FEA Project:
Static Structural and Transient Structural Analysis Report

By Arshia Farooqui

Semester VII, Sec B, ME-1859

—

Finite Element Analysis Lab

—

Submitted to:

Sir Affan Ahmed Khan

Table of Contents

STATIC STRUCTURAL ANALYSIS

To perform linear static structural analysis on a regular hexagonal prism of height 59 m using a linear elastic material with Young's modulus E = 59,000 MPa and Poisson's ratio v = 0.28; the prism is fully fixed at one end and subjected to a remote force of 2013 N applied at one-third of its height from the fixed face. To demonstrate mesh convergence.
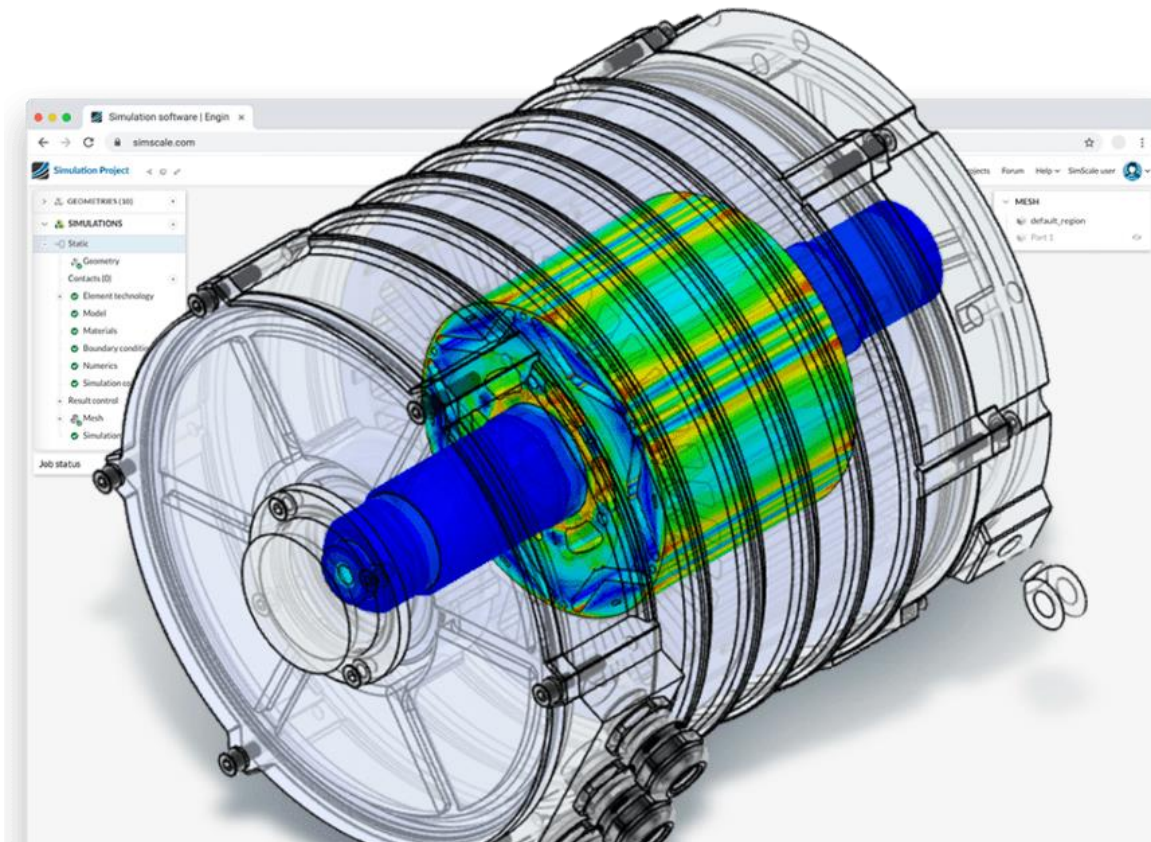
TRANSIENT STRUCTURAL ANALYSIS

To perform a transient structural analysis on a regular hexagonal prism using Structural Steel (since ANSYS does not allow custom materials in the transient system) with default linear elastic material properties. The prism is fully fixed at one end and subjected to four time-dependent forces of magnitude 281.69 N, each applied on a different long face and directed inward toward the prism. The analysis is carried out with a time step of 0.2 s to evaluate the variation of deformation and von Mises stress over time, and to obtain the maximum, minimum, and average structural response throughout the transient loading duration. To demonstrate mesh convergence.
**.**

STATIC STRUCTURAL ANALYSIS

For the Static Structural Analysis, I first selected the **Static Structural** option in ANSYS Workbench. Under **Engineering Data**, I created a material named as **project material** by setting the Young's modulus to **0.059 MPa**, which was calculated by taking the last two digits of my roll **(59)** number multiplied by 10^{-3}. The Poisson's ratio of the material was set to **0.28**.

I then opened **DesignModeler** and selected the **XY-plane** to sketch a hexagon. This sketch was extruded to create a prism with a length of 20 meters. After creating the geometry, the project material was assigned to the prism.
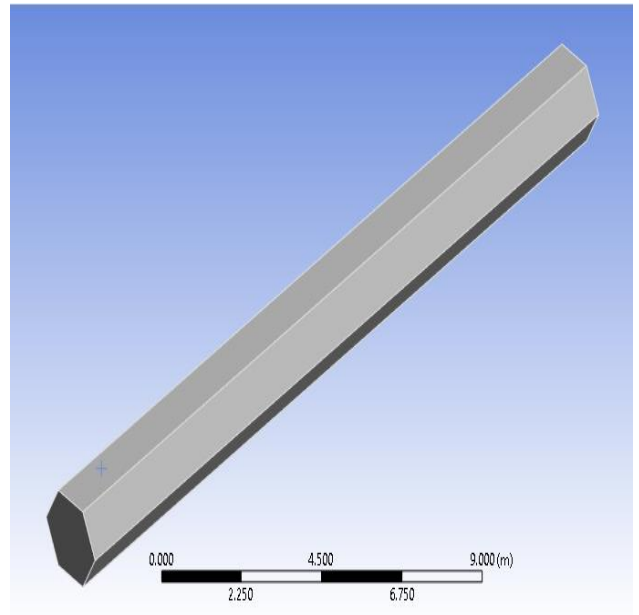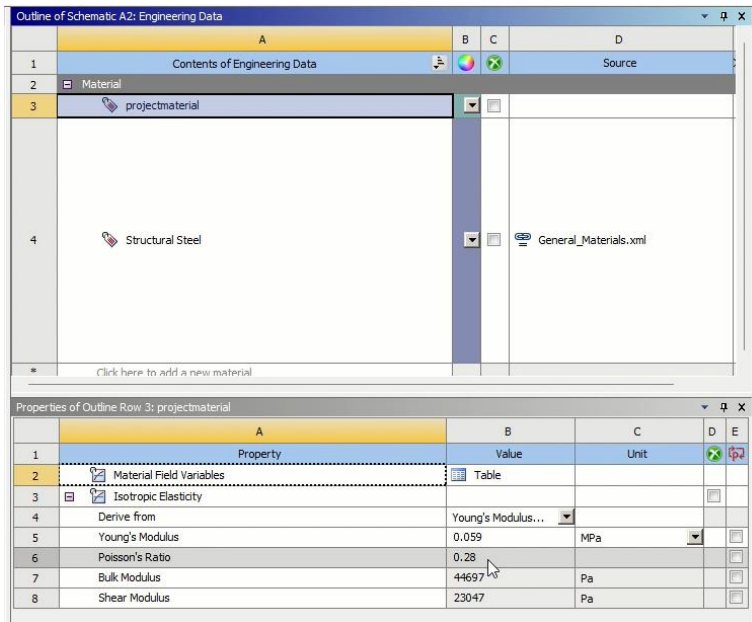

For the meshing process, I used the default mesh settings. The mesh method was set to **Tetrahedron** with a patch conforming approach. The element size and element order were kept at the program-controlled defaults. After generating the mesh, I examined the element quality graph to ensure the mesh quality was satisfactory.
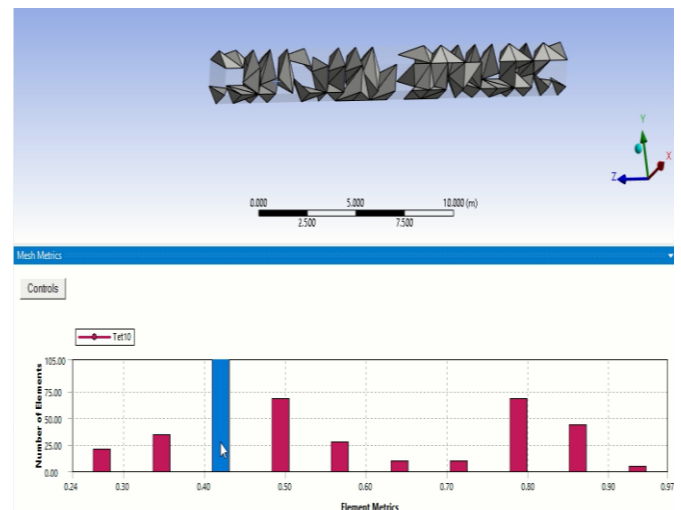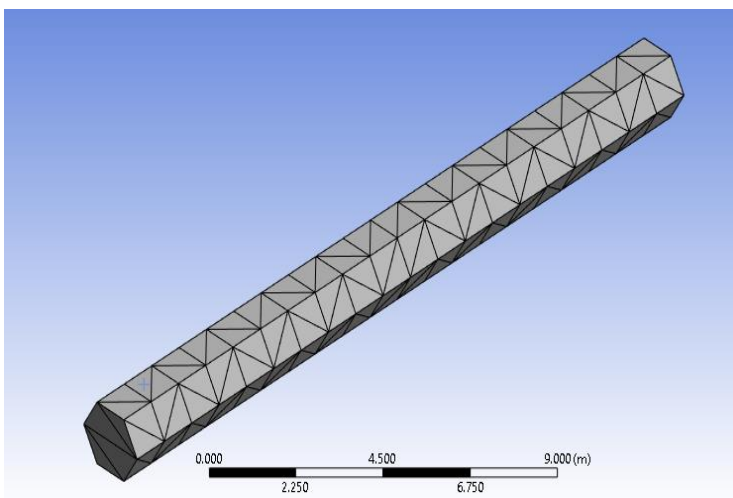
A **Fixed Support** was applied on one of the hexagonal end faces of the prism. A **Remote Force** of 2013 N was then applied at one-third of the prism's total length (20/3 = 6.666 m), located at 6.66 m from the fixed face. The force was applied in the **transverse direction**, perpendicular to the longitudinal axis of the prism, as shown in the simulation window.
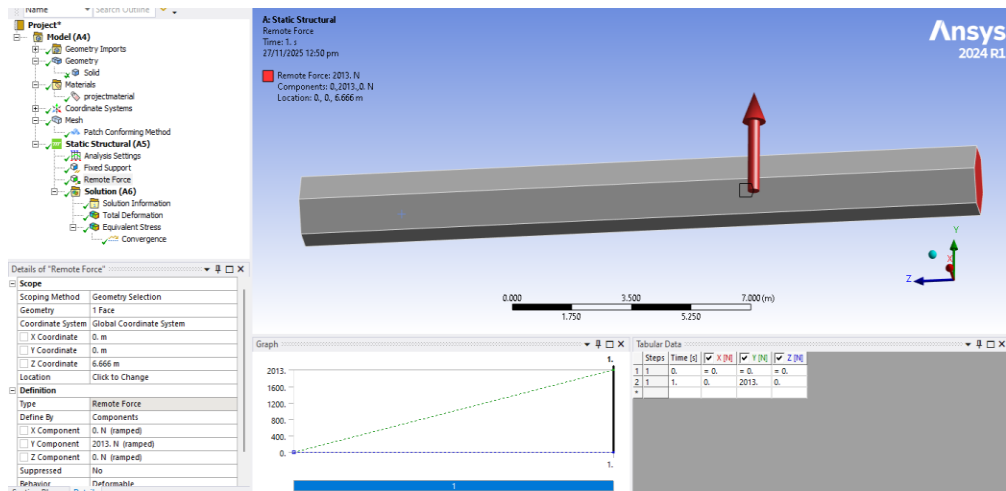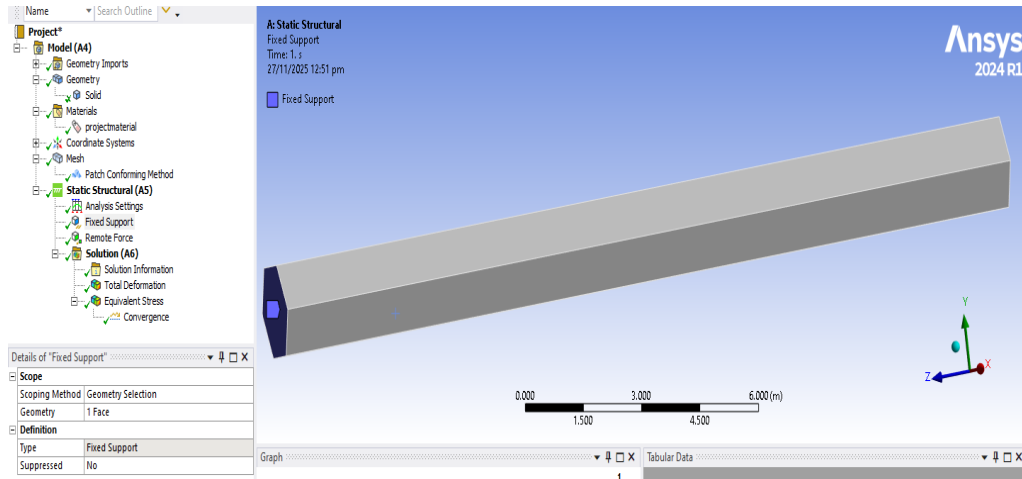
In the solution module, I included **Total Deformation** and **Equivalent von Mises Stress** as the key results to be evaluated. After solving, the deformation and stress results were captured.
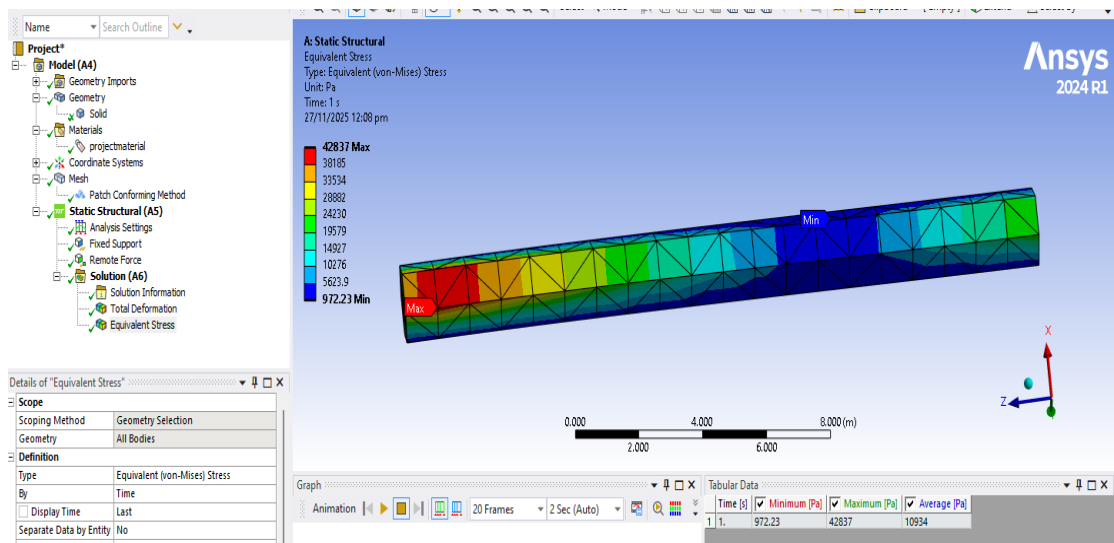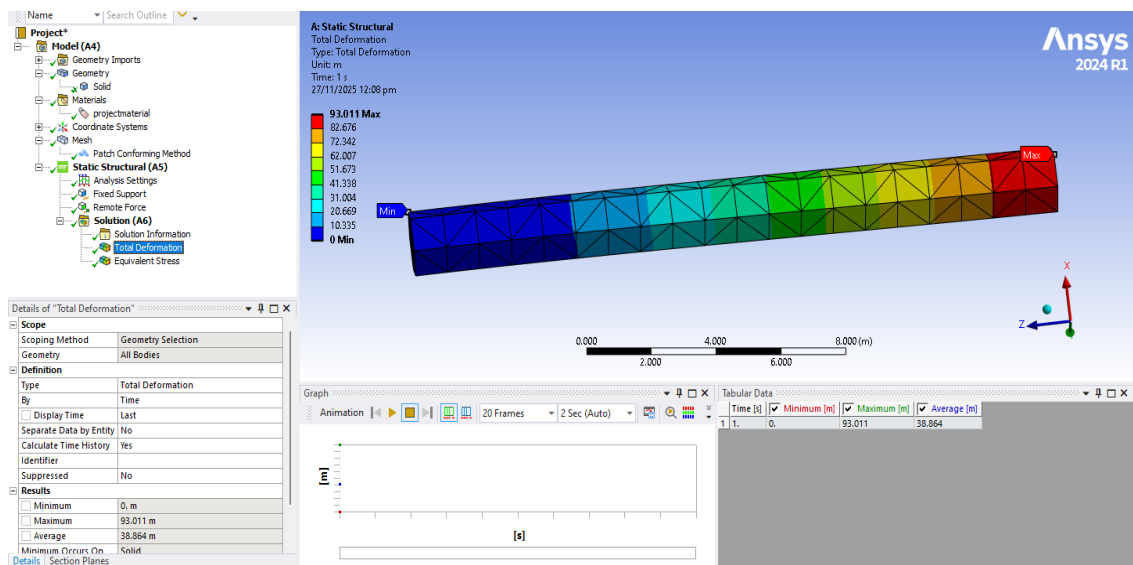
To ensure the accuracy of the results, a **mesh convergence study** was performed with an allowable change of **10%.** Two simulation runs were conducted using different element counts, specifically 387 elements for the first run and 1693 elements for the second. The convergence behavior of the equivalent stress was monitored, and the **corresponding graph** was recorded.
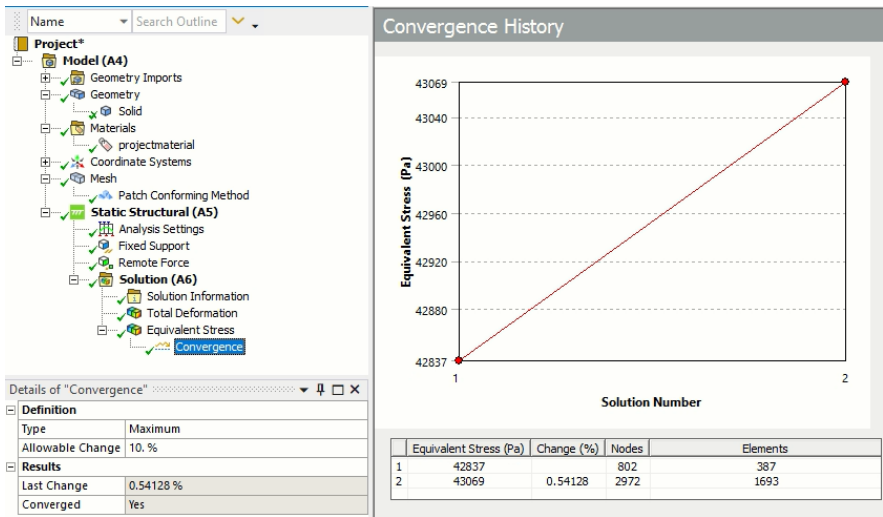
The estimated percent change is **0.54%** which is well below the allowable **10%.**

TRANSENT STRUCTURAL ANALYSIS
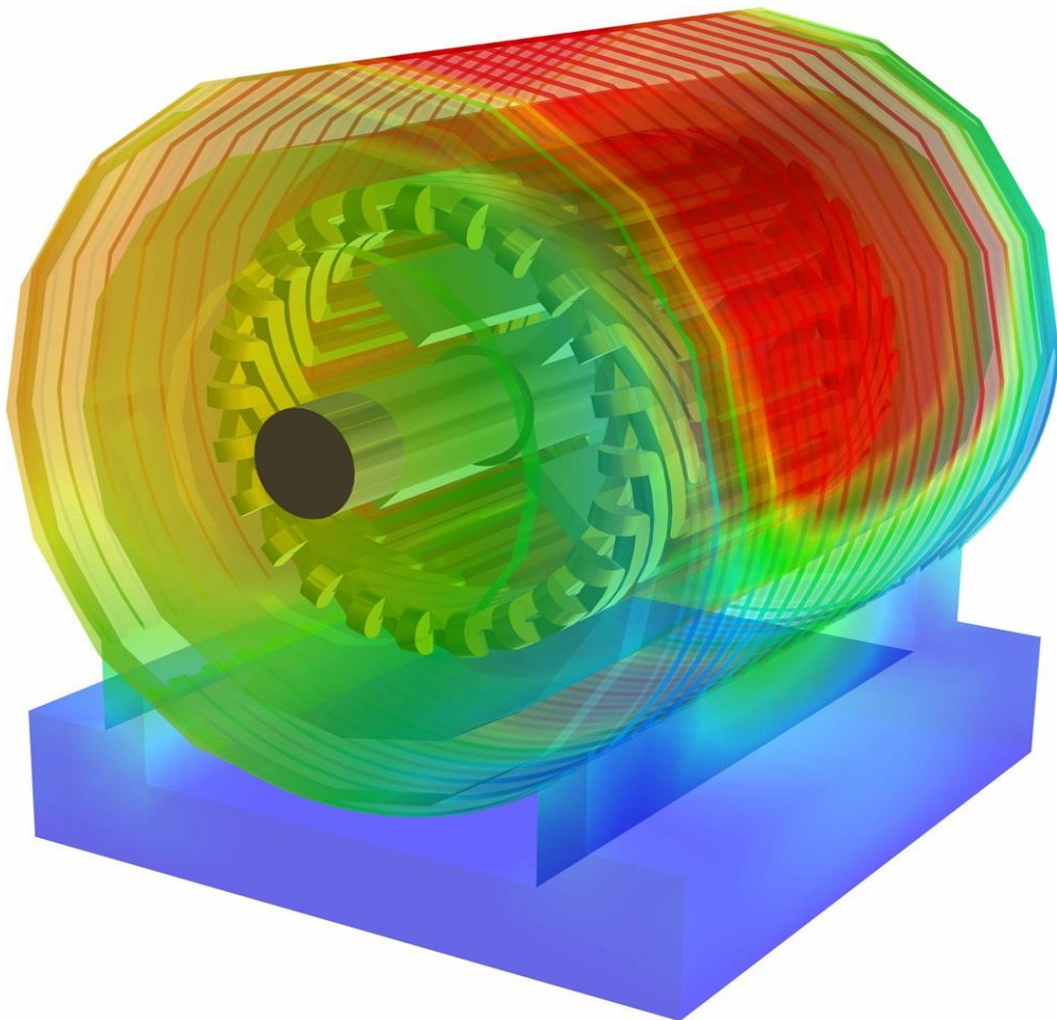
For the Transient Structural Analysis, the same geometry was used. Since ANSYS does not allow custom materials in the transient system, **Structural Steel** with its default Young's modulus and Poisson's ratio was selected.

The prism geometry remained unchanged, and four forces were applied on four of the long faces of the prism. These forces had a magnitude of **281.69 N**, corresponding to the USD to PKR exchange rate on the day of the simulation. Each force was directed inward toward the prism, and the time step for the transient analysis was set to **0.2 seconds**. The forces acting on each face and the corresponding time-varying graphs were recorded.
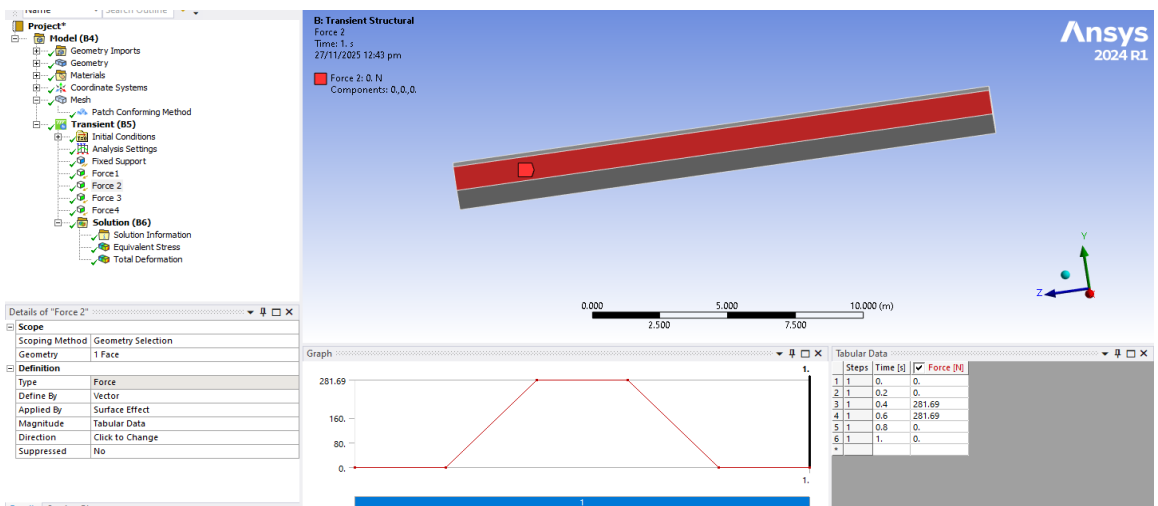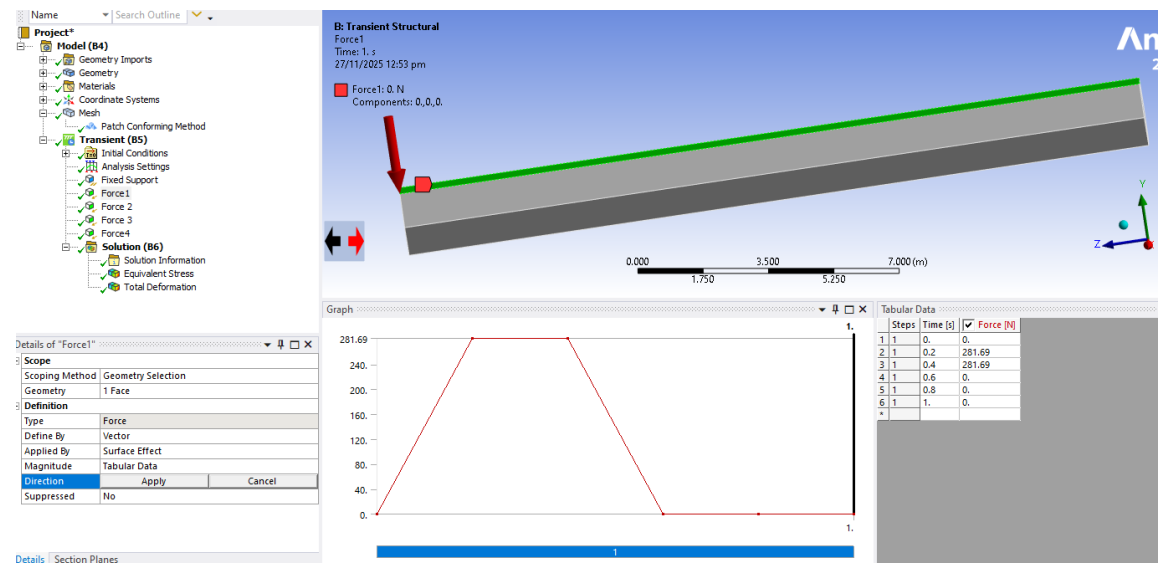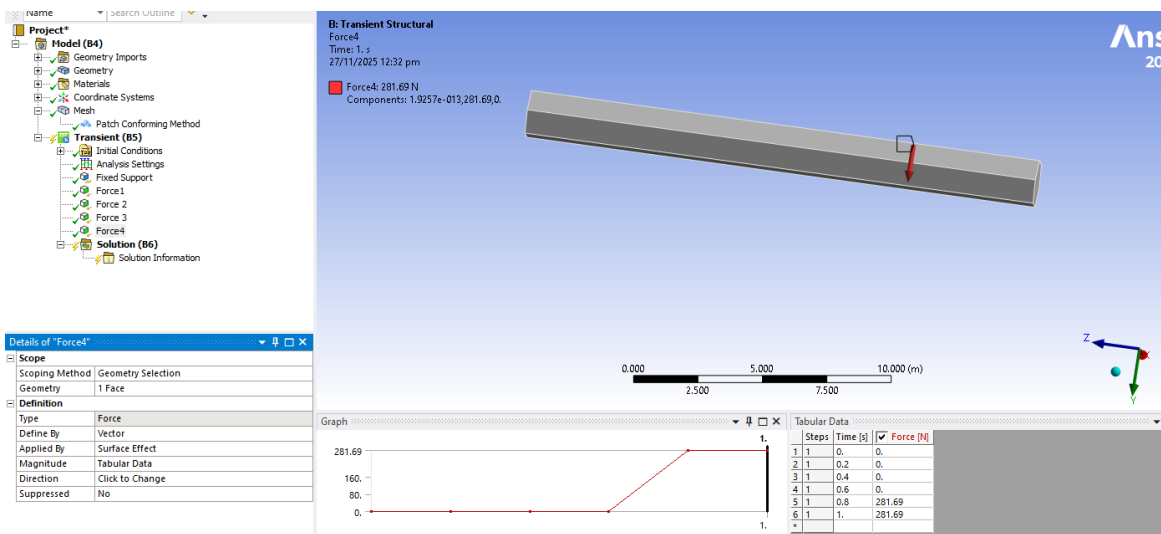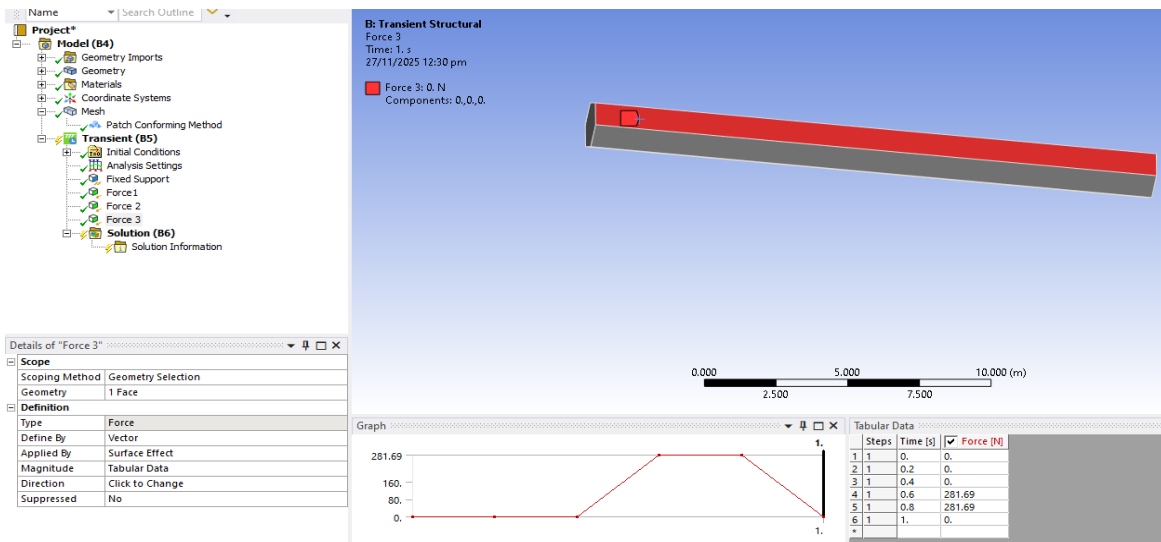
The mesh for the transient analysis was generated in the same manner as for the static case, and the mesh quality was also recorded.
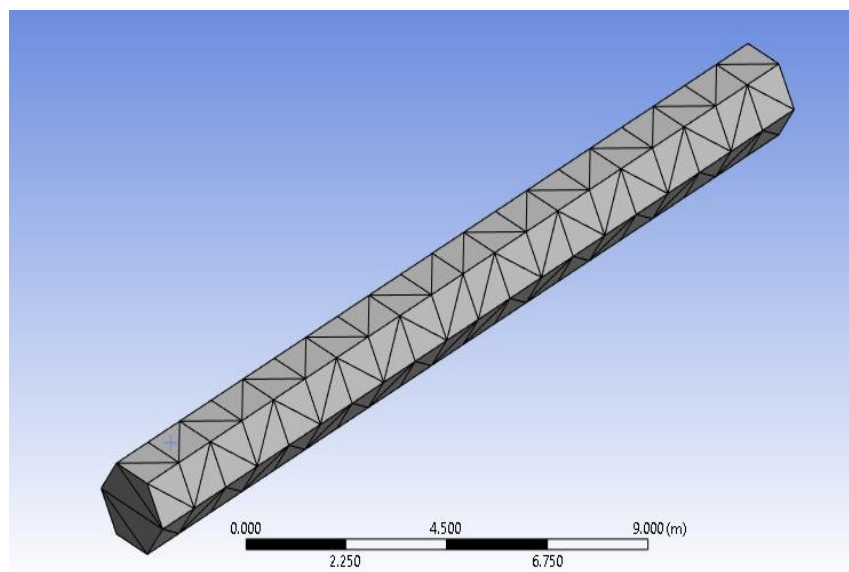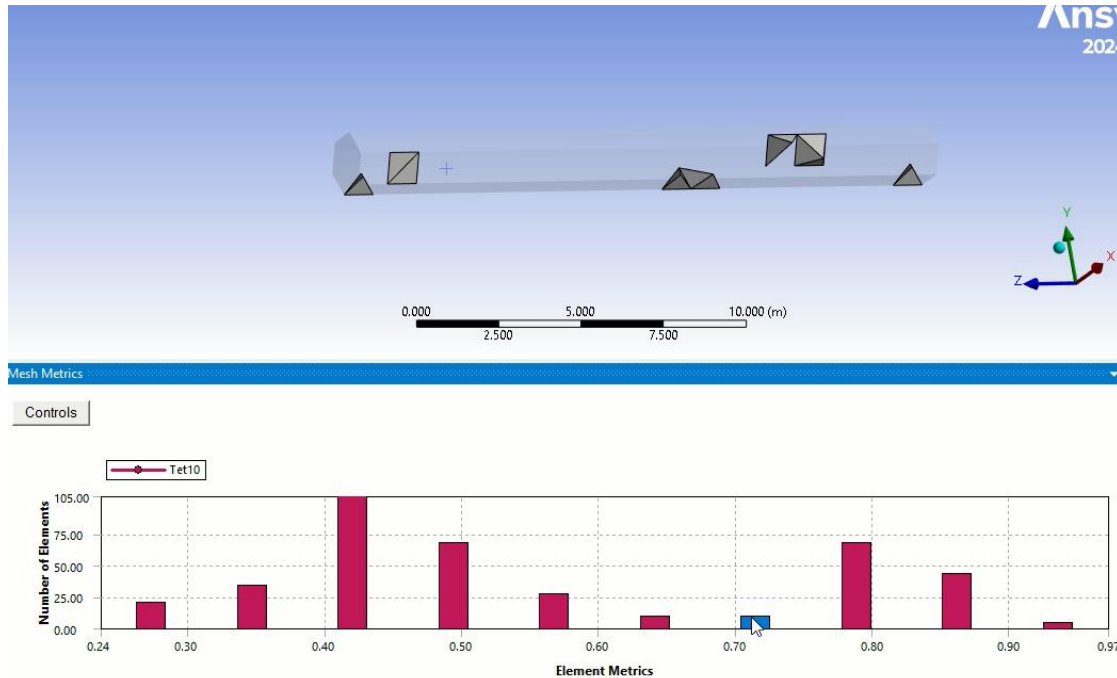
In the solution module, **Total Deformation** and **Equivalent von Mises Stress** were selected as the primary results.
For each recorded time step, ANSYS reported the **minimum**, **maximum**, and **average** stress values calculated across all nodes of the structure.

The minimum stress values represent regions experiencing negligible loading, while the maximum values correspond to the highly stressed zones, which are also visible in the contour plots as the red regions. The average stress provides a statistical indication of the overall stress distribution within the body at each instant.

The graphs for deformation and stress were recorded.
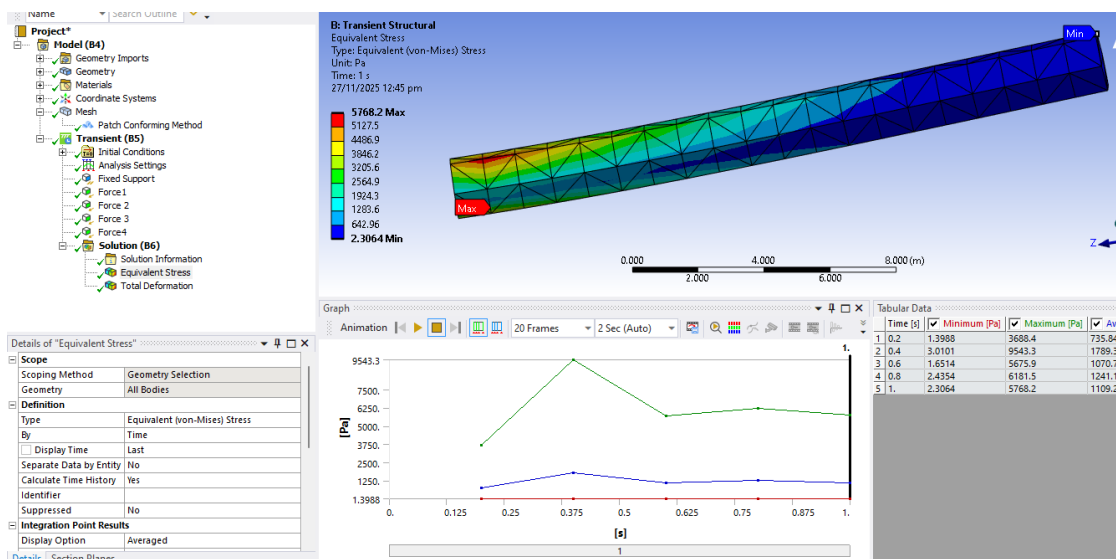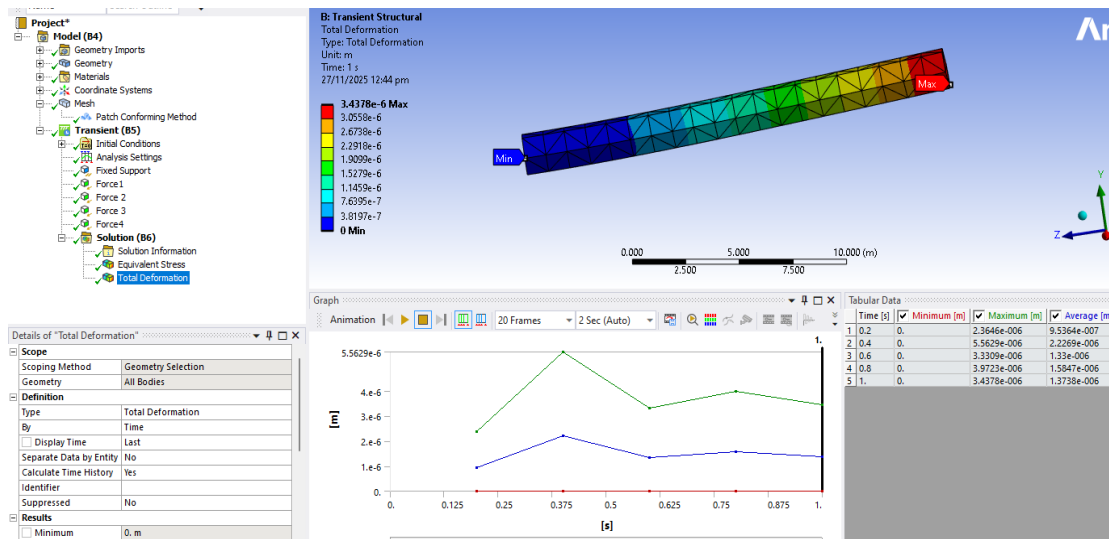
To ensure the accuracy of the results, a **mesh convergence study** was performed with an allowable change of **10%.** Two simulation runs were conducted using different element counts, specifically 387 elements for the first run and 1693 elements for the second. The convergence behavior of the equivalent stress was monitored, and the **corresponding graph** was recorded.

	Equivalent Stress (Pa)	Change (%)	Nodes	Elements
1	5768.2		802	387
2	5799.9	0.54128	2972	1693
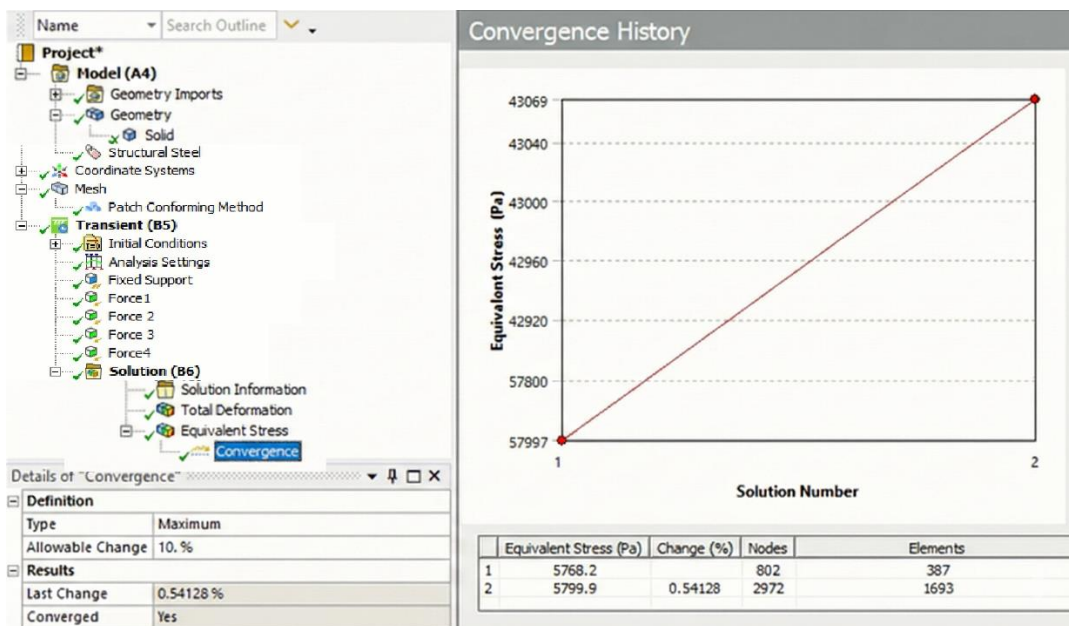
The estimated percent change is **0.55%** which is well below the allowable **10%.**