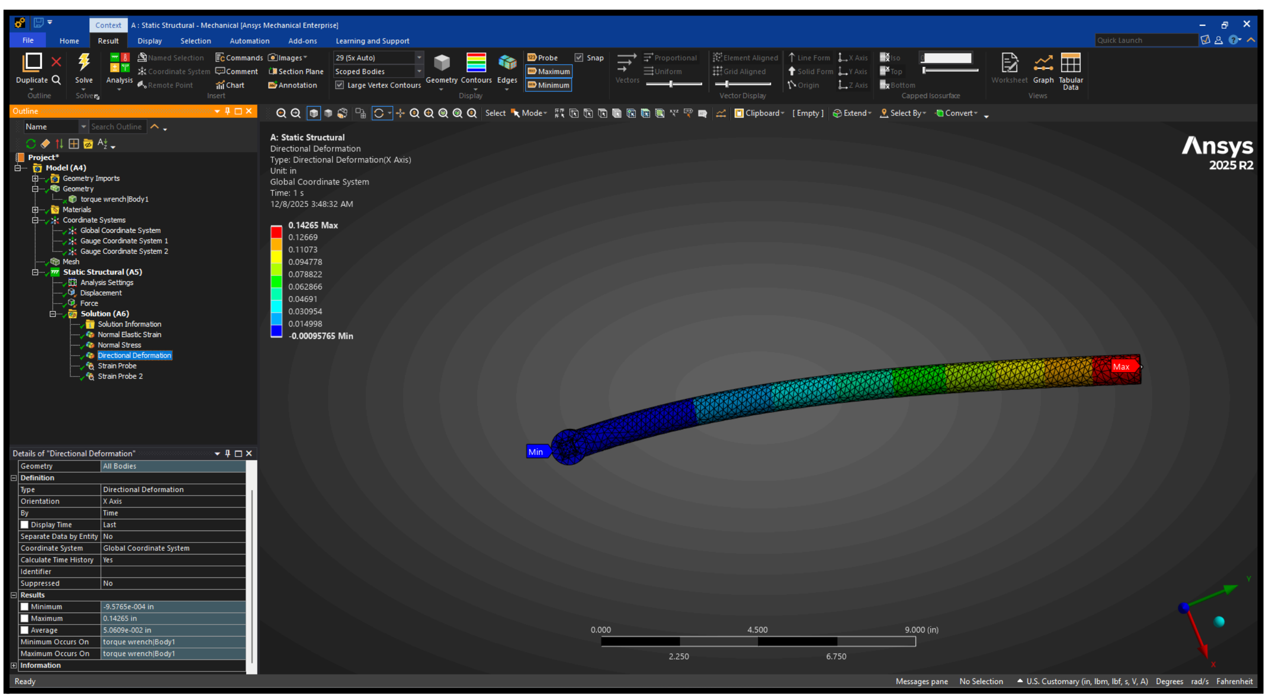
Diagram communicating how loads and boundary conditions were applied in ANSYS



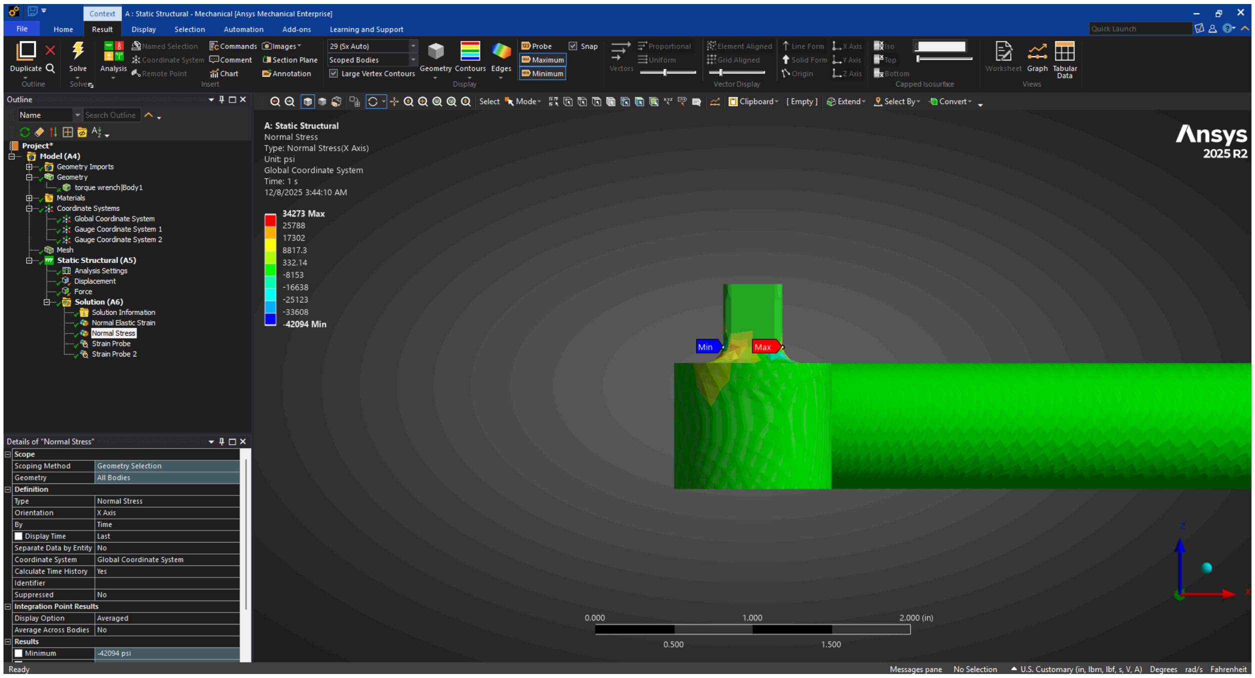


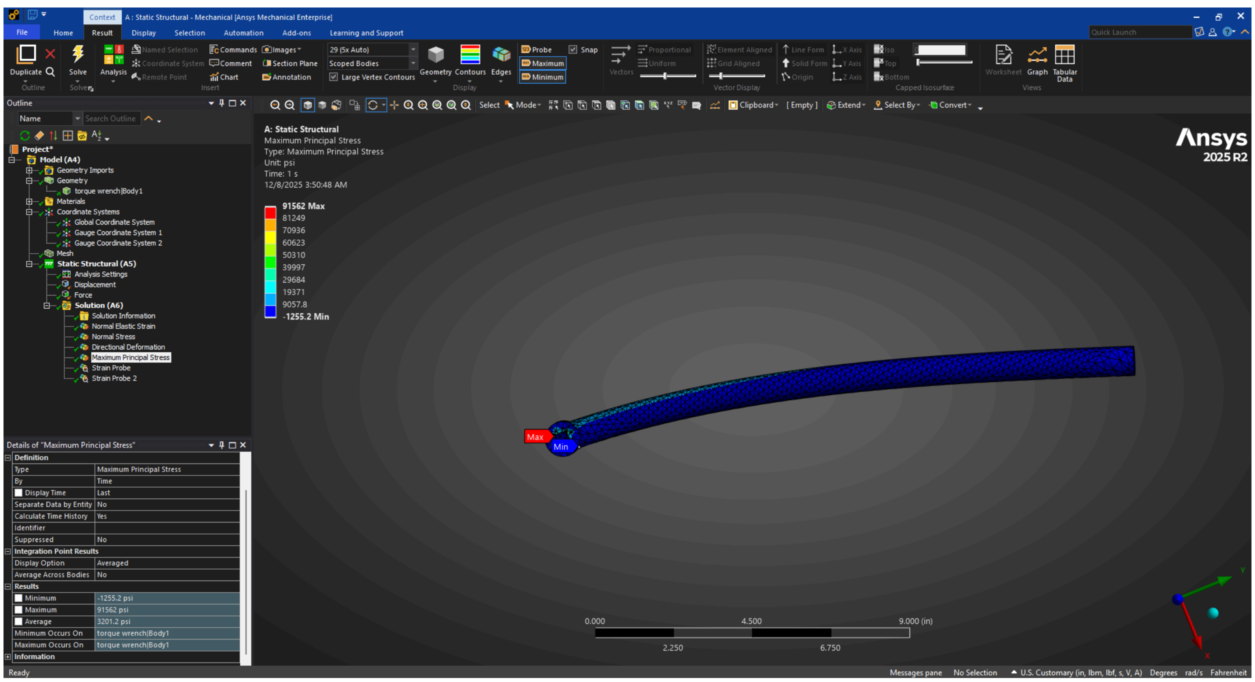
In order to simulate the use case of an instrumented torque wrench a load of 37.5lbf was applied to end of the torque wrench's handle. The head of the torque wrench was constrained to zero deflection in all directions which allowed the rest of the torque wrench to to deflect around it.

Normal strain contours and load point deflection

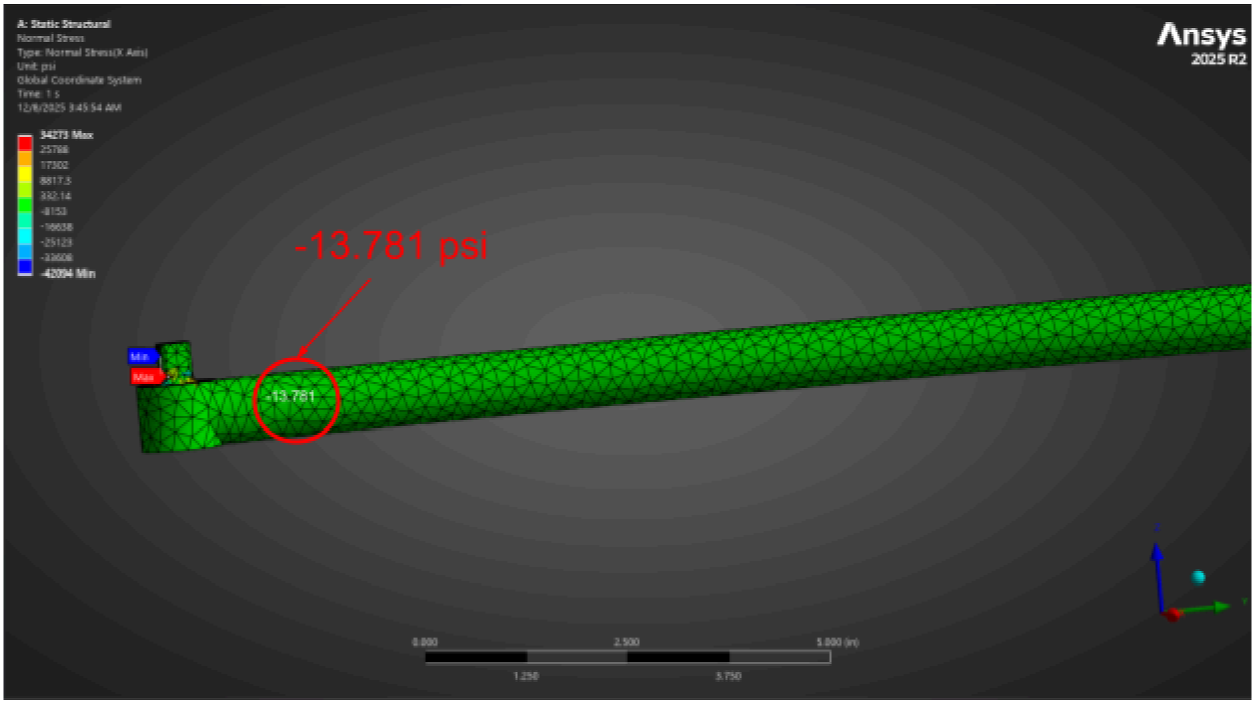


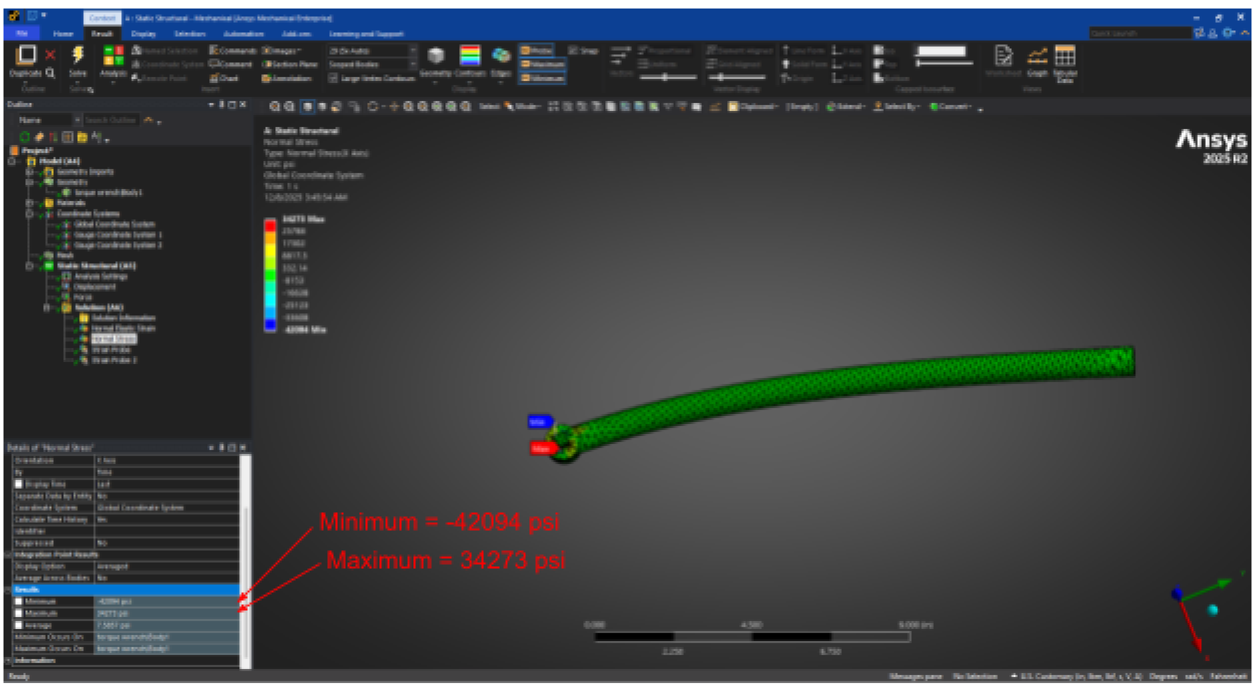
Contour plot of maximum principal stress


A: Static Structural
Normal Stress
Type: Normal Stress(X Axis)
Unit: psi
Global Coordinate System
Time: 1 s
12/8/2025 3:44:10 AM
34273 Max
25788
17302
8817.3
332.14
-8153
-16638
-25123
-33608
-42094 Min
Min
Max
Ansys
2025 R2
0.000
1.000
2.000 (in)
0.500
1.500


A: Static Structural
Maximum Principal Stress
Type: Maximum Principal Stress
Unit: psi
Time: 1 s
12/8/2025 3:50:48 AM
91562 Max
81249
70936
60623
50310
39997
29684
19371
9057.8
-1255.2 Min
Max
Min
Ansys
2025 R2
0.000
4.500
9.000 (in)
2.250
6.750
Minimum -1255.2 psi
Maximum 91562 psi
Average 3201.2 psi
Minimum Occurs On torque wrench\Body1
Maximum Occurs On torque wrench\Body1

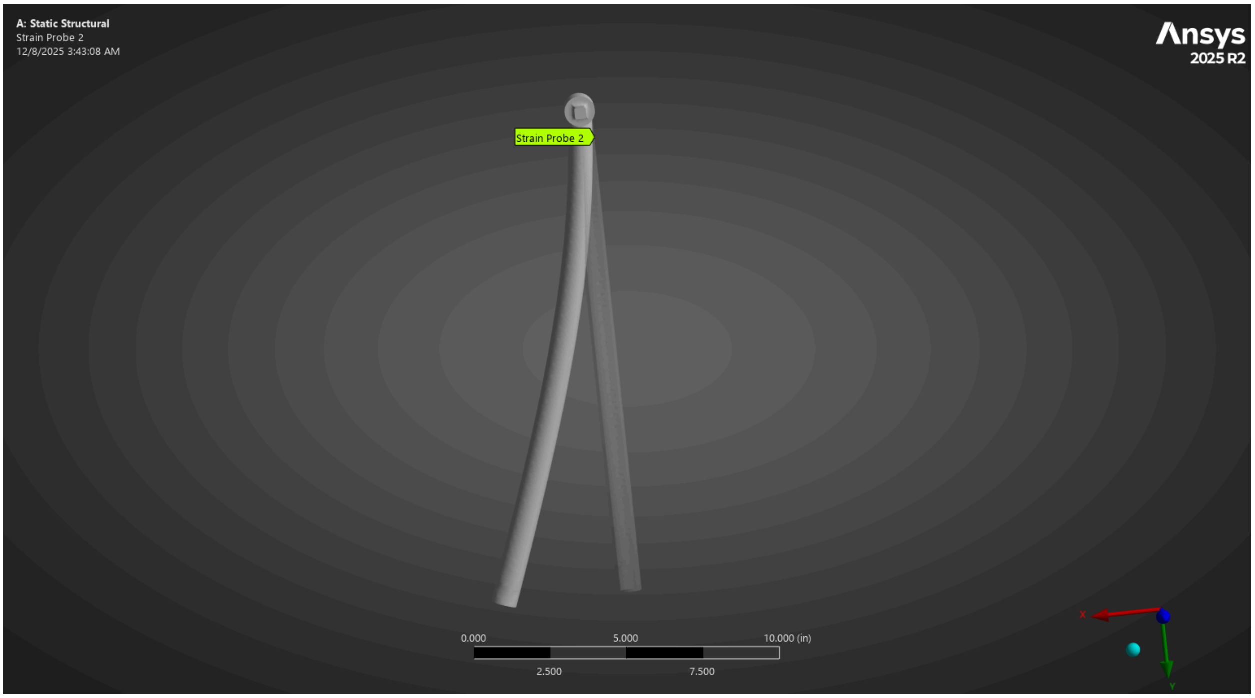
Maximum normal stress

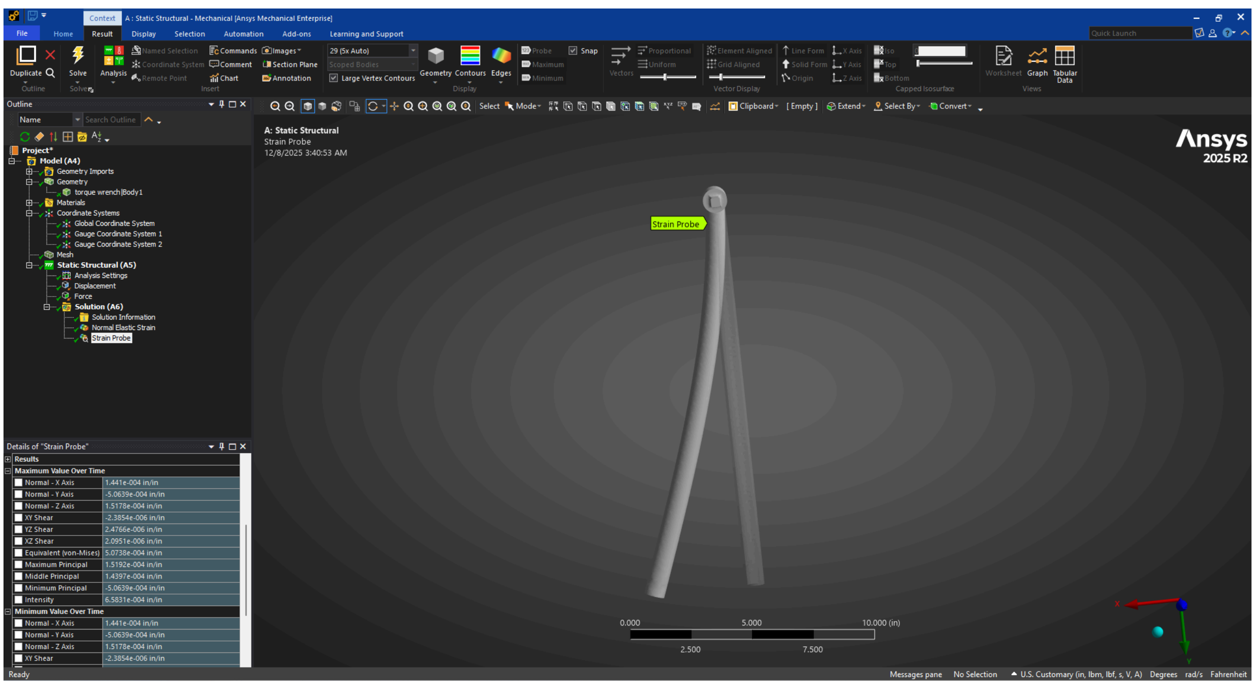


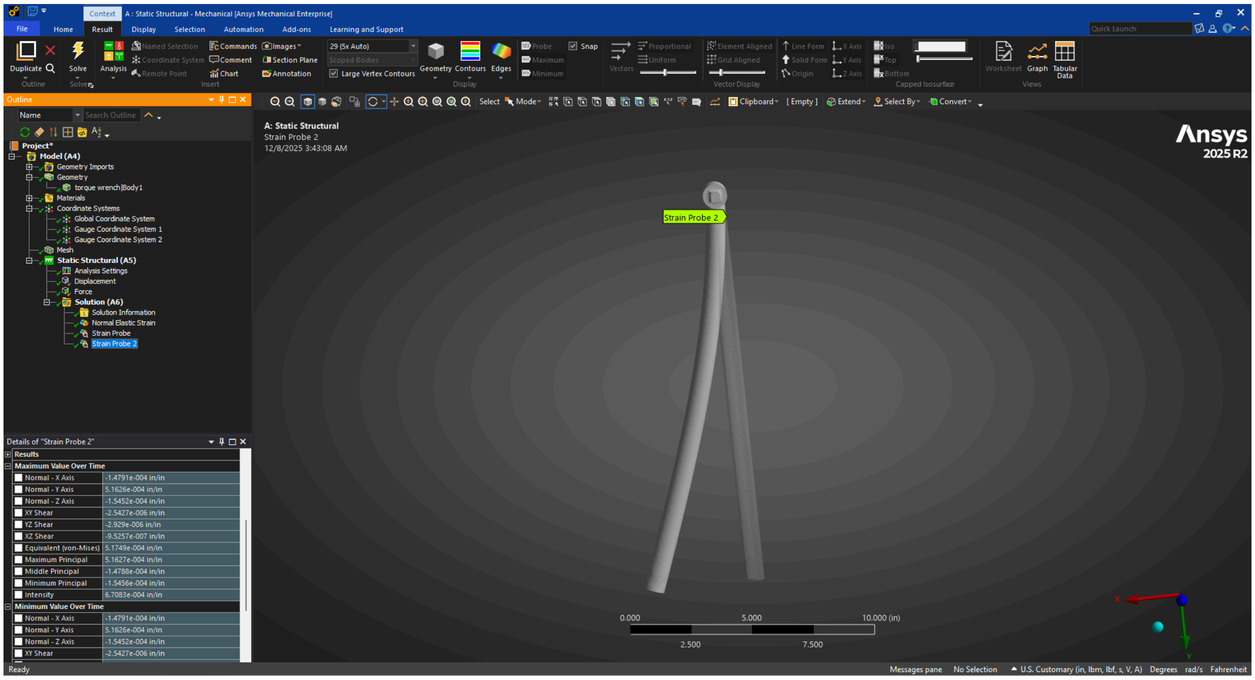


My hand calcs predicted a max stress value of 17461. ANSYS gives me a max stress value of almost two times that. This is likely the result of the fact that there is stress singularities where the constrained part of the torque wrench head meets the unconstrained part. When doing a stress probe around the area I would expect the max stress to be I got a strangely small number.

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


A: Static Structural
Strain Probe 2
12/8/2025 3:43:08 AM
Ansys
2025 R2
Strain Probe 2
0.000
5.000
10.000 (in)
2.500
7.500


A : Static Structural - Mechanical [Ansys Mechanical Enterprise]
Outline
Project*
Model (A4)
Geometry Imports
Geometry
torque wrench|Body1
Materials
Coordinate Systems
Global Coordinate System
Gauge Coordinate System 1
Gauge Coordinate System 2
Mesh
Static Structural (A5)
Analysis Settings
Displacement
Force
Solution (A6)
Solution Information
Normal Elastic Strain
Strain Probe
Details of "Strain Probe"
Results
Maximum Value Over Time
Normal - X Axis 1.441e-004 in/in
Normal - Y Axis -5.0639e-004 in/in
Normal - Z Axis 1.5178e-004 in/in
XY Shear -2.3854e-006 in/in
YZ Shear 2.4766e-006 in/in
XZ Shear 2.0951e-006 in/in
Equivalent (von-Mises) 5.0738e-004 in/in
Maximum Principal 1.5192e-004 in/in
Middle Principal 1.4397e-004 in/in
Minimum Principal -5.0639e-004 in/in
Intensity 6.5831e-004 in/in
Minimum Value Over Time
Normal - X Axis 1.441e-004 in/in
Normal - Y Axis -5.0639e-004 in/in
Normal - Z Axis 1.5178e-004 in/in
XY Shear -2.3854e-006 in/in
A: Static Structural
Strain Probe
12/8/2025 3:40:53 AM
Ansys
2025 R2
Strain Probe
0.000
5.000
10.000 (in)
2.500
7.500
Ready



Output = .15mV/V
Strain at gauge = 150 microstrain

Strain gauge selected

For this application it makes the most sense to use a linear strain gauge. We don't a rosette since we only care about the stain around one axis. They make many relatively cheap strain gauges that are the right dimension to fit on my torque wrench. The the strain gauge I settled on is 6mm-6.3mm. It would make sense to put a strain gauge on each side of the wrench exactly opposite to each other in order to get an averaged strain value.