Mechanical Component Performance Study

Author: Piyush Kumar Sharma

Branch: B.Tech Mechanical Engineering

Tools: ANSYS, SolidWorks, Python, Excel, Power BI

Project Overview

This project focuses on the performance analysis of a mechanical component using stress and strain data obtained from engineering simulations. The analysis was done to identify high-stress regions, understand failure points, and improve component design. Python was used to process and visualize the data, while ANSYS and SolidWorks were used to model and simulate the component.

Tools and Methodology

- ANSYS: For finite element analysis and stress simulation.
- SolidWorks: For 3D modeling and structural design.
- Python (pandas, matplotlib, seaborn): For data analysis and visualization.
- Excel: For initial data review.
- Power BI: For interactive dashboards (optional).

Conclusion

The study demonstrated how simulation data can be used in combination with programming tools to improve mechanical design decisions. The workflow also highlights the practical application of data analysis in engineering, enabling better understanding of component behavior under load.