

Mounting Bracket Design & Static Structural Analysis

Author: Neelay Jain

Branch: Mechanical Engineering

Tools Used: SolidWorks 2024, ANSYS Workbench

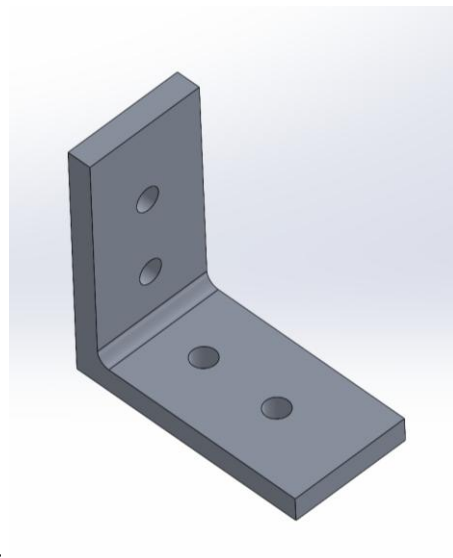
Date: July 2025

1. Objective

The objective of this project is to design a mechanical L-shaped mounting bracket and evaluate its structural integrity under a 500N applied load using Finite Element Analysis (FEA). The goal is to assess stress distribution, deformation, and safety factor using ANSYS Workbench.

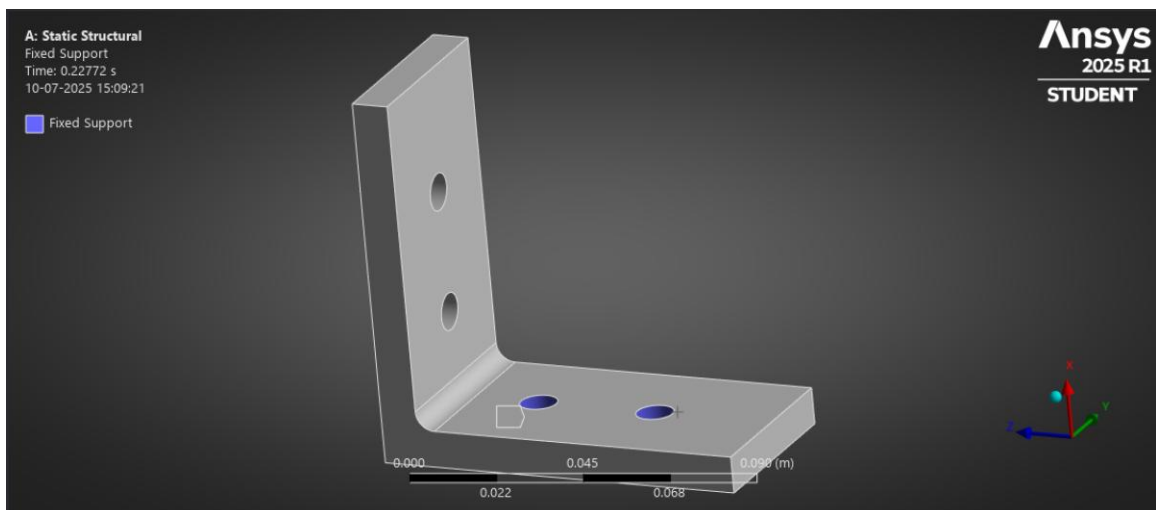
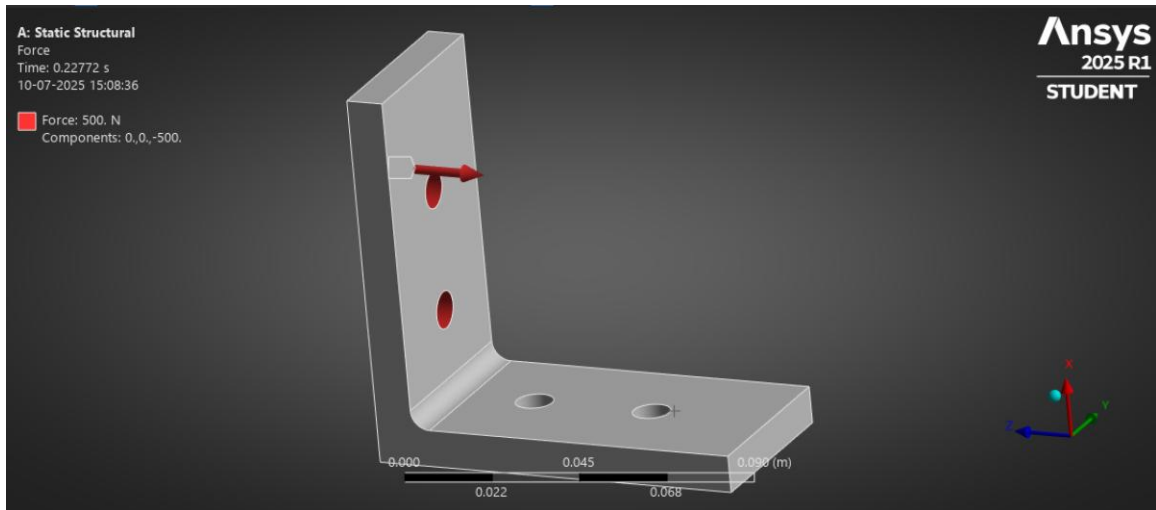
2. CAD Design Summary

- L-bracket: 100mm × 100mm × 10mm thick
- Holes: 10mm diameter, 2 per arm
- Fillet Radius: 5mm
- Material: Aluminum 6061
- CAD Model created using SolidWorks



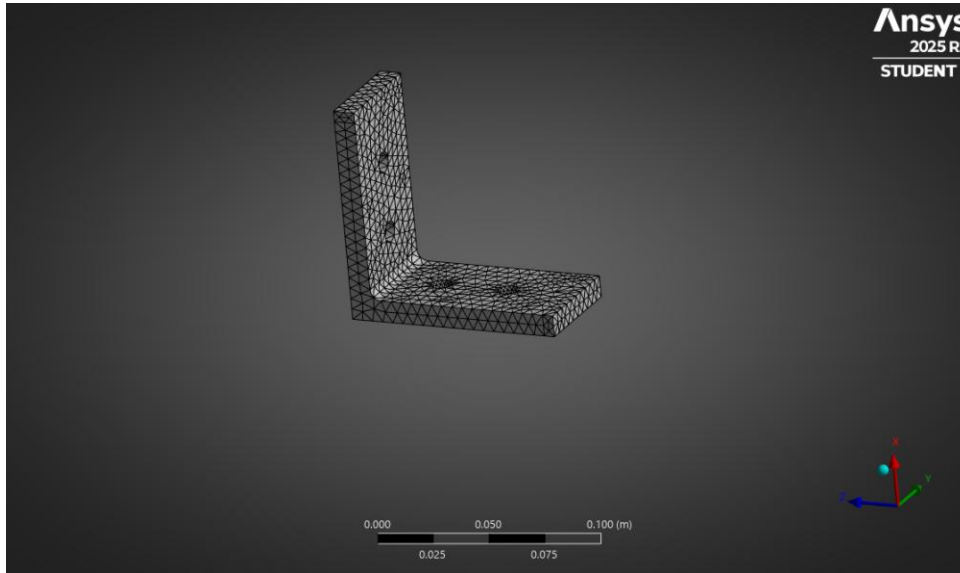
3. Boundary Conditions

- Fixed Support: Lower arm holes (2 faces)
- Force: 500 N downward on top arm holes (Z-direction)



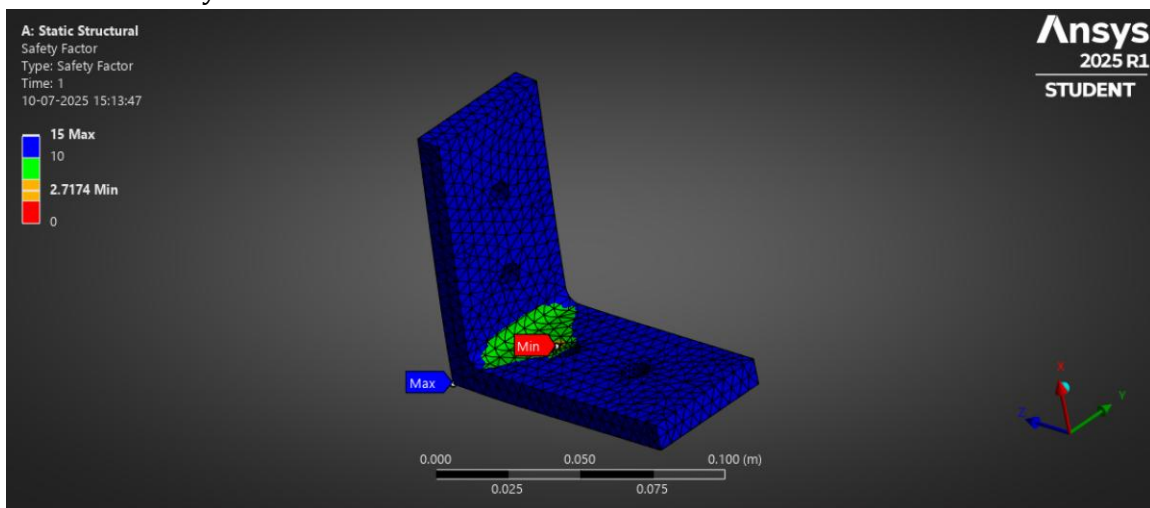
4. Mesh Setup

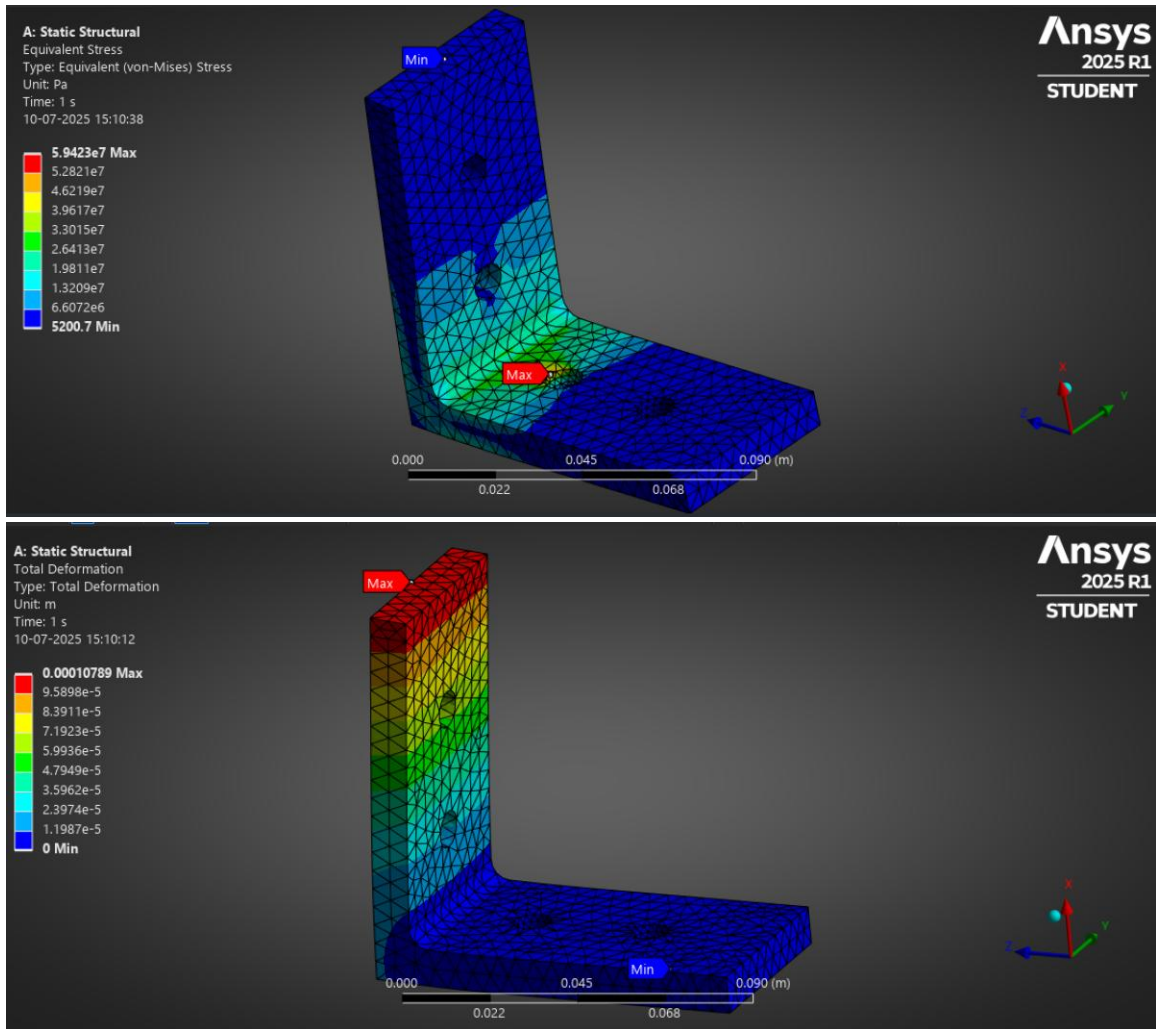
- Global element size: 5 mm
- Local refinement: 2 mm at bolt holes
- Mesh type: Tetrahedral elements



5. Results

- Max Von Mises Stress: $5.93 \times 10^4 \text{ MPa}$
- Max Total Deformation: 0.107mm
- Factor of Safety: 0-15





6. Conclusion

The mounting bracket successfully withstood the applied load of 500 N. The maximum stress observed was within the allowable limits of the selected material (Aluminum 6061), and the deformation remained minimal. The Factor of Safety indicates the design is structurally sound and suitable for practical use.