Finite element Analysis (FEA) Lab Project.
2025

Submitted by: Muhammad Anus (ME-1865).
Section: A.
Semester: 7th.

Submitted to: Sir Affan.

1. Static Structural Analysis

This section details the setup and results for the steady-state mechanical analysis of the hexagonal component.

1.1. Material Properties

The material properties were defined for a custom material using the student's roll number for one of the primary mechanical properties.

The Young's Modulus was defined using the last two digits of the roll number (ME-1865) multiplied by 1000. As shown in figure 1

Property	Value	Unit
Young's Modulus	65,000	MPa
Poisson's Ratio	0.28	-
Tensile Yield Strength	300	MPa

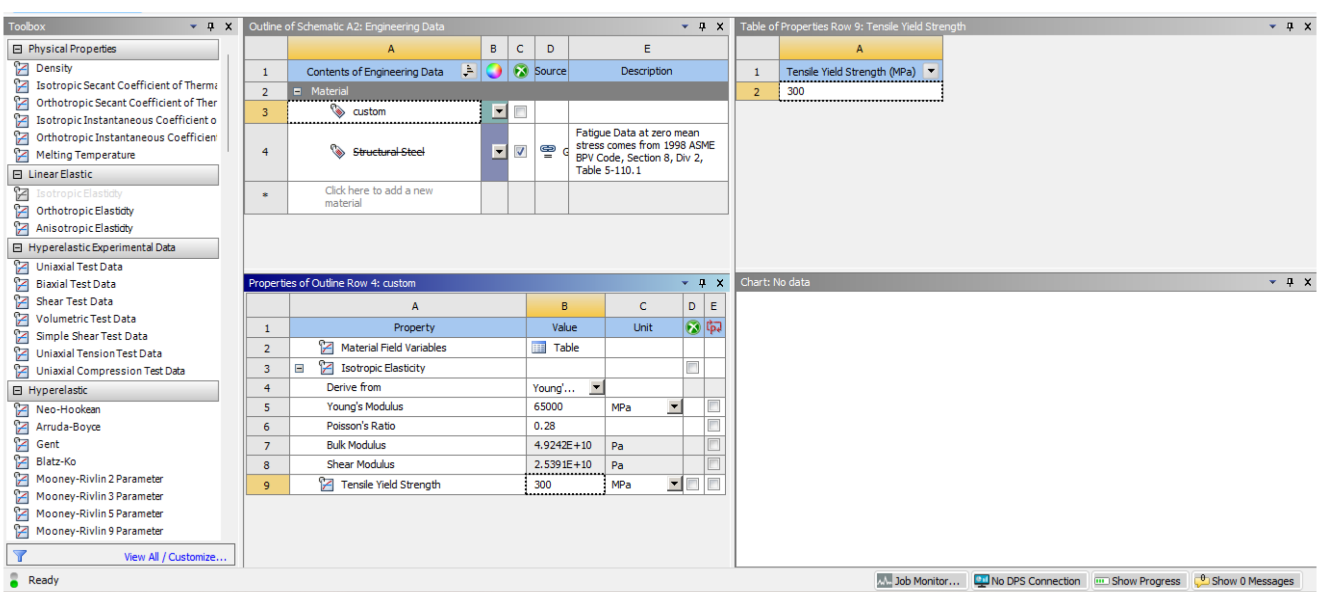


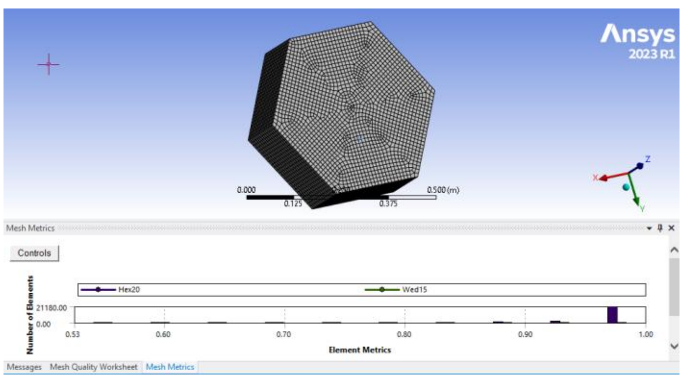
Figure 1: Material properties

1.2. Meshing

The quality of the mesh is paramount for obtaining accurate results in Finite Element Analysis. The overall mesh was generated using hexahedral elements, specifically Hex20 (20-node brick elements), which are high-order elements providing superior solution accuracy compared to linear elements.

To ensure high local fidelity, especially in regions critical to stress and deformation, **Face Meshing** was explicitly applied. Face Meshes were used for accuracy on all six primary faces of the hexagonal structure (Face Meshing 2 through 7 in the project tree). This strategy forces the mesher to generate a structured, higher-quality mesh distribution on these surfaces, which translates into more reliable stress and deformation outputs.

Mesh Metrics Analysis:
The Mesh Metrics graphs show a predominantly excellent element quality. The distribution is heavily weighted toward high-quality elements (values close to 1.00 on the Element Metrics chart). The two primary element types, Hex20 and Wed15 (wedge elements, likely transitional near complex features), confirm that the majority of the volume is filled with high-fidelity 3D elements, contributing significantly to the stability and accuracy of the simulation.



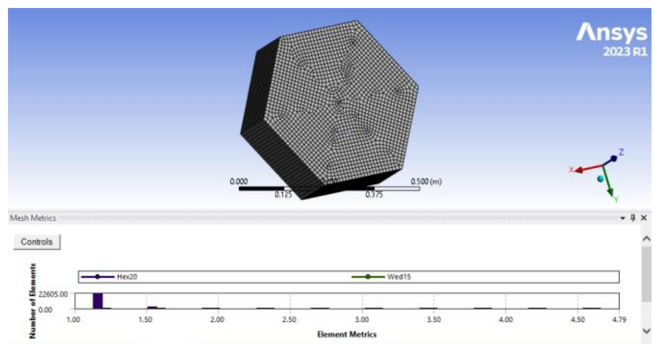
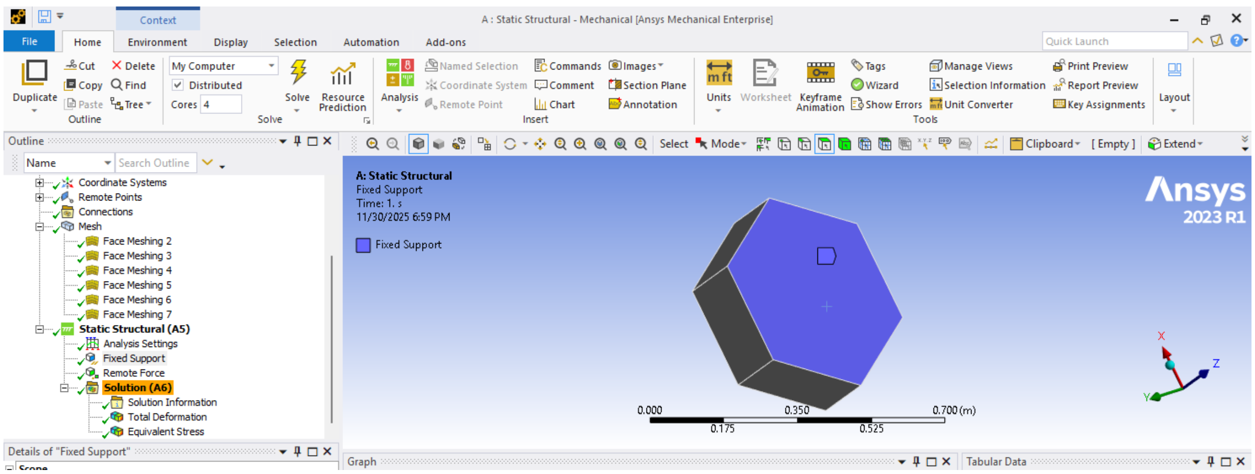


Figure 2 and 3: Meshing

1.3. Boundary Conditions

The analysis was secured by fixing the component and applying an external load.
| Fixed Support | One face of the hexagonal component was subjected to a Fixed Support boundary condition, constraining all degrees of freedom. A remote force was applied to the opposing face. The force magnitude was calculated by taking the base load (2034 N) by adding the birth date (27 Jun 2002). Shown in figure 4 and 5.



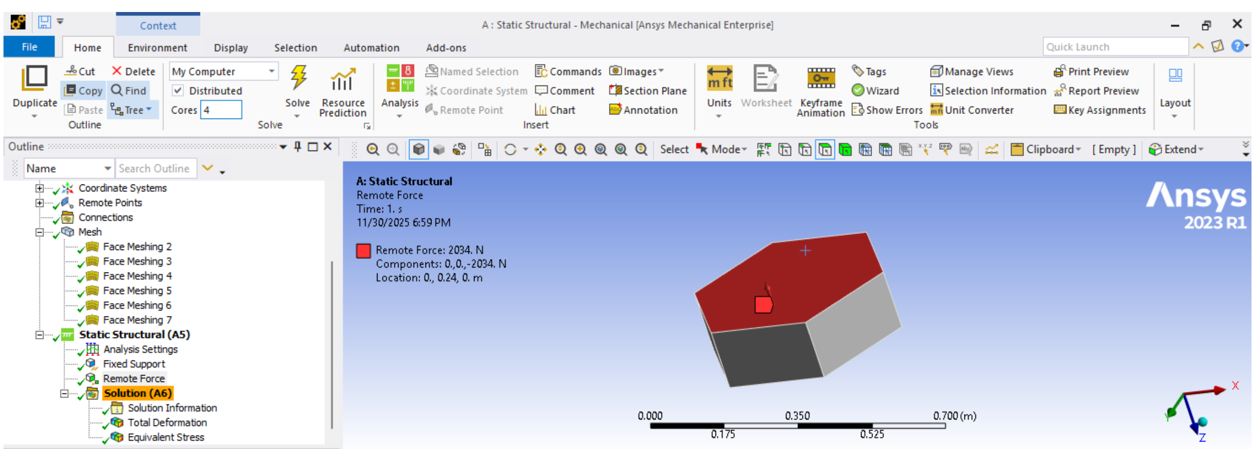


Figure 4 and 5: Boundary conditions

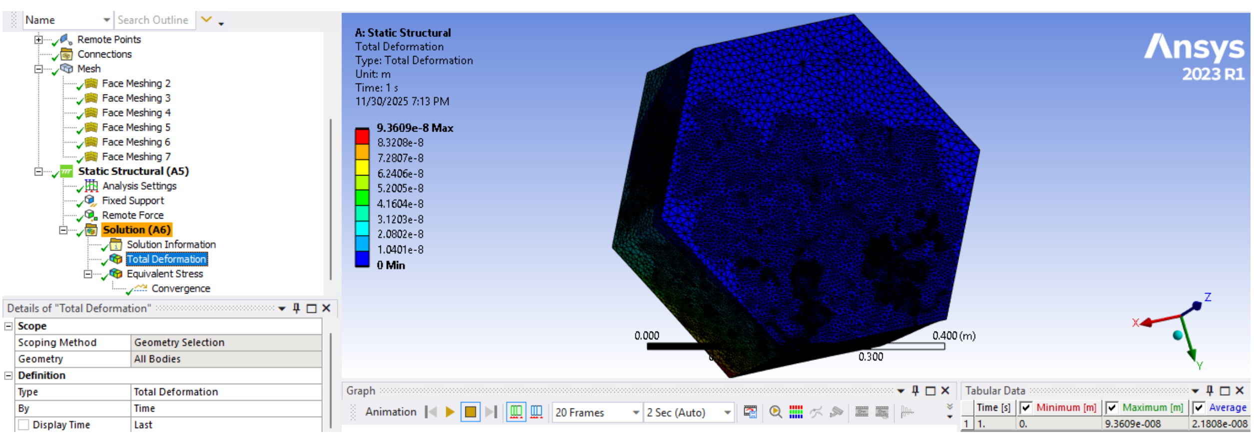
1.4. Results and Discussion

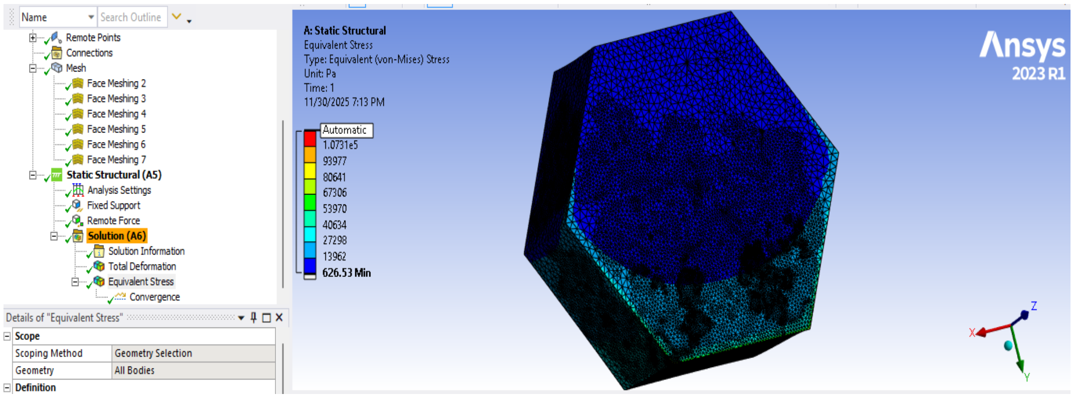
The FEA simulation successfully converged after a prescribed number of loops, yielding the stress and deformation profiles under the defined load case.

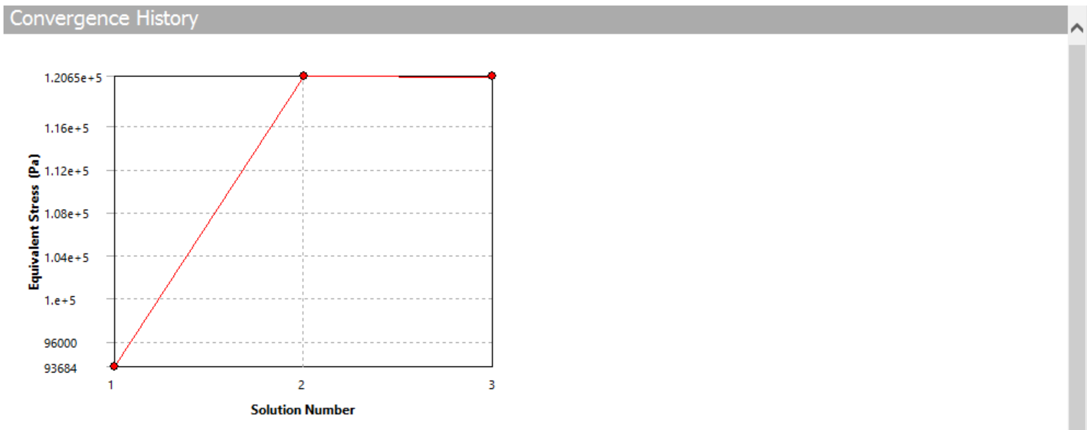
Convergence and Iterations:

The simulation was set to run for 10 loops to achieve convergence. The convergence history shows that the equivalent stress solution converged rapidly, stabilizing within 3 solution steps with a minimal change percentage between the final two steps (2.349e-10change). This rapid stabilization confirms the mesh quality was adequate and the solution found a stable equilibrium state quickly, even though the analysis was prepared for up to 10 loops.
Stress and Deformation:
The results show the maximum Equivalent (Von-Mises) Stress and Total Deformation experienced by the component.

- **Maximum Equivalent (Von-Mises) Stress: 107300 Pa**
  - **Discussion:** The maximum stress concentration is visible near the fixed support boundary and the sharp corners of the hexagonal face . This is a common phenomenon where geometric discontinuities and constrained boundaries induce localized stress peaks. Since the maximum stress **107300 Pa** is significantly lower than the material's Tensile Yield Strength (300 MPa}), the component is well within the elastic limit and shows no risk of plastic deformation or failure under this static load.
- **Maximum Total Deformation: 9.36 x 10 ^-8 mm**
  - **Discussion:** The deformation is minimal, which is expected given the high stiffness of the material (E = 65,000 MPa) and the relatively small applied load. The deformation is highest on the face where the remote force is applied, illustrating the component's compliance in the direction of the load. The negligible deformation further confirms the structural integrity and rigidity of the component under these static operating conditions.







	Equivalent Stress (Pa)	Change (%)	Nodes	Elements
1	93684		38629	19404
2	1.2062e+005	25.139	187163	115816
3	1.2065e+005	2.3491e-002	430320	292183

Figure 5,6,7: Deformation, Stress, Convergence.

2. Transient Structural Analysis

This section utilizes the same component geometry and material properties to analyze the time-dependent (transient) response to changing loads.

2.1. Boundary Conditions

The transient structural analysis was set up using a fixed support on one face, similar to the static analysis, but with variable forces applied over time to the other faces.

Condition	Description
Fixed Support	One face of the component was fully constrained.

Time Step	All analysis settings used a **time step of** 0.2s

The different sides of the hexagon were subjected to varying loads, as specified in the following tabular format:

Load Label	Side Applied	Force Magnitude
Force 1 (B)	Upper-left side	1,000 N
Force 2 (C)	Right side	2,000 N
Force 3 (D)	Lower-right side	3,000 N
Force 4 (E)	Bottom-left side	4,000 N

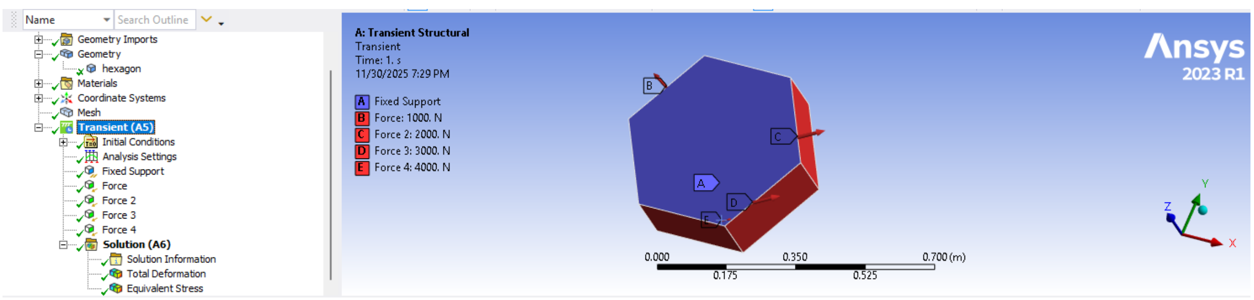


Figure 8: Transient Boundary conditions.

2.2. Transient Results and Discussion

The transient analysis calculated the structural response (stress and deformation) at each time step, showing how the component reacts to the loading sequence.

Transient Total Deformation:

- **Maximum Total Deformation: 3.088 x 10^-7mm**
  - **Discussion:** The maximum deformation is slightly higher than in the static analysis,

which is typical as transient solutions can capture inertial or dynamic effects that might increase displacement, even if the load is gradually applied. The maximum deformation is concentrated in the corners furthest from the fixed support and where the highest loads (Force 4 at 4000 N) are applied. The small magnitude of deformation indicates a highly stiff system with minimal inertial effects, suggesting the structure is very stable under the applied time-varying forces.

Transient Equivalent (Von-Mises) Stress:

- **Maximum Equivalent (Von-Mises) Stress: 7.769 x 10 ^5 Pa**
    - **Discussion:** Similar to the deformation, the stress values are higher in the transient case compared to the single static load application, reflecting the combined and time-dependent effect of multiple simultaneous loads. The highest stress is concentrated near the fixed support and the load application points, consistent with expected structural behavior. The maximum stress **7.769 x 10 ^5 Pa** remains several orders of magnitude below the yield strength (300 MPa), confirming that the hexagonal component maintains its structural integrity and remains in the elastic regime throughout the applied transient loading scenario. The stress distribution suggests that the component design is robust for both steady-state and time-varying loading conditions.

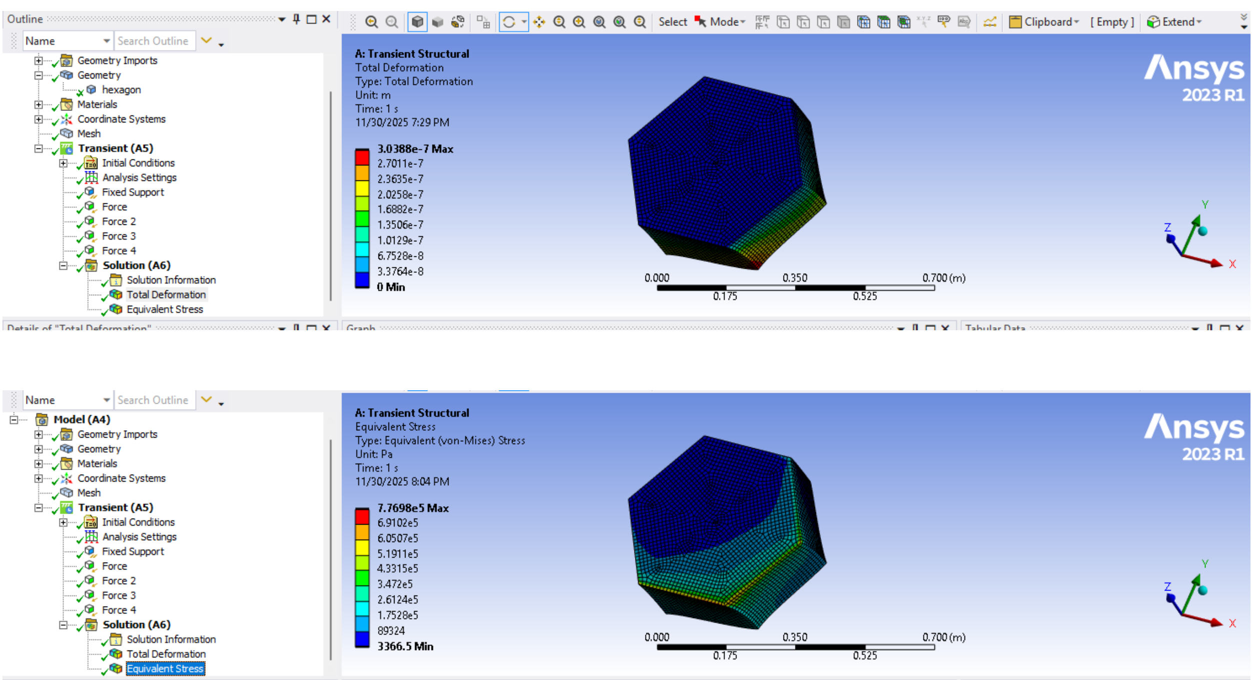


Figure 9,10: Transient Results.