Finite Element Analysis (Caster Fanuc Bracket)

Sohrab Kashif (K00266024)

TUS

Aim of the Project

The aim of this project is to perform Finite Element Analysis (FEA) to ascertain the ability of caster brackets to withstand the load of a 2.2-tonne CNC machine. This entails analysing von Mises stress, total deformation, and factor of safety also demonstrating harmonic response, ensuring the brackets' structural integrity under varying condition

Background

Finite Element Analysis (FEA) is a powerful computational tool used in modern engineering to predict and understand the behaviour of structures and materials under various conditions. By breaking down systems into smaller elements and nodes, FEA enables engineers to solve complex equations, ensuring accuracy in design and optimization. With applications spanning aerospace, automotive, civil engineering, and more, FEA plays a vital role in enhancing reliability and efficiency in engineering designs. Different types of stress, such as axial, shear, and bending, affect materials differently under loading conditions. However, von Mises stress is a measure that combines all these stresses into a single scalar value. In FEA, von Mises stress serves as a critical parameter for evaluating the structural integrity and safety of engineering designs

Methodology

This analysis is carried out by using ANSYS WORK bench.

- **Data Preparation**: Convert SolidWorks drawings to IGS format for compatibility with ANSYS Workbench.
- **Static Structural Analysis Setup**: Define problem parameters and loading conditions.
- **Mesh Generation**: Create a mesh to discretize the geometry for accurate analysis.
- **Element Sizing**: Optimize computational efficiency with appropriate element sizing.
- **Boundary Conditions**: Specify boundary conditions and constraints for realistic simulations.
- **Total Deformation**: Visualize displacement under loading conditions.
- **Von Mises Stress**: Assess material strength and identify stress concentrations.
- **Factor of Safety**: Ensure structural integrity by calculating safety margins.
- **Harmonic Analysis**: Study dynamic response to vibrations and loads

Meshing

Figure 1 represents the quality of mesh

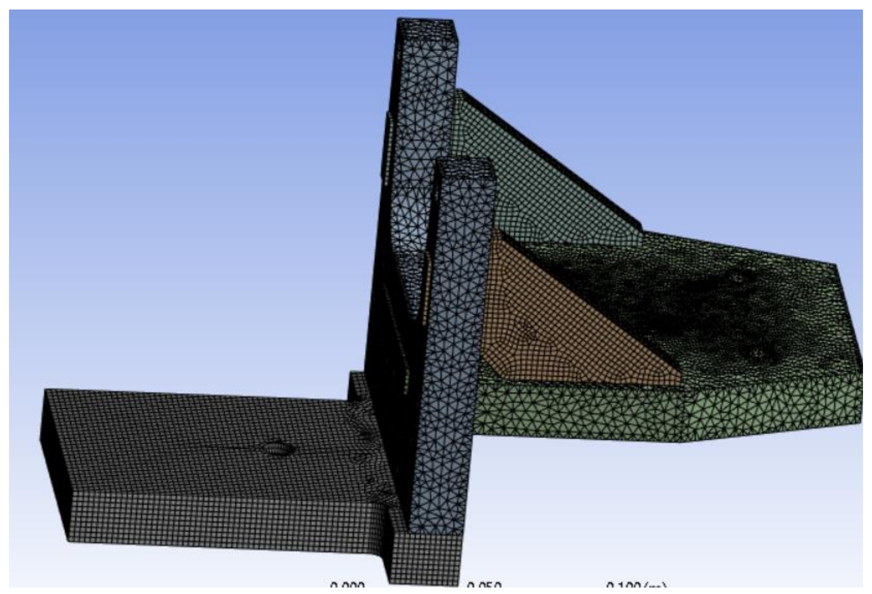

Figure 1: Mesh Quality

Results

Von misses stress

The Von Mises stress analysis reveals areas of high stress concentration across the bracket, with a peak stress value of 113.62 MPa, indicating the material's overall structural integrity under applied loads, Figure 2 indicates critical areas where stress is maximum and minimum.

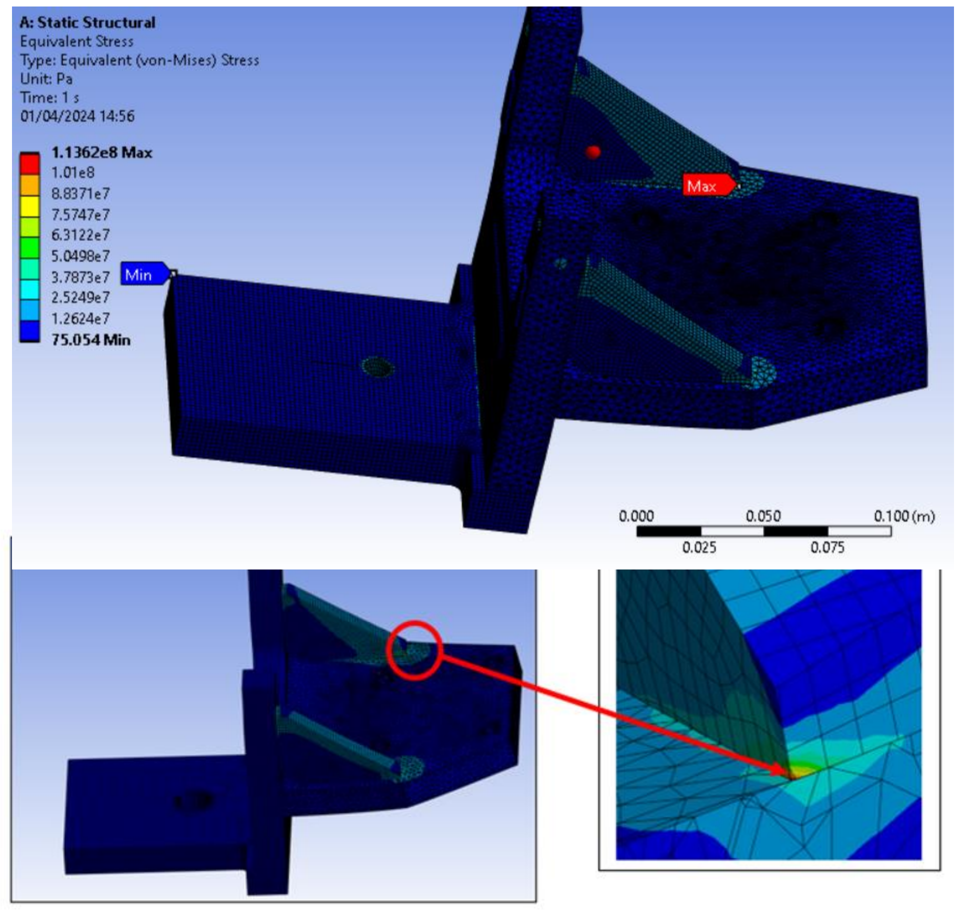

Figure 2: von misses stress and critical areas

Total Deformation

The total deformation analysis shows how much the bracket deflects under load, with observed deformation of 0.056768 mm, indicating minimal distortion and ensuring structural integrity.

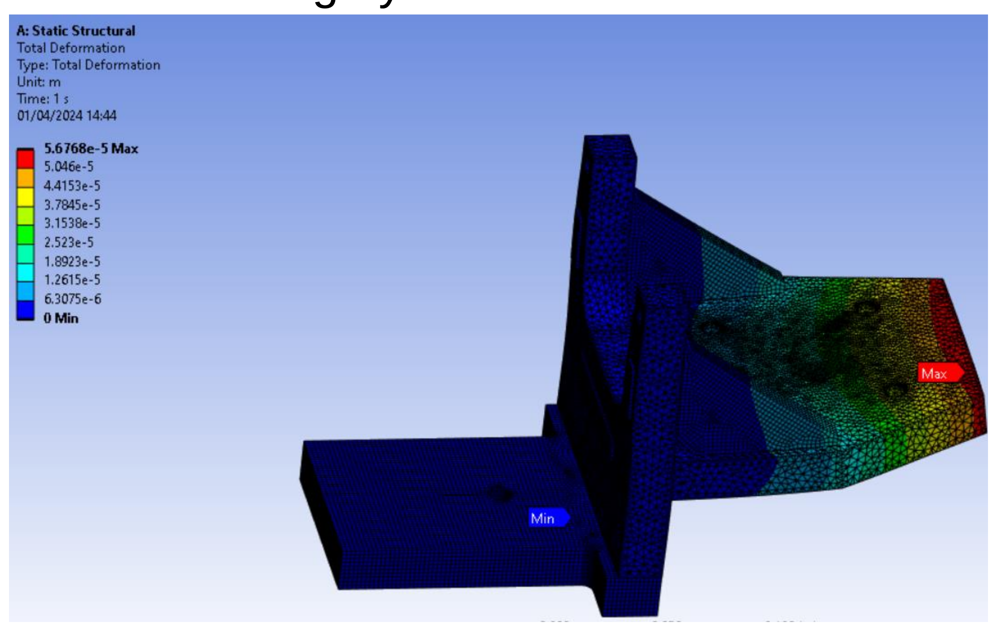

Figure 3: Total Deformation

Factor Of Safety

The factor of safety (Fos) of 3.1 ensures the bracket's ability to handle loads well beyond expectations, instilling confidence in its reliability and preventing potential failure

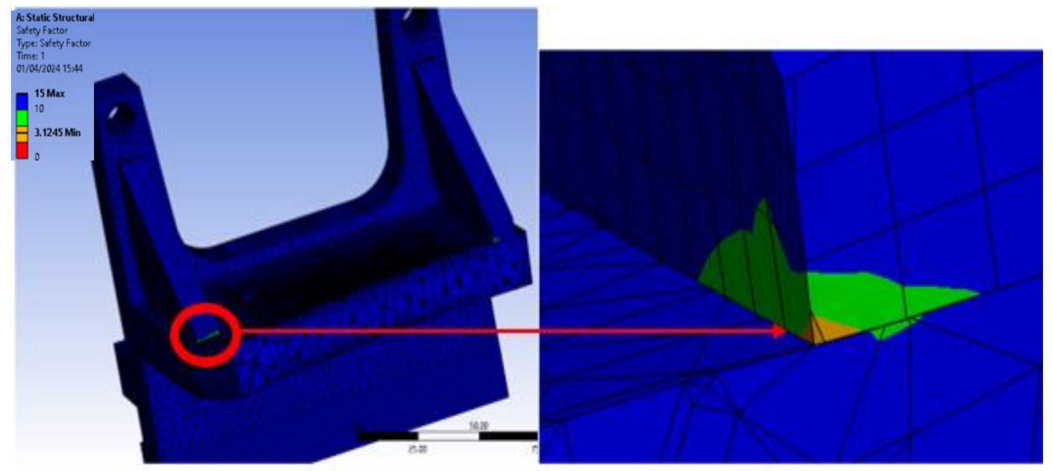

Figure 4 : Factor of Safety

Conclusion

The static structural analysis demonstrates that the bracket design meets the necessary safety requirements for supporting the CNC Faunac Robodrill machine. The Von Mises stress analysis, total deformation assessment, and factor of safety evaluation collectively affirm the structural integrity and load-bearing capacity of the bracket, ensuring safe and efficient operation during the movement of the 2.2-tonne CNC machine

References

ANSYS Workbench

SOLIDWORKS