FEA Lab Project Report

Title: Static and Transient Structural Analysis of a Simple Polygon Structure

Course: Finite Element Analysis Lab

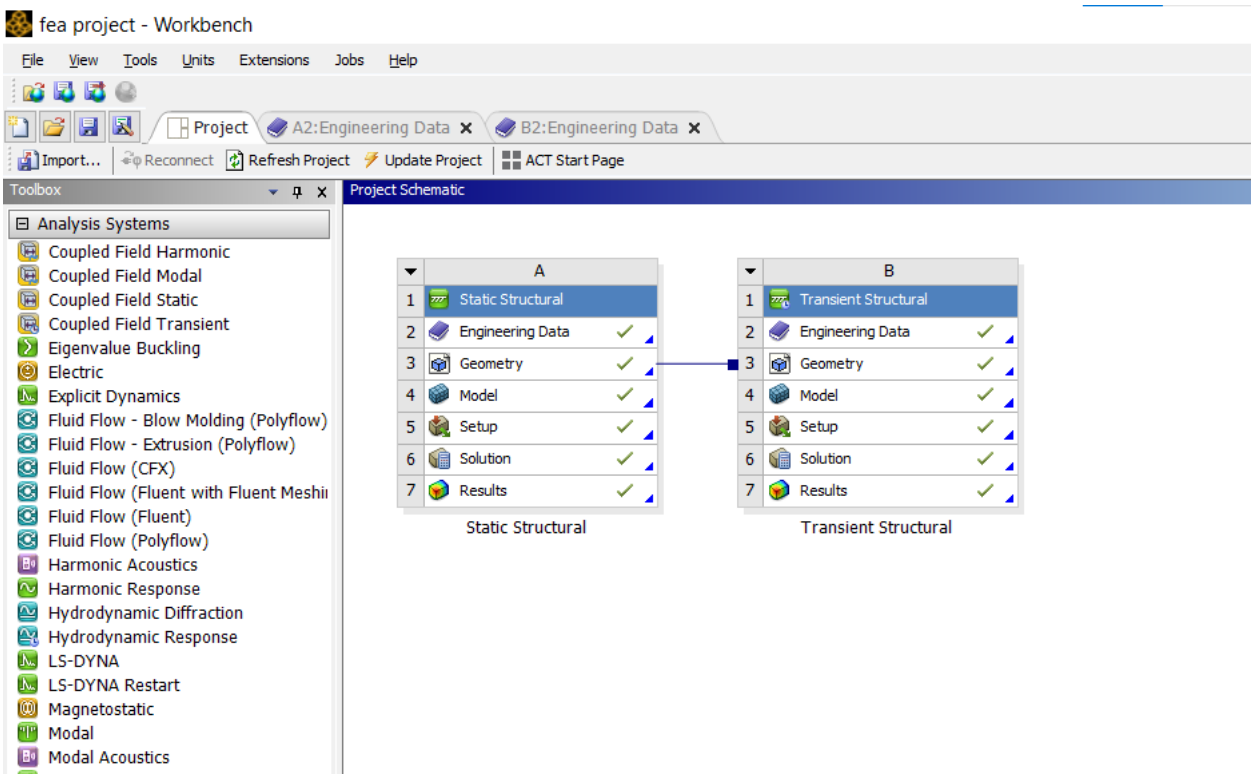
Student Name: Rafay Alam Zuberi

Roll Number: ME-1862

1. Introduction

The objective of this project is to perform Static and Transient structural simulations on a simple geometric structure using ANSYS. The material properties, boundary conditions, loading criteria, and simulation parameters are defined based on my roll number and date of birth.

This report documents the modeling procedure, material assignment, boundary conditions, meshing, simulation settings, mesh convergence study, and results for both static and explicit simulations.



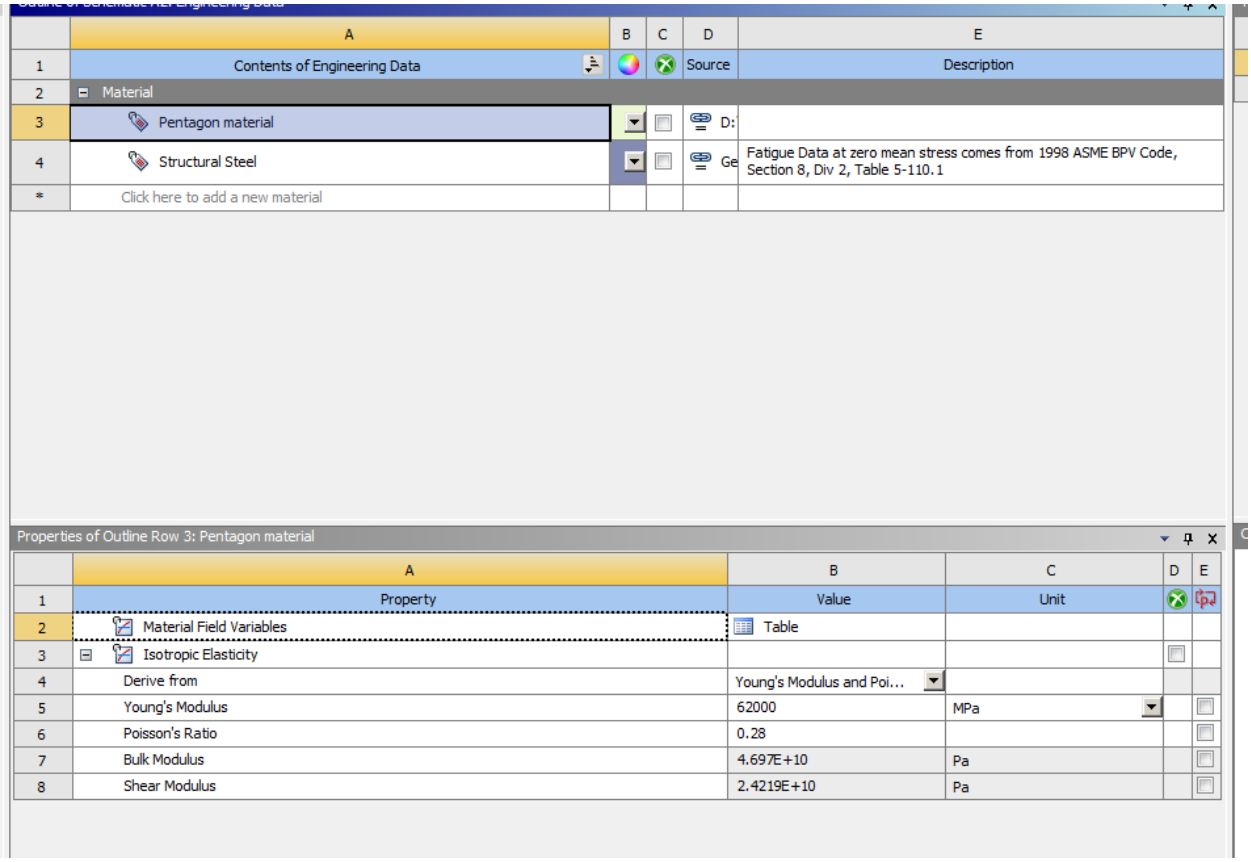
2. Problem Definition

2.1 Geometry Selection

- Selected Shape: (Pentagon, roll number 1862 – Even number)
- Geometry was created in Solidworks.

2.2 Material Properties

Material properties were assigned based on: - Elastic Modulus (E): 62× 10,000 MPa - Poisson's Ratio (v): 0.28



	A	B	C	D	E
1	Contents of Engineering Data			Source	Description
2	Material				
3	Pentagon material			D:	
4	Structural Steel			Ge	Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1
*	Click here to add a new material				

Properties of Outline Row 3: Pentagon material

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Young's Modulus and Poi...			
5	Young's Modulus	62000	MPa		
6	Poisson's Ratio	0.28			
7	Bulk Modulus	4.697E+10	Pa		
8	Shear Modulus	2.4219E+10	Pa		

2.3 Boundary Conditions

- One face of the pentagon is fully fixed.
- A Remote Force is applied at 1/3rd length of the fixed face.
- Magnitude of force = Sum of digits of Date of Birth i.e 2018 (6th July 2005)

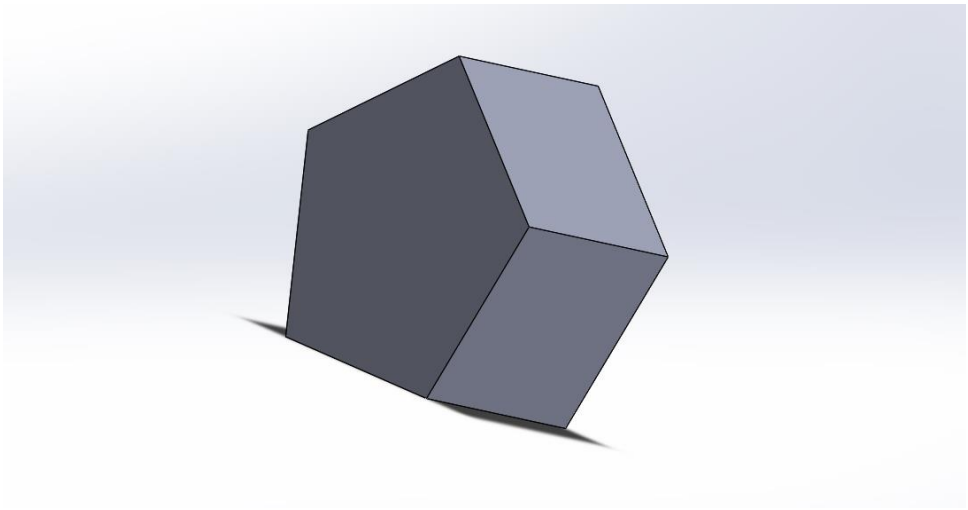
2.4 Explicit Analysis Loading Criteria

- Time step: 0.2 seconds
- Applied force = 280.66

---

3. Geometry and Modeling

A Pentagon was sketched and extruded to required thickness. The orientation was selected such that one complete face lies horizontally for fixing and ease of force application.

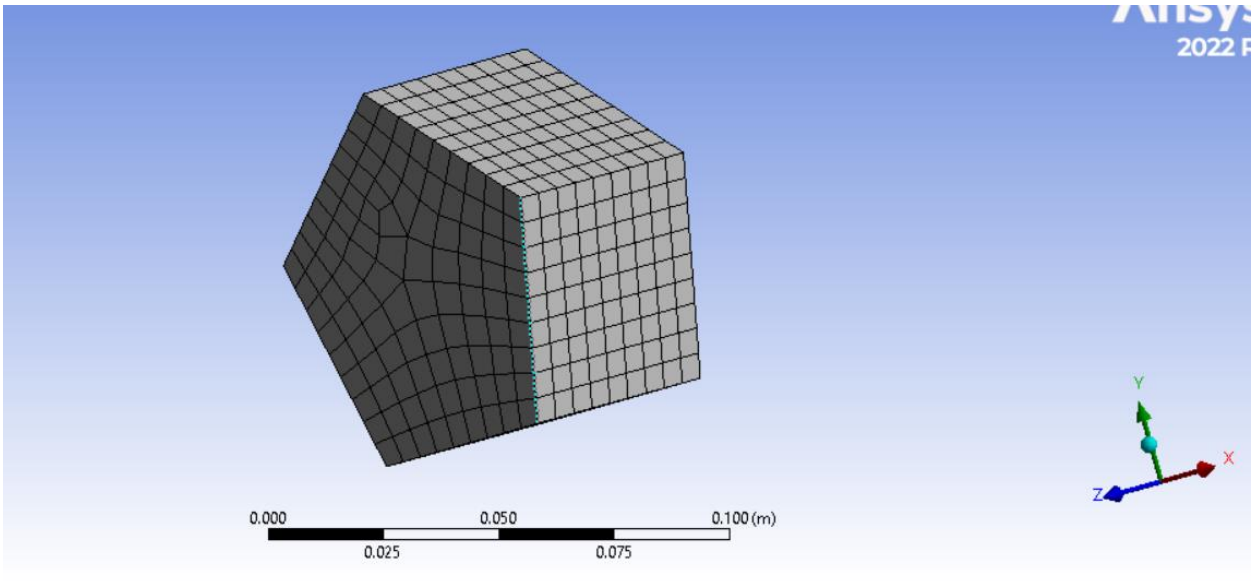


---

4. Meshing

Meshing plays a crucial role in the accuracy of FEA simulations. The following mesh types were used: - Element Type: hexagonal

A mesh convergence study was performed for static structural analysis only as there isn't any option of mesh convergence in transient structural analysis.

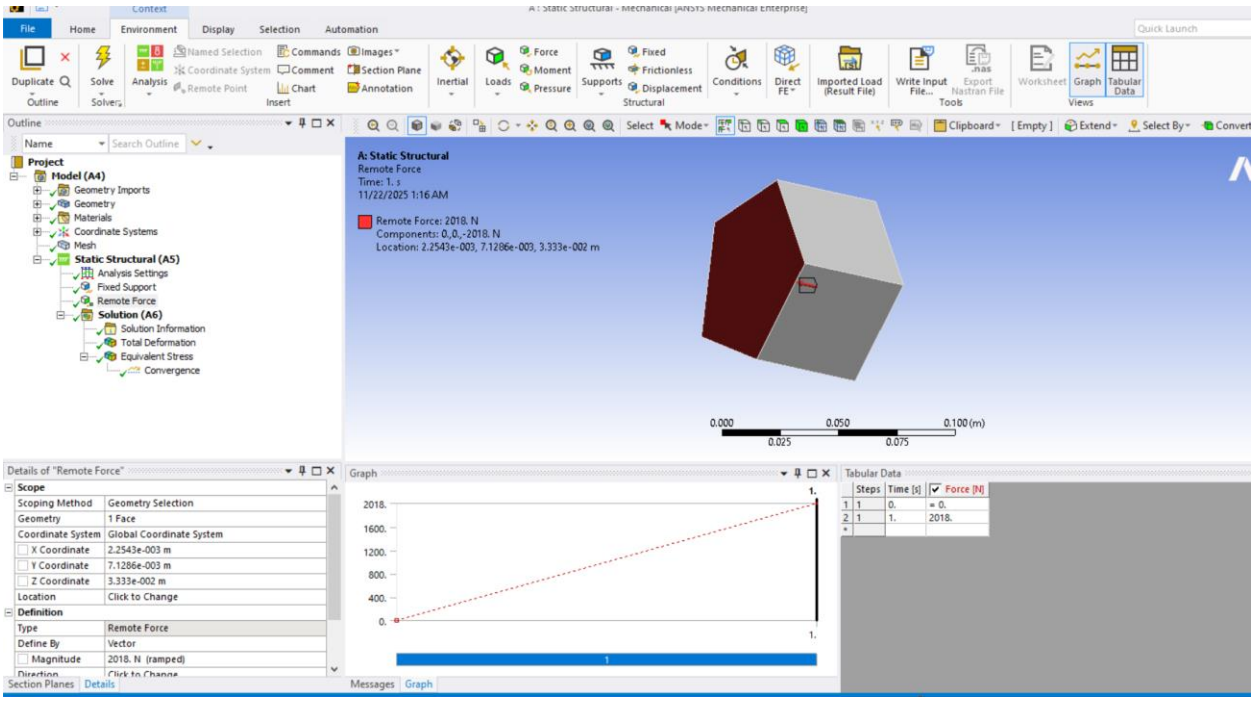


---

5. Static Structural Simulation

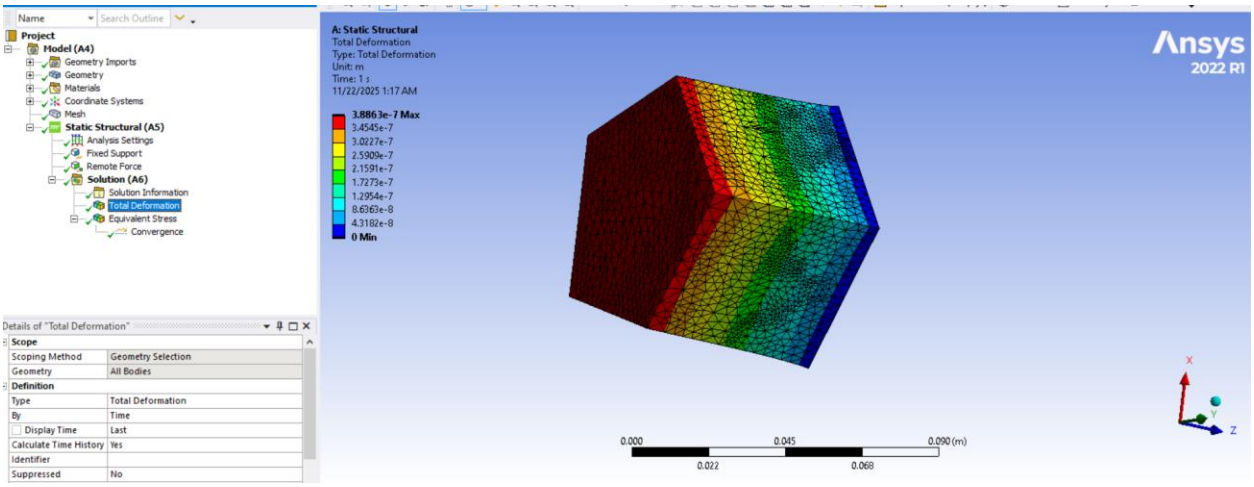
5.1 Boundary Conditions

- **Fixed Support:** One face fully constrained.
- **Remote Force:** Applied at 1/3rd region of fixed face.

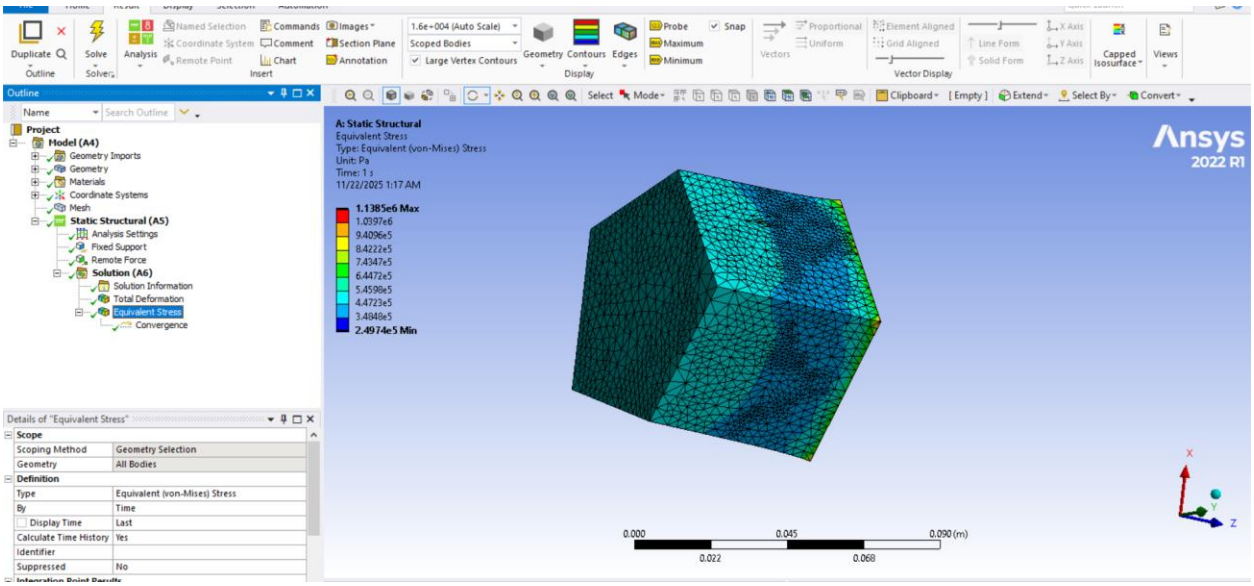


5.2 Results

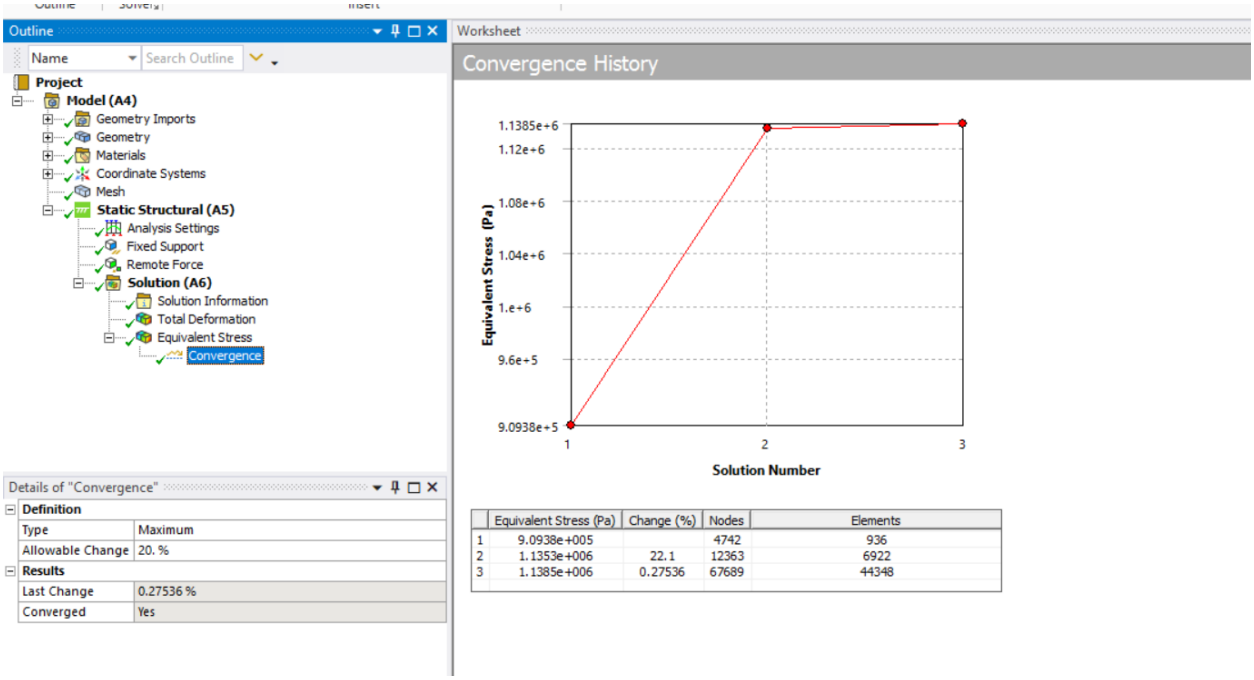
5.2.1 Total Deformation



5.2.2 Equivalent Stress (von Mises)



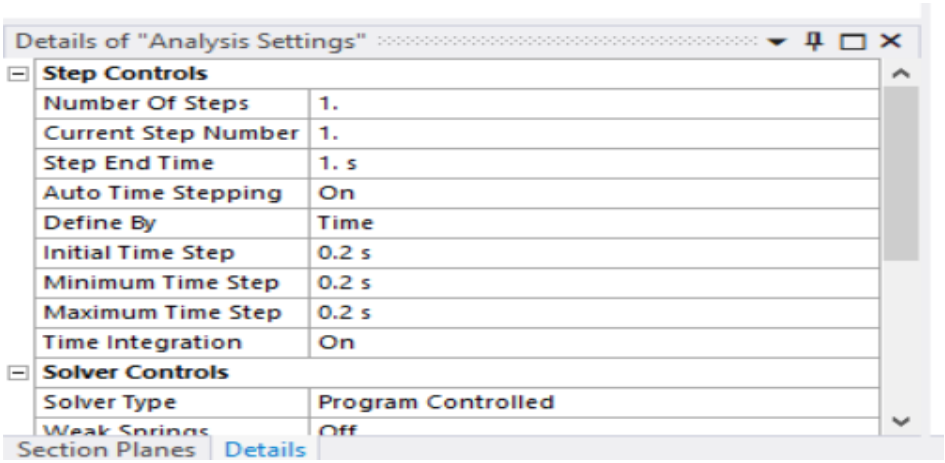
5.3 Mesh Convergence (Static)



6. Transient Structural Simulation

6.1 Loading and Time Setup

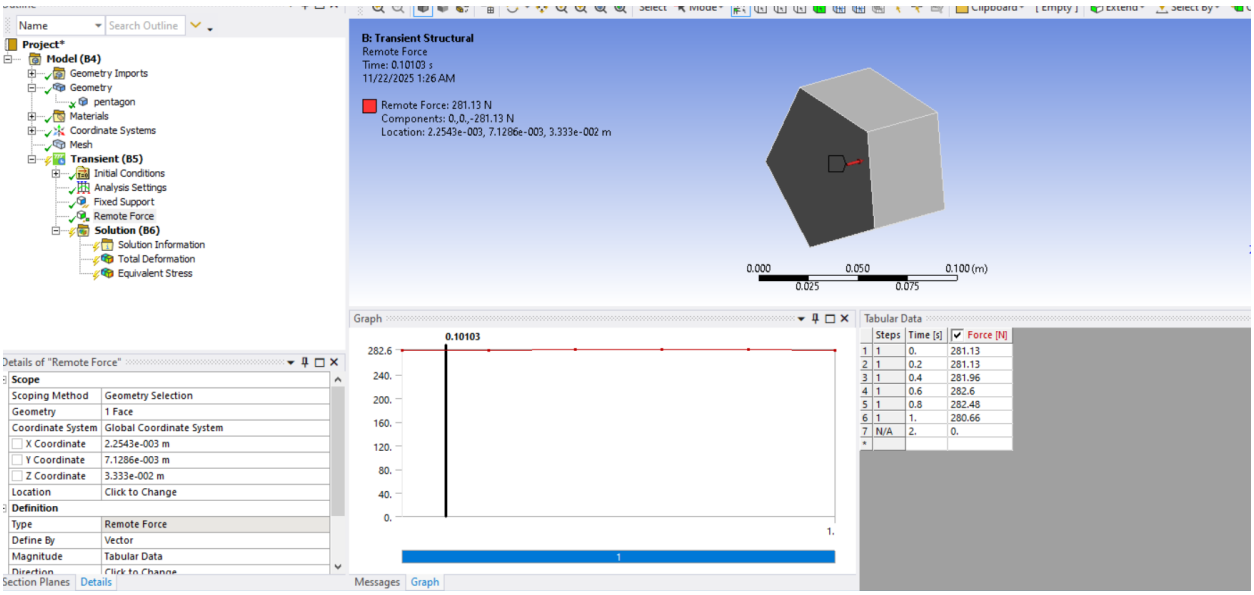
For this project, the transient (explicit type) simulation was performed using 5 time steps, each having a step size of 0.2 seconds, resulting in a total simulation time of 1.0 second.



Force Input Method

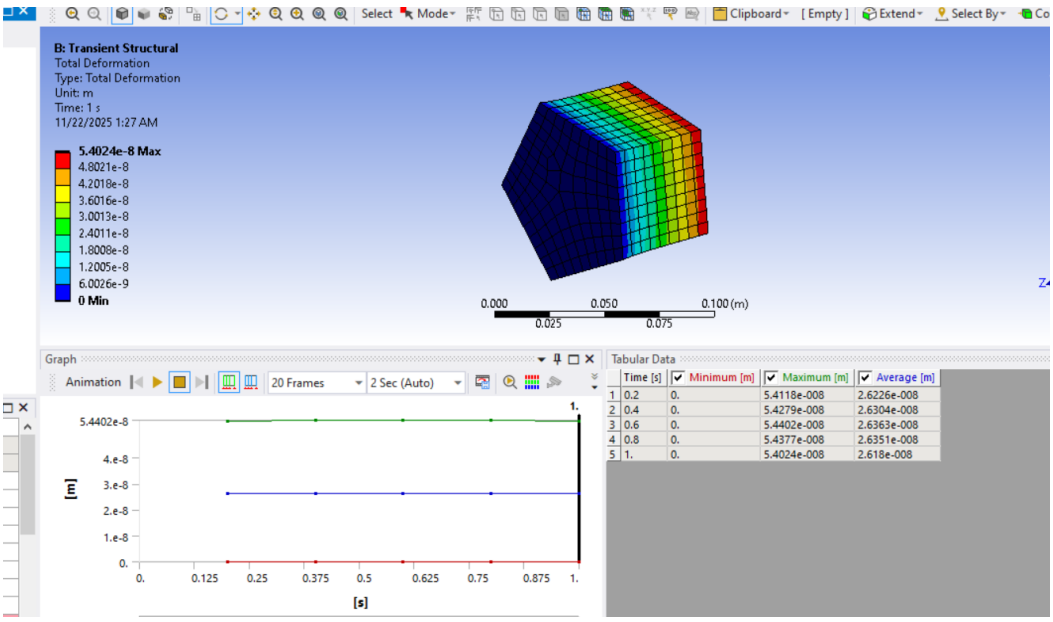
Instead of a single constant force, the applied force at each time step was taken from the peak force values extracted from the USD to PKR exchange-rate graph of the day.

These peak values were mapped into the load curve for the 5 simulation steps.

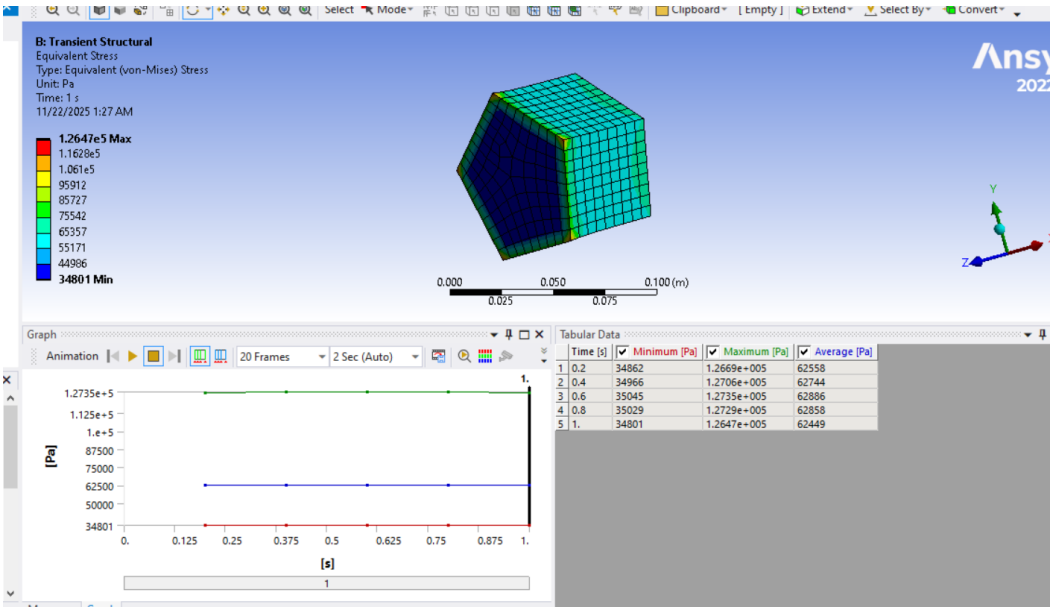


6.2 Results

6.2.1 Deformation vs Time



6.2.2 Stress-Time Response



6.2.3 Strain Energy