

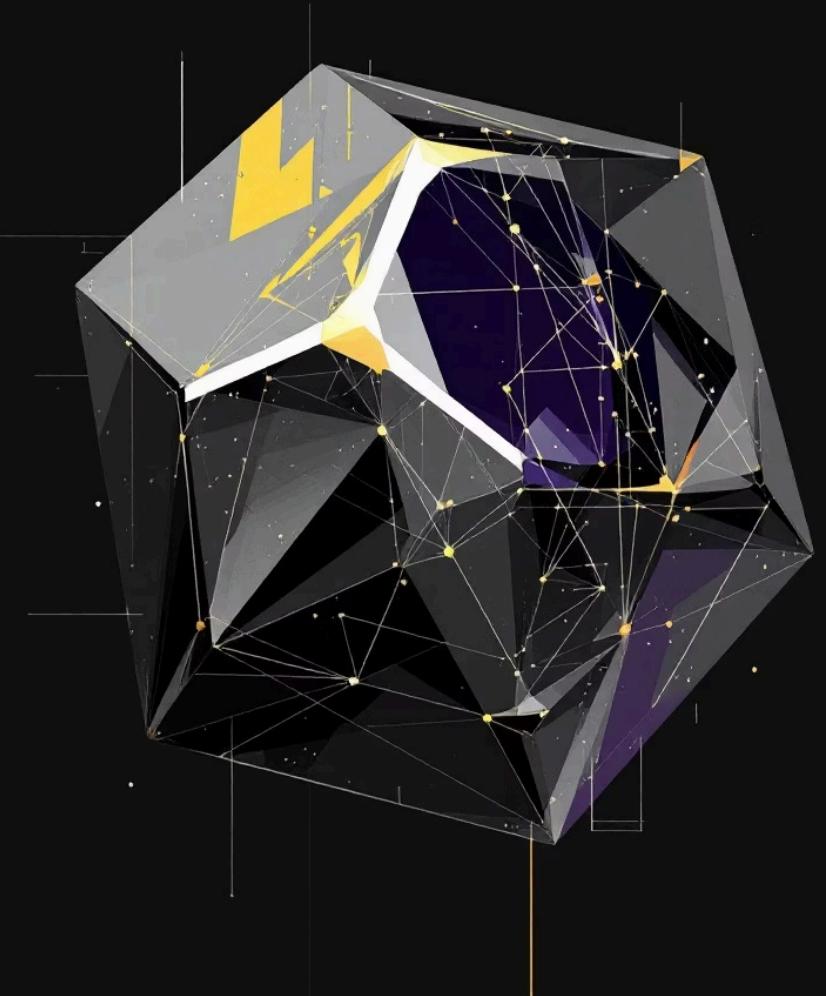
# FEA Project: Static & Transient Structural Analysis of a Hexagonal Prism

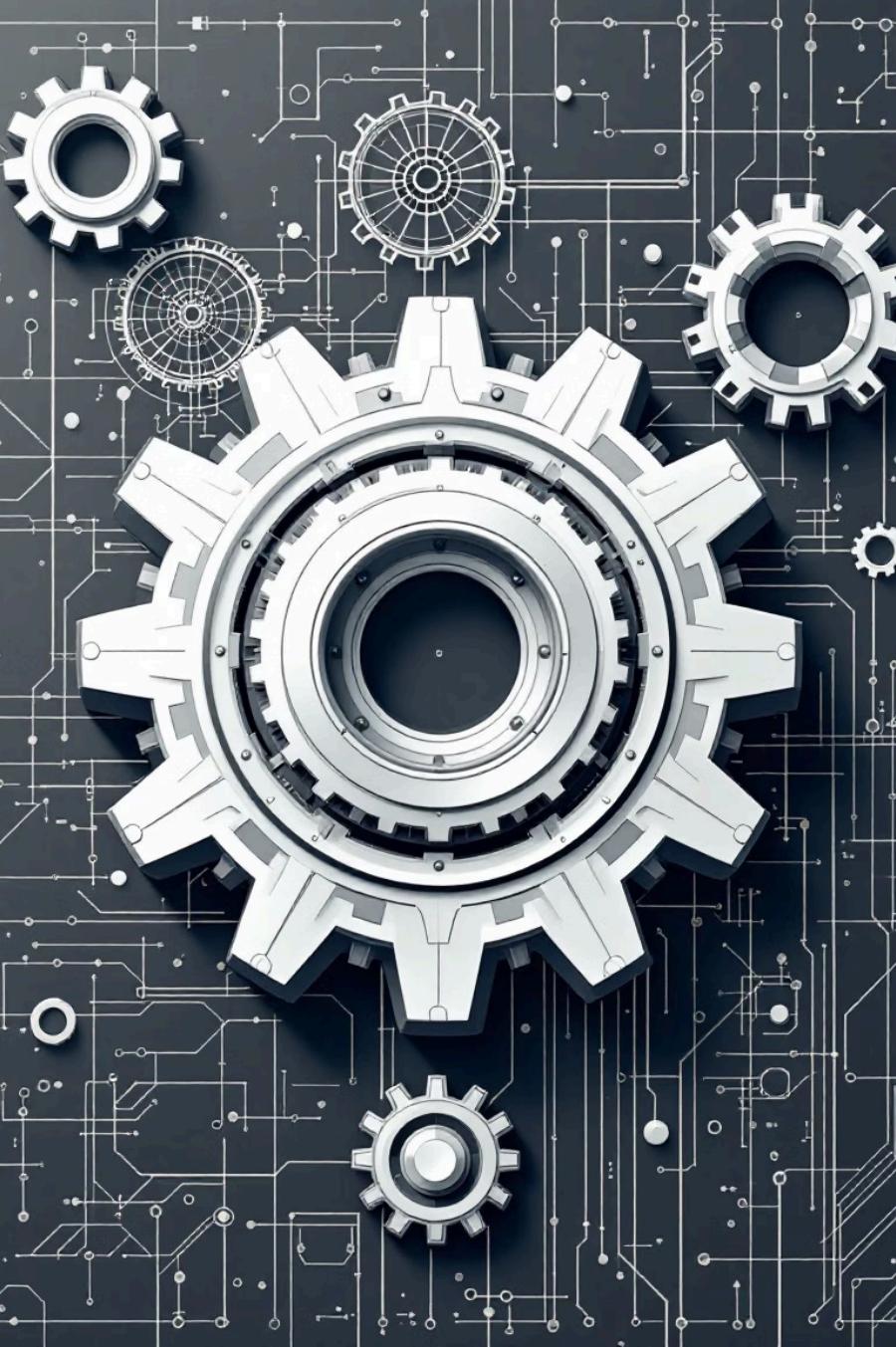
Performed in ANSYS Workbench

By Arshia Farooqui

Semester VII, Sec B, ME-1859 – Finite Element Analysis Lab

Submitted to: Sir Affan Ahmed Khan





# Project Overview

01

---

## Static Structural Analysis

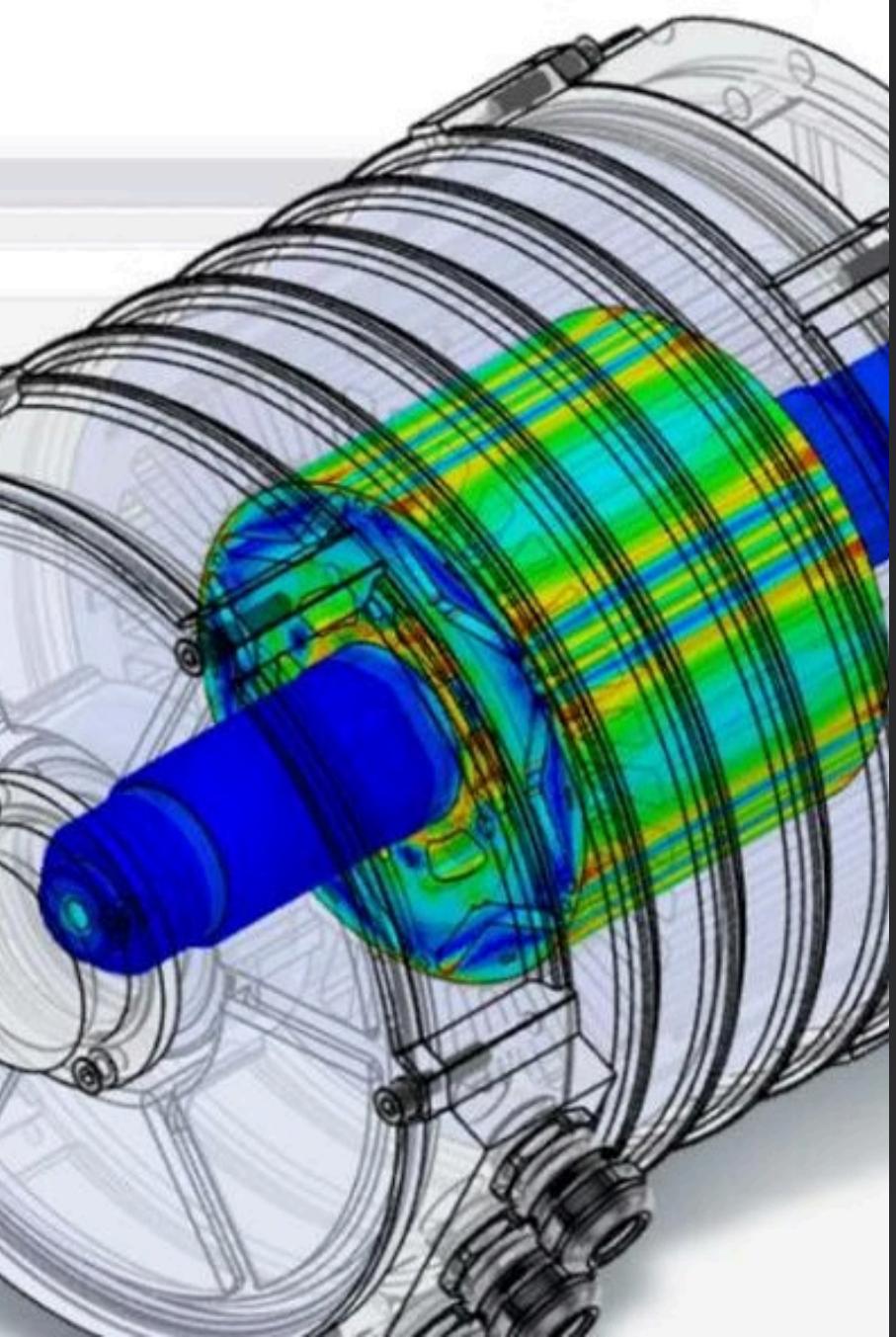
Detailed examination of a hexagonal prism under static loads.

02

---

## Transient Structural Analysis

Investigation of the prism's behavior under time-dependent forces.



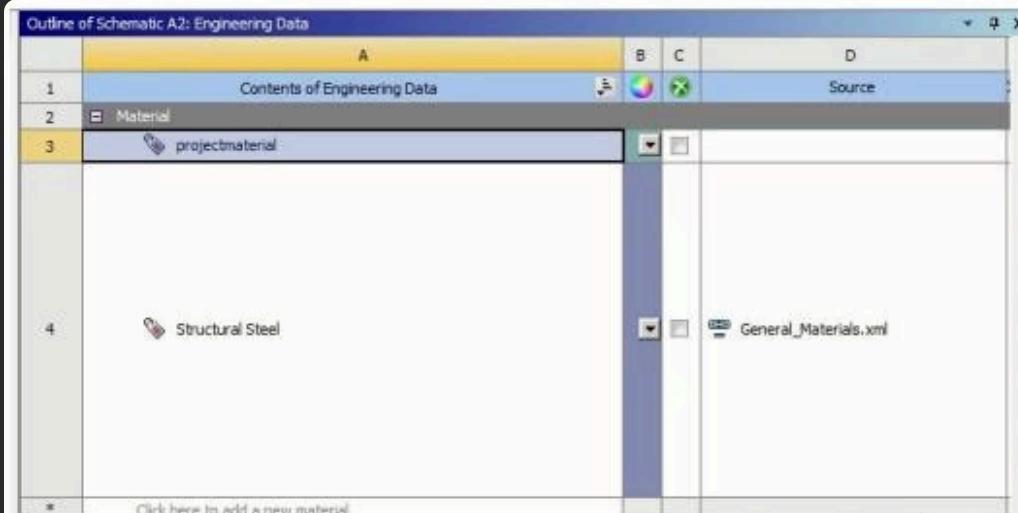
# STATIC STRUCTURAL ANALYSIS

This section details the methodology and results of the static structural analysis performed on the hexagonal prism.

# Material & Geometry Definition

## Custom Material: "Project Material"

- Elastic Modulus (E): 0.059 MPa
- Poisson's Ratio ( $\nu$ ): 0.28

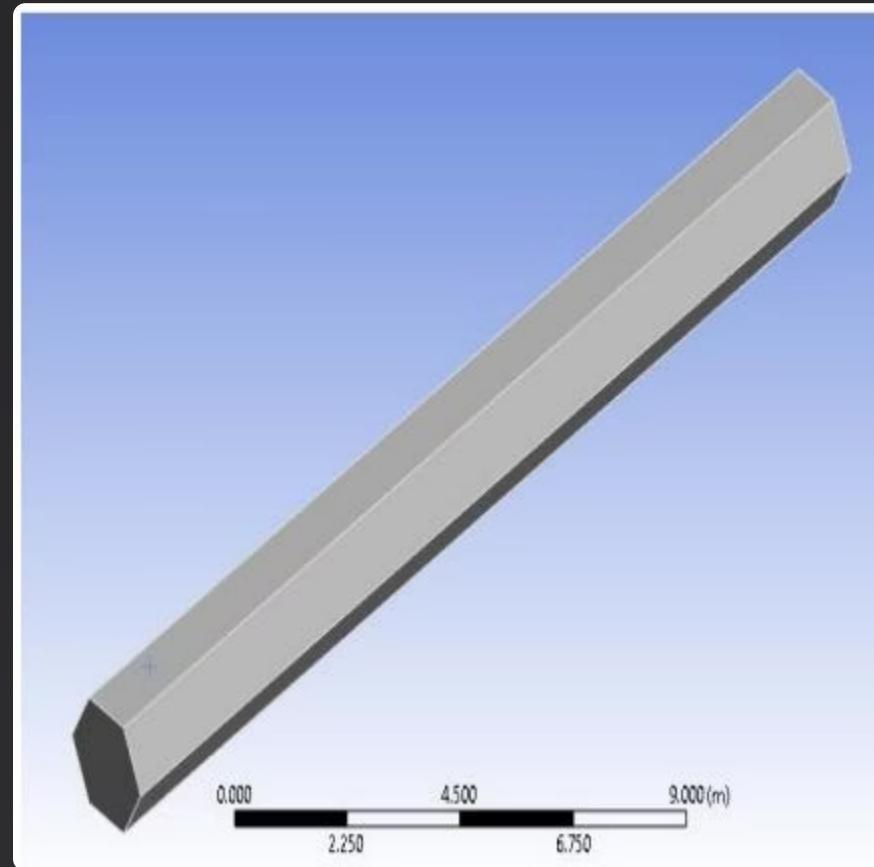


The screenshot shows the 'Outline of Schematic A2: Engineering Data' window. The 'projectmaterial' entry is selected in the list, which is categorized under 'Material'. Below the list, there is a note: 'Click here to add a new material'. At the bottom, there is a detailed table titled 'Properties of Outline Row 3: projectmaterial'.

| A                        | B                   | C    | D               | E |
|--------------------------|---------------------|------|-----------------|---|
| Property                 | Value               | Unit | [Unit dropdown] |   |
| Material Field Variables | Table               |      |                 |   |
| Isotropic Elasticity     |                     |      |                 |   |
| Derive from              | Young's Modulus...  |      |                 |   |
| Young's Modulus          | 0.059               | MPa  |                 |   |
| Poisson's Ratio          | 0.28                |      |                 |   |
| Bulk Modulus             | 44697 $\times 10^9$ | Pa   |                 |   |
| Shear Modulus            | 23047               | Pa   |                 |   |

## Hexagonal Prism Geometry

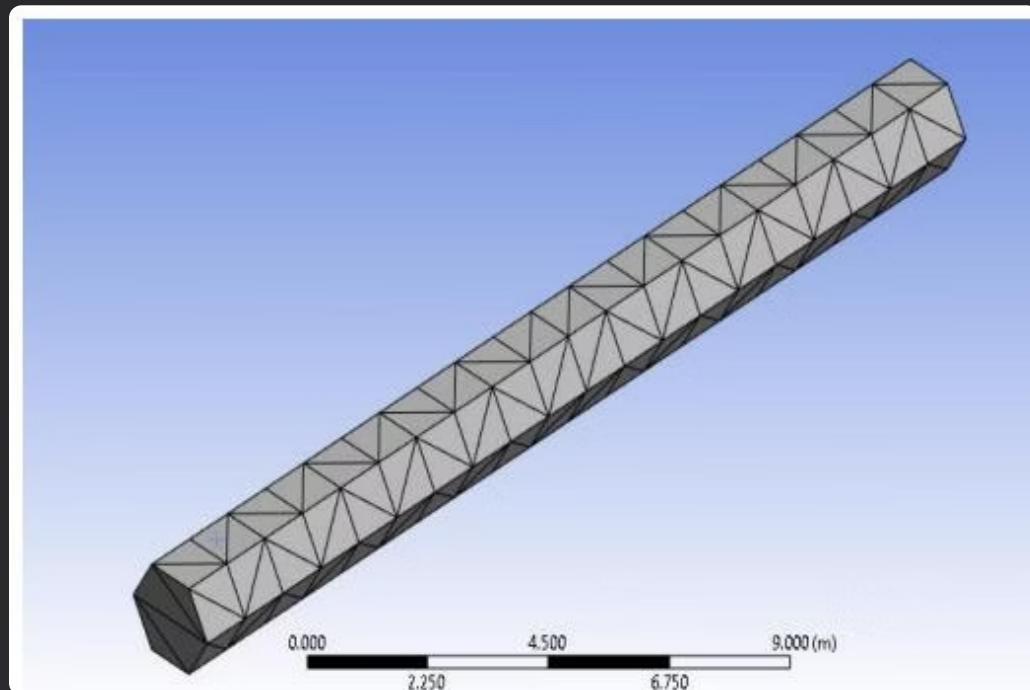
- Regular hexagonal base
- Extruded height: 20 meters



# Meshering & Boundary Conditions (Static)

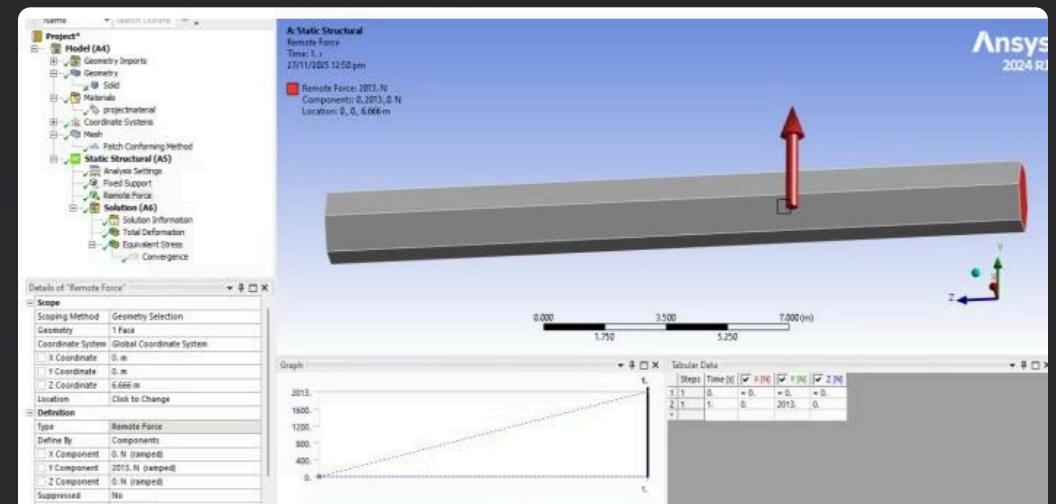
## Meshing Details

- Method: Tetrahedrons (Patch Conforming)
- Element Size & Order: Program-controlled (default)
- Element Quality: Satisfactory histogram confirmed



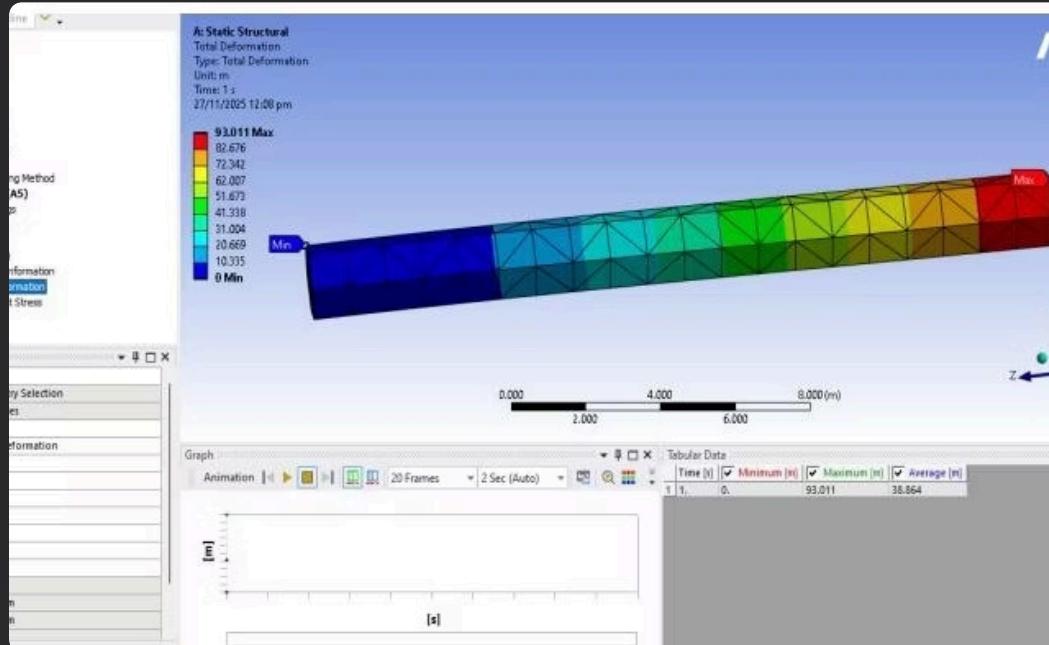
## Boundary Conditions

- Fixed Support: Applied to one end face
- Remote Force: 2013 N
- Location: 6.67 m from fixed face (1/3rd height)
- Direction: Transverse (perpendicular to prism axis)



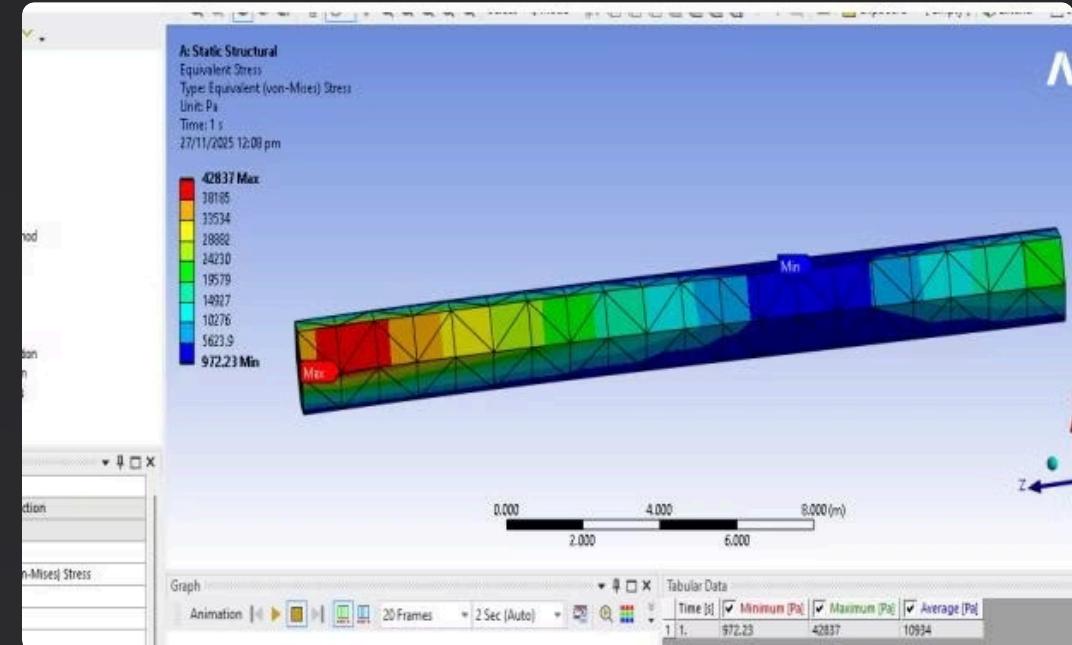
# Static Analysis Results

The model was successfully solved, yielding the following key outputs:



## Total Deformation

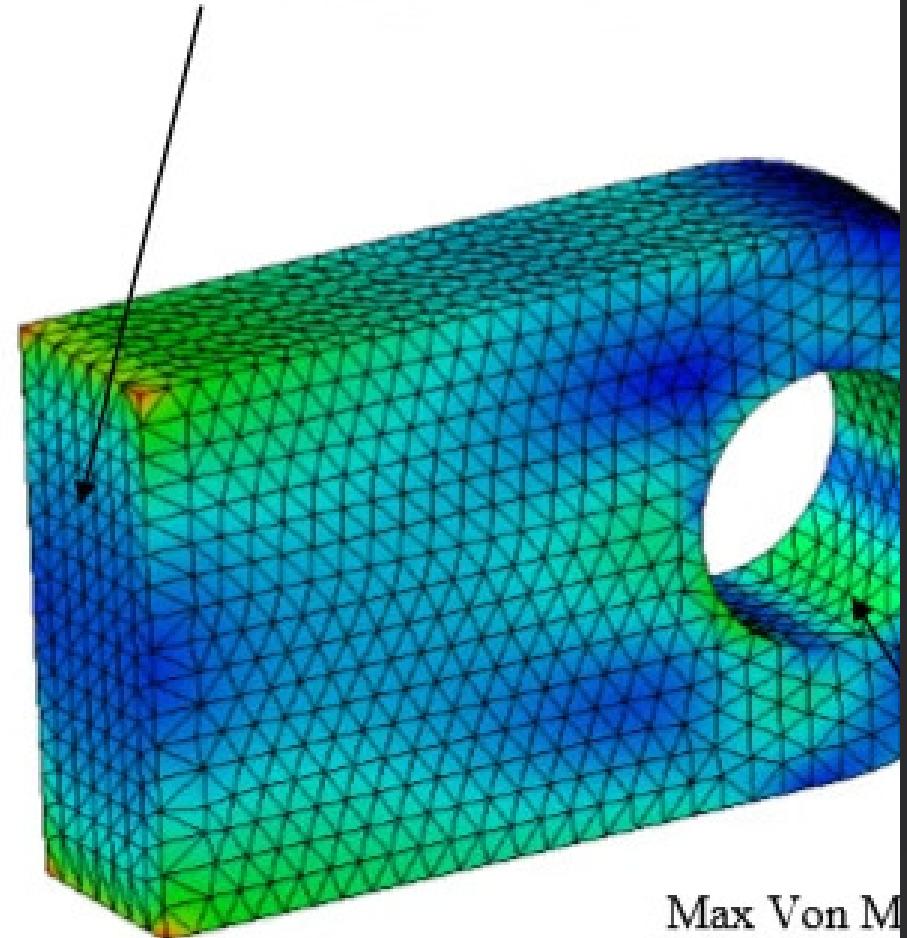
Visualizing the overall displacement under applied loads.



## Equivalent (von-Mises) Stress

Analyzing stress distribution within the material.

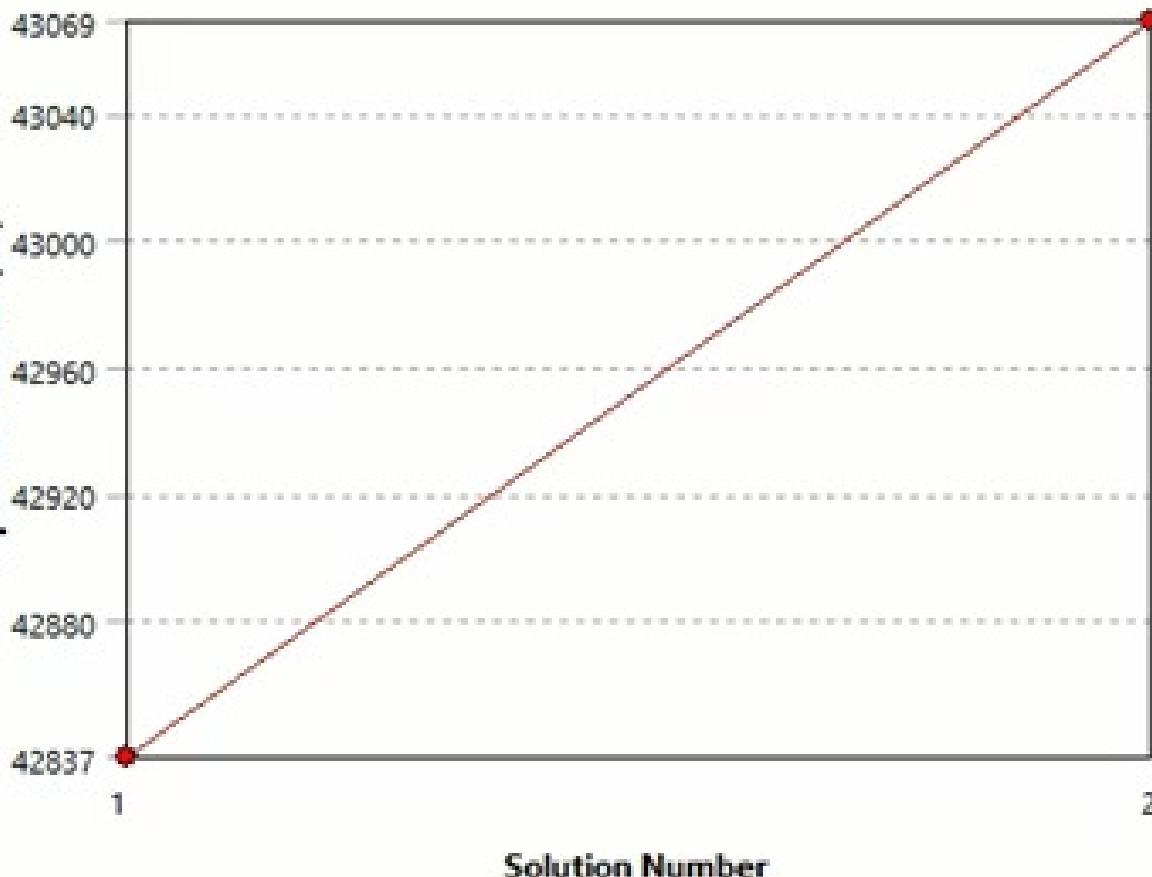
Max Von Mises Stress  
at attachment surface



Max Von Mises Stress  
at bottom surface

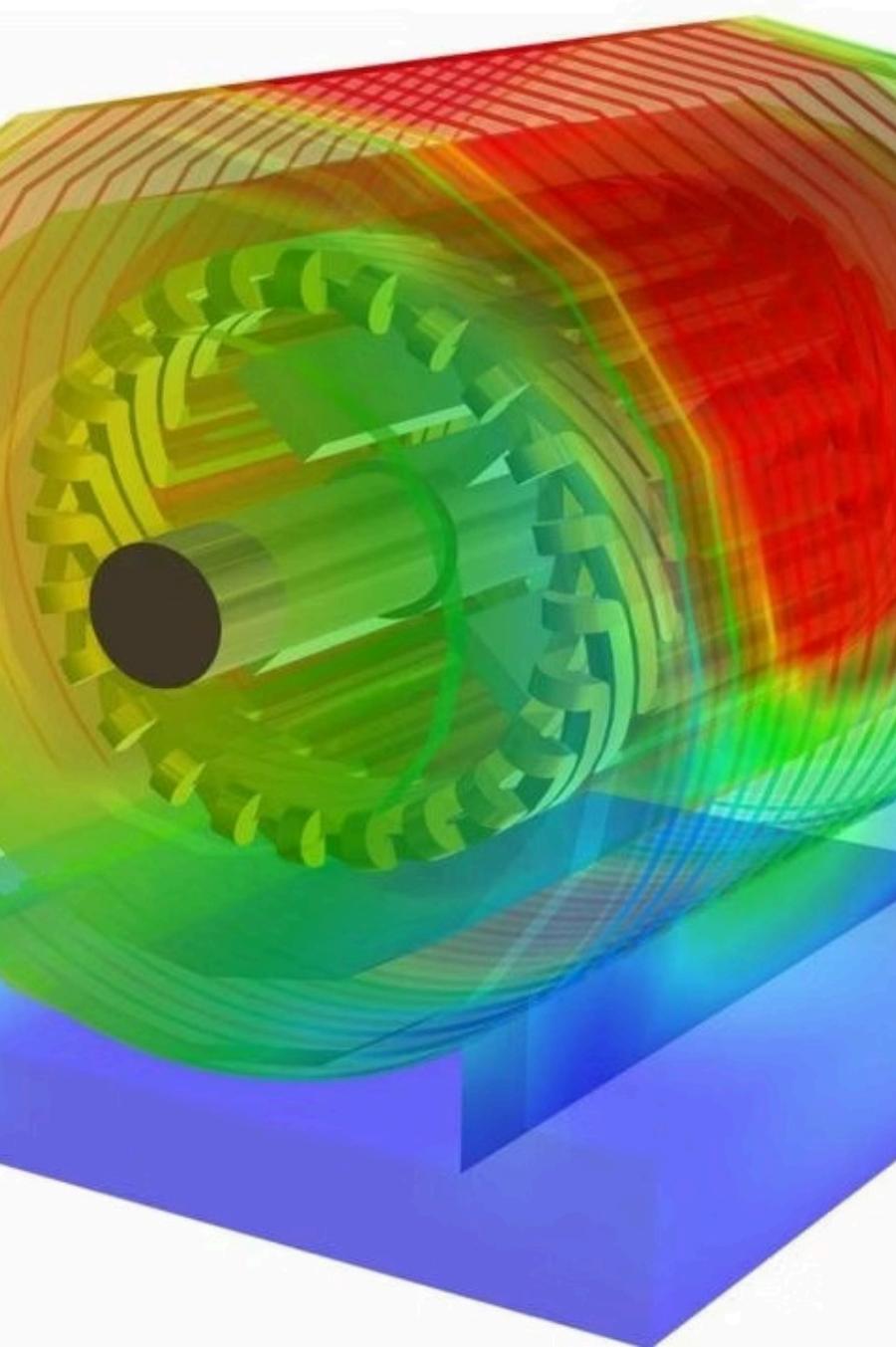
## Mesh Convergence (Static)

## Convergence History



- Two mesh densities: Coarse (387 elements) and Fine (1693 elements).
- Monitored: Equivalent von-Mises Stress.
- Percent Change: 0.54% (below 10% limit).
- Conclusion: Results are mesh-converged and reliable.

| Equivalent Stress (Pa) | Change (%) | Nodes | Elements |
|------------------------|------------|-------|----------|
| 42837                  |            | 802   | 387      |
| 43069                  | 0.54128    | 2972  | 1693     |



# Transient Structural Analysis

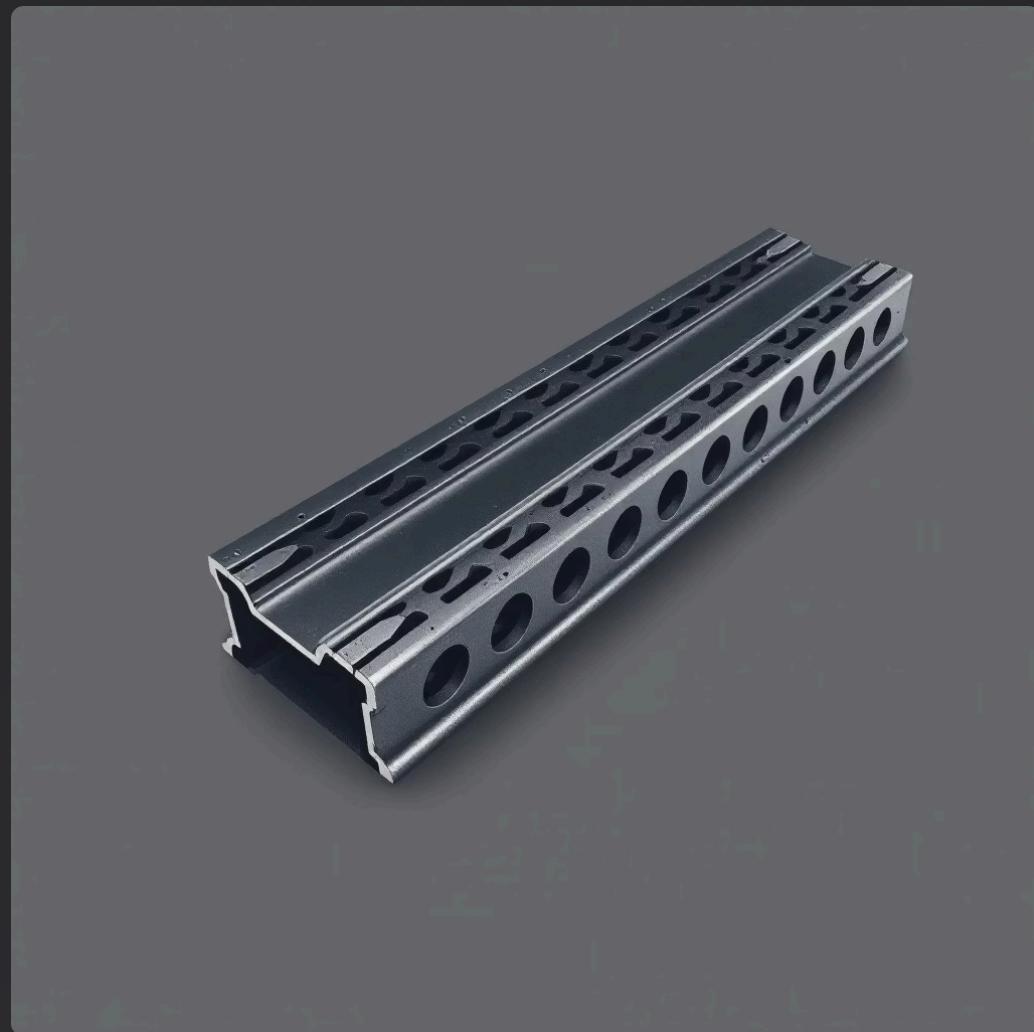
This analysis investigates the hexagonal prism's dynamic response over time.

The same 20m hexagonal prism geometry was reused for this dynamic simulation.

# Material & Loading (Transient)

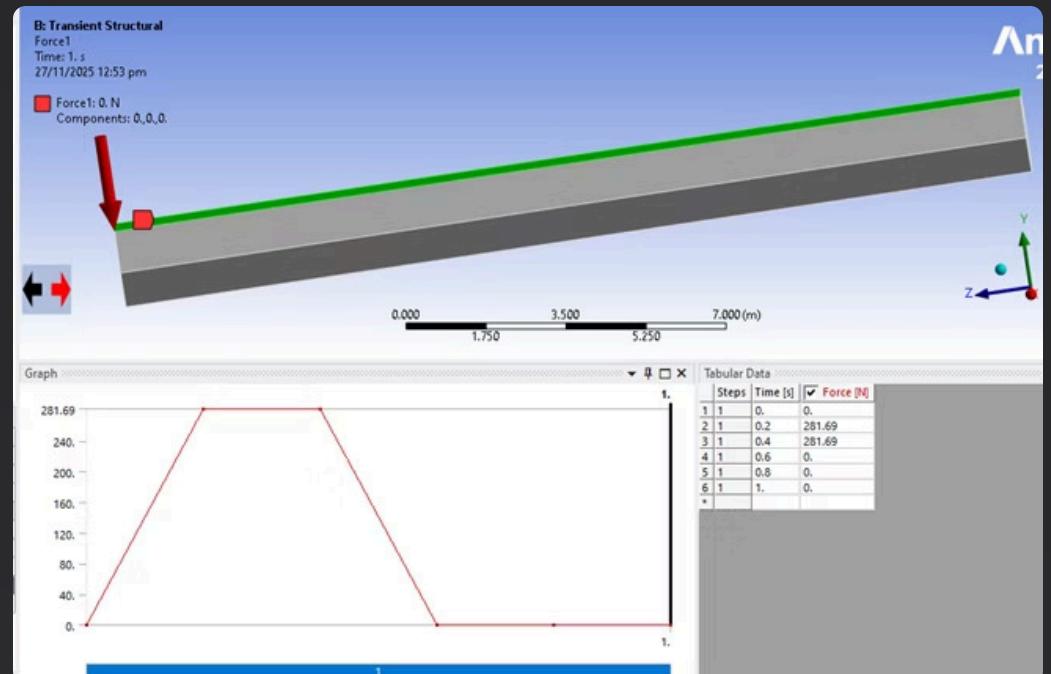
## Material Limitation

- Custom materials are not supported in Transient Structural.
- Default **Structural Steel** used ( $E = 200 \text{ GPa}$ ,  $\nu = 0.3$ ).

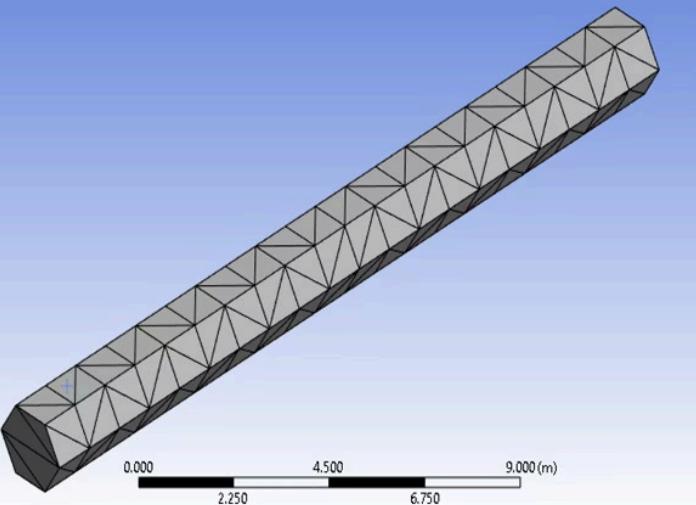


## Dynamic Loading

- Four compressive forces applied (281.69 N each).
- Applied on four lateral faces, directed inward.
- Time step: 0.2 seconds.
- Forces defined as tabular/time-dependent data.



# Meshing (Transient)



## Consistent Mesh Generation

Mesh generated using the same tetrahedron method as the static analysis.

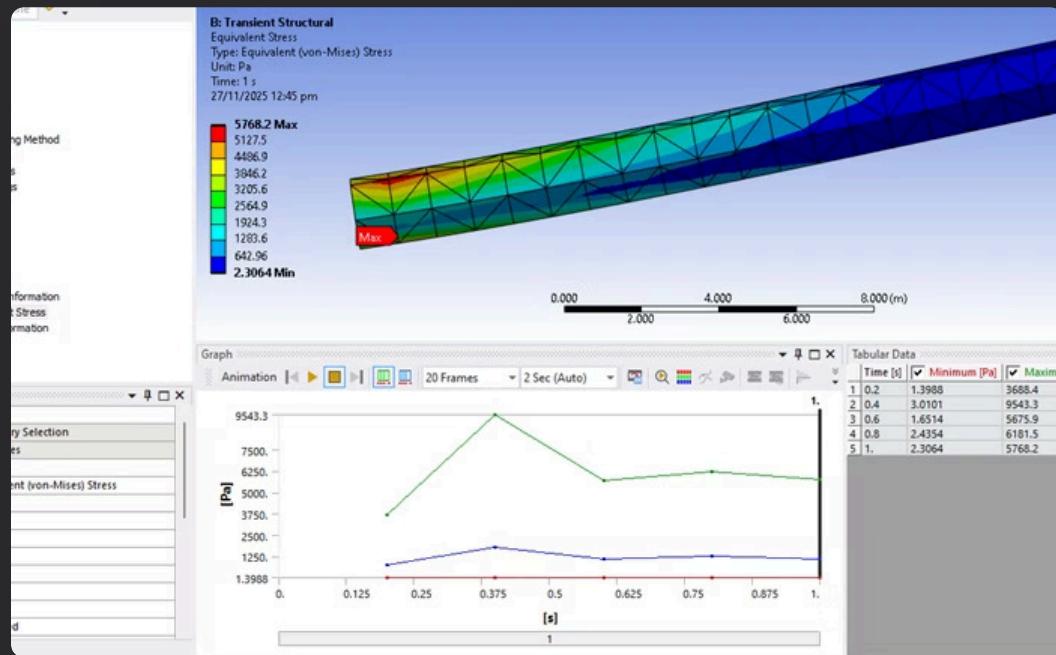
## Default Sizing

Program-controlled default sizing maintained for element size and order.

## Mesh Quality Maintained

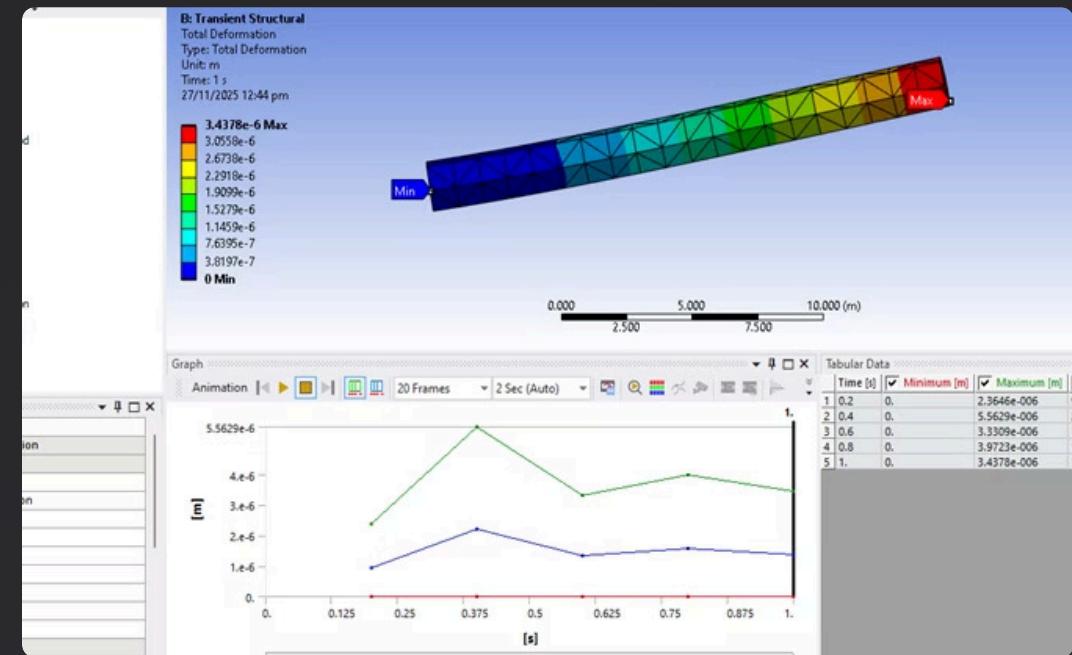
Mesh quality histogram recorded, ensuring consistent and reliable mesh quality.

# Transient Analysis Results



## Equivalent (von-Mises) Stress

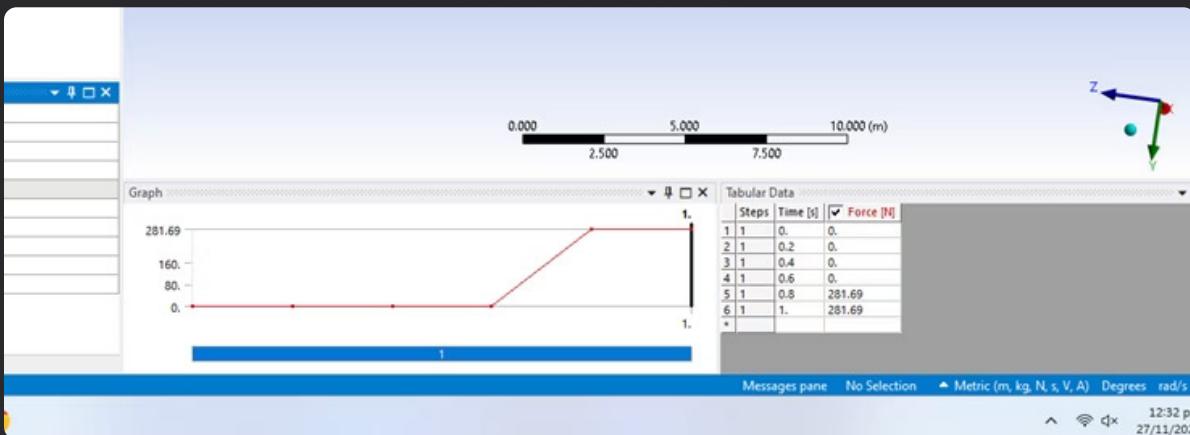
Analyzing stress distribution within the material.



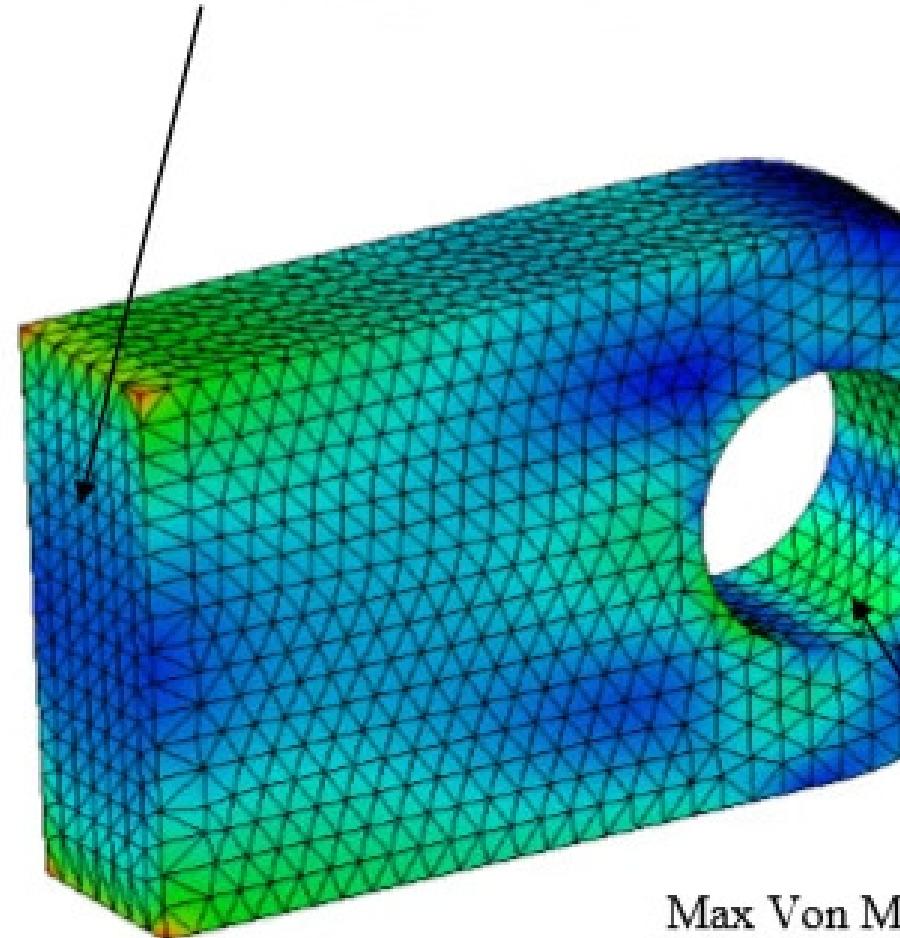
## Total Deformation

Visualizing the overall displacement under applied loads.

- For each time step, ANSYS reported:
- Minimum stress (low-stress zones)
- Maximum stress (red regions in contour)
- Average stress (overall structural response)
- Time-history graphs recorded



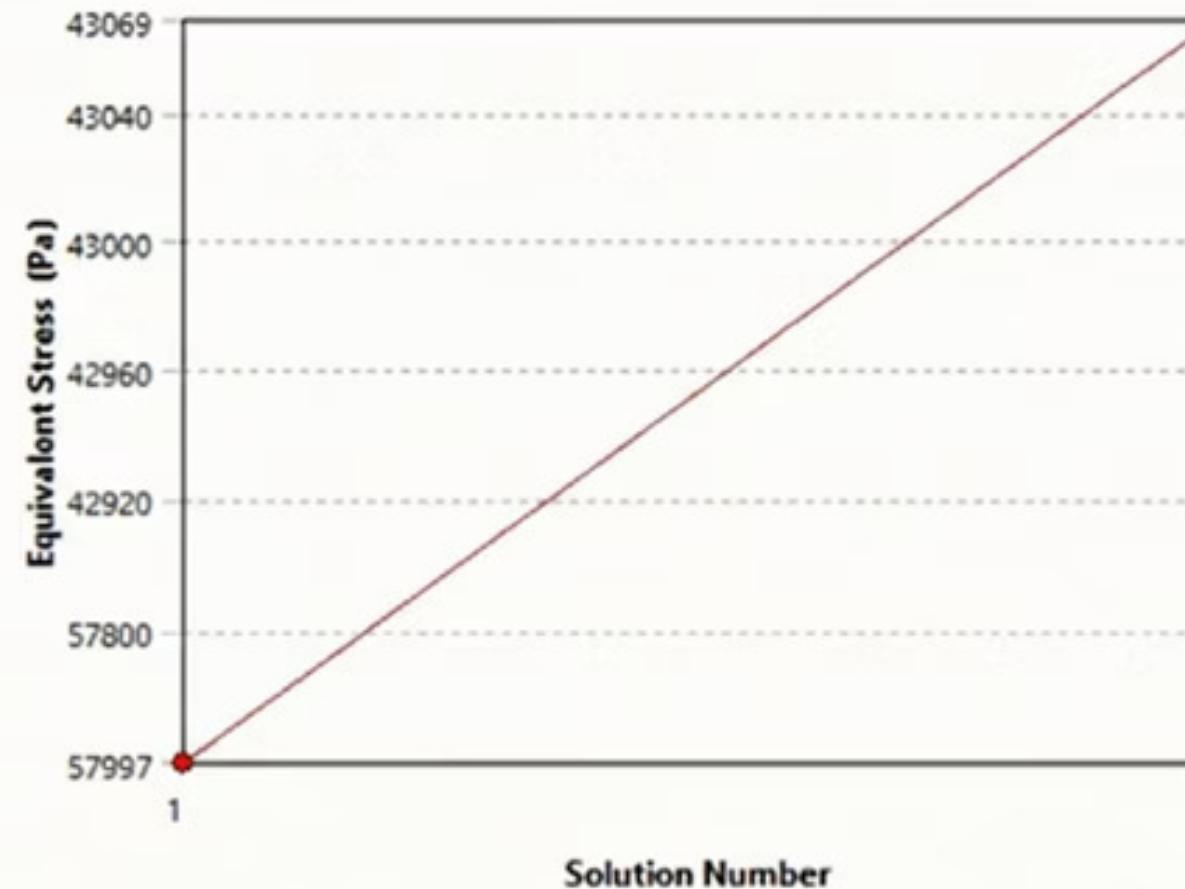
Max Von Mises Stress  
at attachment surface



Max Von M  
max displac  
bottom surfac

## Mesh Convergence (Transient)

## Convergence History



- Two meshes compared:
  - 387 elements (coarse)
  - 1693 elements (fine)
- Equivalent stress monitored over time
- Convergence graph generated
- Estimated percent change: **0.55%**
- Well below 10% → Results confirmed converged

|   | Equivalent Stress (Pa) | Change (%) | Nodes | Elements |
|---|------------------------|------------|-------|----------|
| 1 | 5768.2                 |            | 802   | 387      |
| 2 | 5799.9                 | 0.54128    | 2972  | 1693     |

# THANK YOU