



NUST

NATIONAL UNIVERSITY
OF SCIENCES & TECHNOLOGY



Report, Ansys Lab



Akhtar Abbas Basum

ME-1863

Instructor Sir Affan

INTRODUCTION

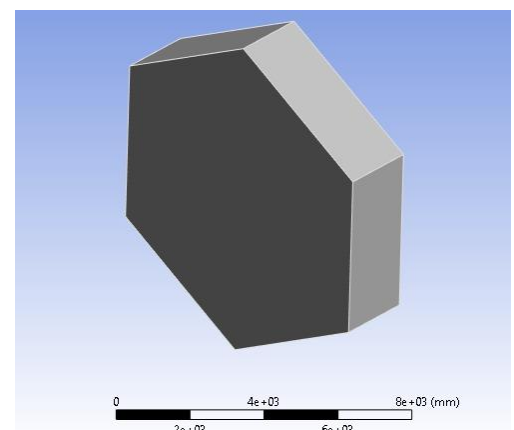
This report presents the complete workflow and results of both Static Structural and Explicit Dynamic (or Transient) simulations conducted on a hexagonal component. The objective was to evaluate the stress distribution behavior of the part under the specific loading and constraints defined in the project requirements. The geometry was created in SolidWorks and imported into ANSYS Workbench. The instructor-provided method for selecting Young's Modulus (E) and Applied Force (F) was followed throughout the analysis. All key steps, boundary conditions, results, and the mesh convergence study are detailed below.

Static Structural

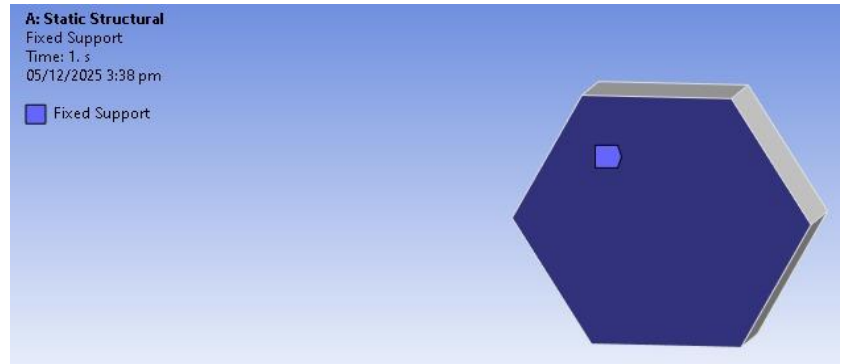
1. Initial Assumptions & Material Properties The Static Structural simulation was performed according to the guidelines provided by the instructor. The following specific input parameters were calculated and used: • Young's Modulus: Derived from the last two digits of the roll number (1863). = 63,000 MPa • Poisson's Ratio: 0.28 • Force Magnitude: Derived from the Date of Birth formula (20/09/2004).

Outline of Schematic A2: Engineering Data					
	A	B	C	D	E
1	Contents of Engineering Data				Source
2	Material				Description
3	Akthar				
*	Click here to add a new material				

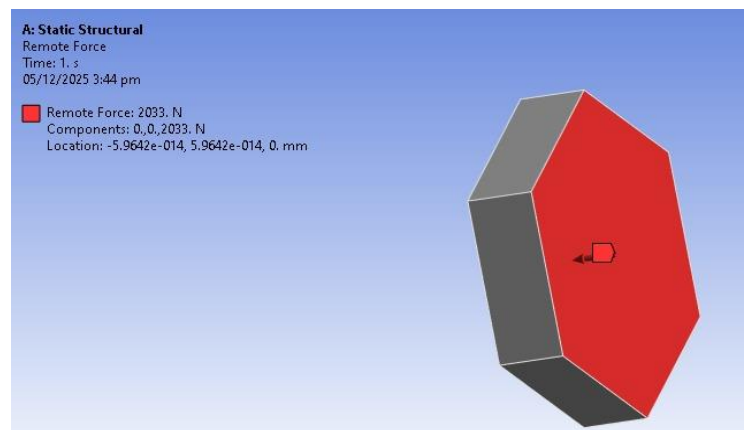
Properties of Outline Row 3: Akthar					
	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Young'...			
5	Young's Modulus	63000	MPa		
6	Poisson's Ratio	0.28			
7	Bulk Modulus	4.7727E+10	Pa		
8	Shear Modulus	2.4609E+10	Pa		



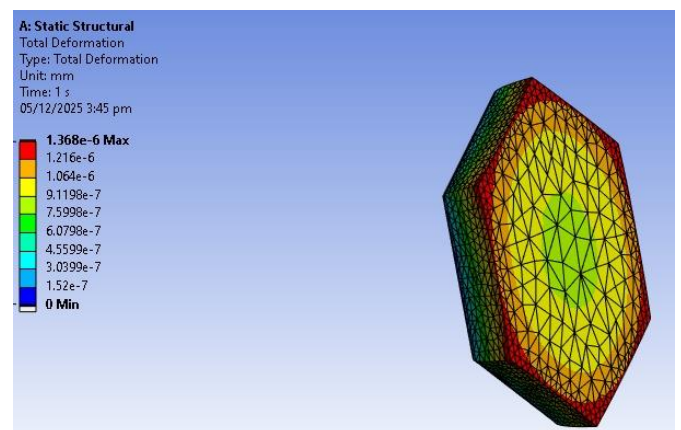
- 2. Fixed Support** One full side (face) of the hexagon was selected as the Fixed Support. This boundary condition fully restrained that face in all degrees of freedom, ensuring that the structure reacted only to the applied load without unwanted rigid-body motion



- 3. 1.3 Remote Force Application A** Remote Force of 2033 N was applied on the same fixed face. According to the project requirement, the load must be applied at 1/3 of the fixed face height. • A Remote Point was created at this specific coordinate. • The force was applied normally to the face to ensure correct load transfer without artificially constraining the geometry deformation.

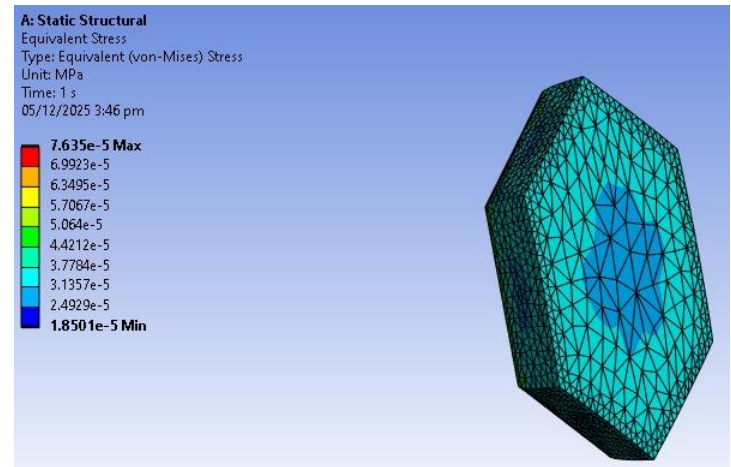


- 4. Meshing and Mesh Convergence Study** The hexagon was meshed using tetrahedral elements. To demonstrate mesh convergence (the point where results stop changing significantly with finer mesh), multiple mesh sizes were tested. • Coarse Mesh: Initial run. • Medium Mesh: Refined elements. • Fine

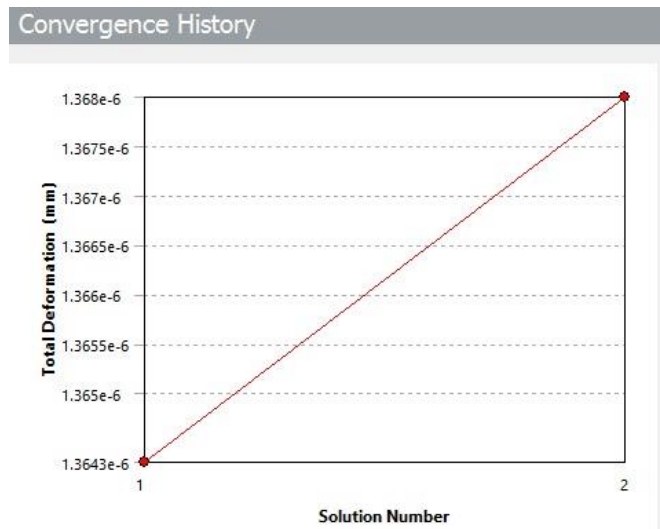


Mesh: Final high-quality mesh. For each mesh density, the Maximum von-Mises Stress was recorded and plotted.

- 5.** The Static Structural simulation produced stress fields consistent with the applied force (2023 N) and material stiffness (63,000 MPa). The component displayed expected elastic behavior. Due to the high Young's Modulus, the deformation was minimal, and the stress distribution remained within safe limits.

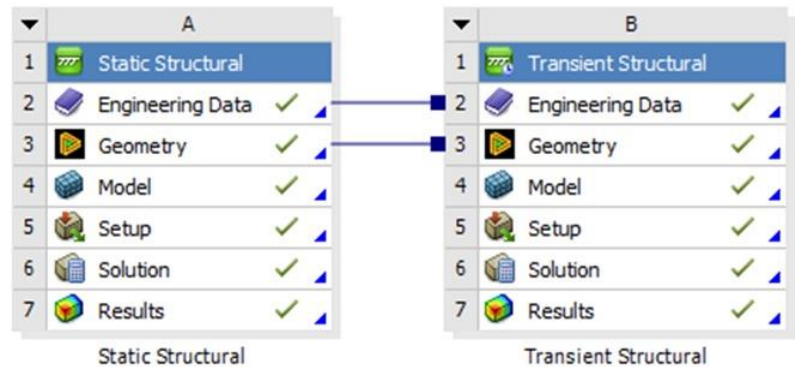


- 6.** The hexagon was meshed using tetrahedral elements. To demonstrate mesh convergence (the point where results stop changing significantly with finer mesh), multiple mesh sizes were tested.
- Coarse Mesh: Initial run.
 - Medium Mesh: Refined elements.
 - Fine Mesh: Final high-quality mesh.
- For each mesh density, the Maximum von-Mises Stress was recorded and plotted..

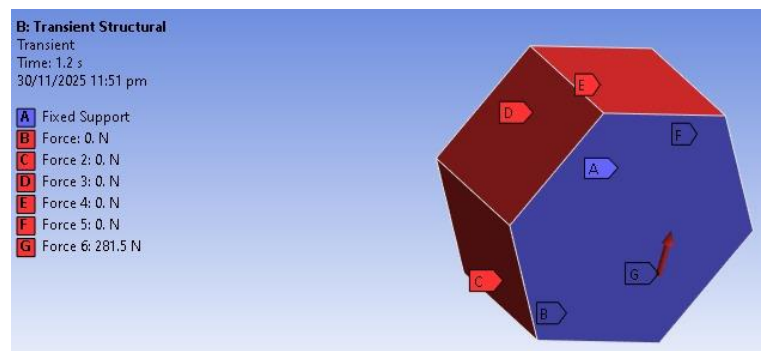


Transient Structural

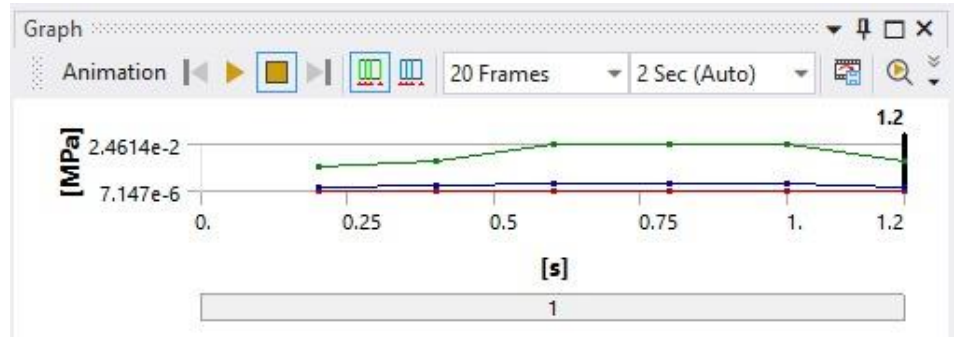
1. Initial Assumptions For the second part of the project, an Explicit Dynamics (or Transient) analysis was performed to observe the time-dependent response of the structure. • Young's Modulus: 63,000 MPa (Same as Static) • Poisson's Ratio: 0.28 • Force Magnitude: Based on the current Dollar to PKR rate. o Rate used: 282 N • Time Step: 0.2 seconds (Mandatory Requirement)



2. A Remote Force of 282 N was applied to the fixed face at the remote point (1/3 height). This lower force magnitude (compared to the static case) allows for the observation of stress wave propagation and dynamic response without immediately causing extreme deformation.



3. Time Step & Solver Settings • End Time: Set sufficiently long to capture the response. • Time Step: Manually controlled to 0.2 s as per instructions. •



Energy Error Handling: If "Energy error too large" warnings occurred, the Energy Reference Cycle was adjusted (e.g., from 0.1 to 500) to stabilize the explicit solver.

4. Stress and deformation results were extracted over the simulation time. The mesh convergence study was repeated for

Tabular Data				
	Time [s]	✓ Minimum [MPa]	✓ Maximum [MPa]	✓ Average [MPa]
1	0.2	2.5168e-005	1.2571e-002	1.7745e-003
2	0.4	9.7653e-006	1.5874e-002	2.6121e-003
3	0.6	9.4489e-006	2.4402e-002	3.5273e-003
4	0.8	9.5191e-006	2.4281e-002	3.5037e-003
5	1.	1.0583e-005	2.4614e-002	4.1501e-003
6	1.2	7.147e-006	1.5838e-002	2.224e-003

the dynamic case: 1. Simulation ran with decreasing mesh sizes (e.g., 20mm, 15mm). 2. 15 mm was selected as the optimal mesh size to balance accuracy with the long solve times typical of Explicit Dynamics.

5. The simulation successfully captured the transient behavior of the hexagonal structure under the dynamic load of 282 N. Due to the high stiffness ($E = 63,000 \text{ MPa}$), the structural response remained elastic. The stress values stabilized as the mesh was refined, fulfilling the convergence requirement.

