Final Report

MECH 321 – Mechanics of Deformable Solids

Prof. Jianyu Li

Design and Analysis of a Structural Bracket

April 18, 2025

Group 14

Pierre Arbaji 261078645
Michael Salameh 261108086

Abstract

This report presents the design, analysis, manufacturing, and testing of a two-dimensional structural bracket for mechanical assemblies. The primary objective is to achieve a target stiffness, specifically, a resisting force of 300 N at a 0.7 mm displacement while minimizing the component's weight. To meet these objectives, the design leverages Finite Element Analysis (FEA) for optimized performance and emphasizes manufacturability constraints, ensuring compliance with laser cutting requirements.

Introduction

In this project, the bracket is designed from an acrylic sheet, having a constant thickness of 0.118 inches (approximately 3.0 mm), with fixed pin joint locations at points A, B, and C, and a vertical load is applied at point D. The component's geometry is confined within a 220 mm by 170 mm envelope with strict constraints on internal cut-outs and minimum material radii around the pins. Material properties, including a Young's modulus of 2.8 GPa and a tensile strength of 69 MPa, inform the design process and the subsequent mechanical testing [1]. The mechanical properties of acrylic and the geometry of the acrylic sheet are provided in the table and figure respectively below.

Table 1. Mechanical Properties of Acrylic [1]

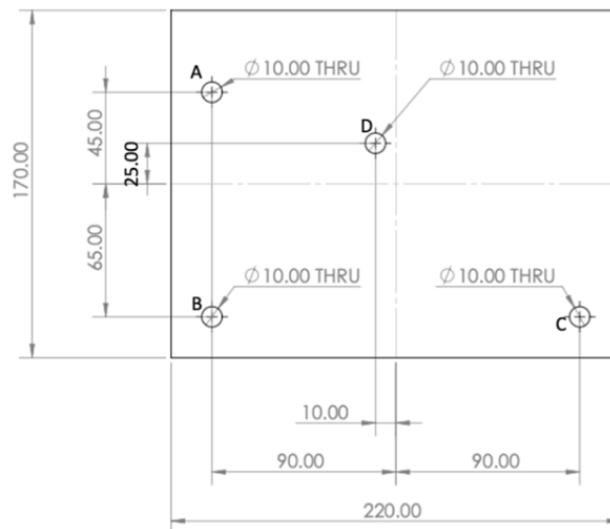| Properties	Values
Young's Modulus	2.8 GPa
Poisson Ratio	0.35
Tensile Strength	69 MPa
Density	1190 kg/m³

Figure 1. Geometry Constraints of the Acrylic Sheet and Pins (A-D) [1]

In designing the bracket, a key performance objective is to meet the stiffness requirement, which is defined as achieving a resisting force of 300 N when a vertical displacement of 0.7 mm is applied at pin D. The bracket is expected to deliver the specified stiffness while concurrently minimizing weight. This balance between stiffness and weight is fundamental to the design, ensuring efficient material use and structural integrity.

Finally, the testing and validation phase is designed to verify the bracket's performance against its FEA predictions. The manufactured component will undergo mass measurement, stiffness testing up to a

displacement of 0.7 mm, and strength testing up to failure or a maximum resisting force of 10.0 kN. These tests will confirm that the stiffness is within the required 300 N at the prescribed displacement and provide insights into the overall structural integrity and potential areas for design improvement

The report details the design process, from initial concept generation and analytical estimation to developing an optimized bracket geometry. Comparisons between FEA predictions and measured test results are discussed, highlighting successes and the areas for potential improvement.

Main Report

Application of Design Process

The design process for the structural bracket project began with a comprehensive concept generation phase, where multiple preliminary designs were sketched and evaluated considering the specified geometric, material, and performance criteria. Early brainstorming sessions allowed us to identify innovative configurations that could meet the target stiffness of 300 N at a 0.7 mm displacement while minimizing weight. First, the acrylic sheet was sketched on Abaqus analogously to Figure 1 as shown below.
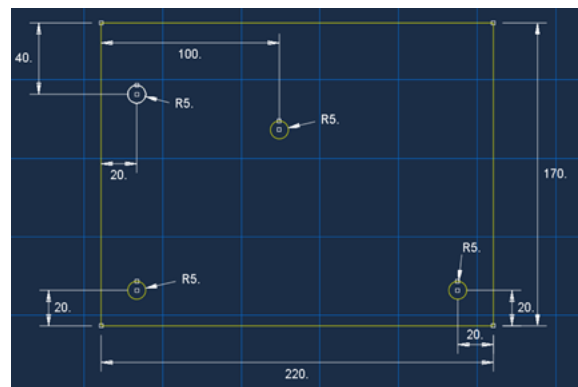

Figure 2. Acrylic Sheet Geometry

The initial concepts developed were inspired by the main objectives of the project: optimizing weight whilst minimizing stress concentrations. Several concepts, each of which mitigate the weight of the specimen to be tested to different degrees and include several fillets to minimize stress concentrations, were synthesized using Abaqus, some of which are depicted in Figure 3 below.

Figure 3. Initial Design Concepts

Following the initial concept development, the project moved into a phase of analytical estimation, that is Finite Element Analysis (FEA) Software was used to assess the feasibility of the different concepts generated. Firstly, Abaqus was used to impose certain boundary conditions on each of the pins. Specifically, the pins on each corner of the part were given pinned boundary conditions, that is a displacement of 0 in all directions, while the pin receiving the load was given to have a vertical displacement of 0.7 mm. This is shown for one of the concepts in the figure below.
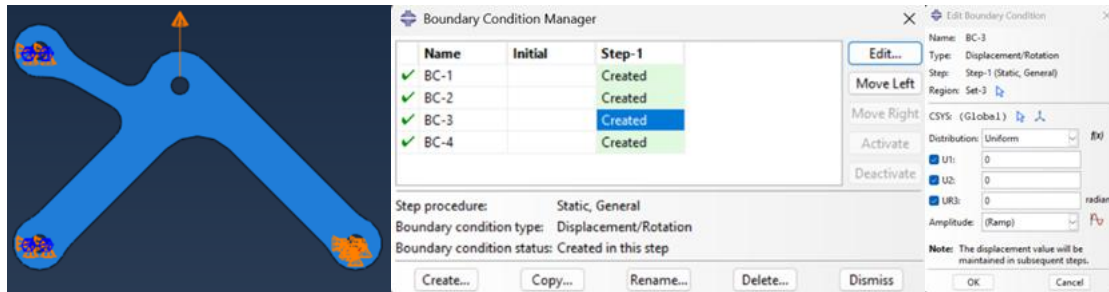

Figure 4. Boundary Conditions

After imposing the boundary conditions, Abaqus was used to generate meshes for the concepts shown in Figure 2. A preliminary FEA analysis was then simulated to observe the performance of the potential candidates. It must be noted that the refinement of the meshes was not of great importance yet, as the primary objective in this step was to assess the feasibility of using each of the outlined concepts. A particular design consideration was the potential failure of the material. Therefore, a key parameter to observe was the maximum Von-Mises stresses generated in the specimen. If it were to exceed the tensile strength of the material, then the design concept would be eliminated. The initial generation of meshes of one of the concepts is displayed in Figure 4.
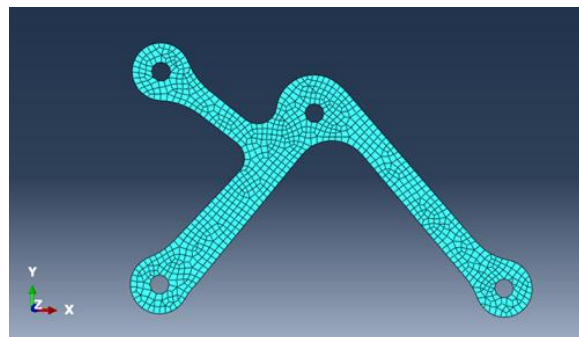

Figure 5. Initial Mesh Generation

Following the preliminary FEA analysis, it was noted that 2 of the specimens, that is the thinner specimens, were performing poorly with relation to the maximum Von-Mises stress generated. Specifically, the maximum Von-Mises stress generated was considerably larger. Therefore, both concepts were eliminated, and iterations were performed, as described in the paragraph below, to the chosen concept shown in Figure 5.

Figure 6. Chosen Design Concept

In the next section of the report, further simulations will be run on the chosen design concept, where the reaction force produced will be measured.

Quality of Finite Element Analysis

The selected design concept was then refined through iterative FEA simulations. During this stage, the team developed detailed computer models to simulate the bracket's structural behavior under the specified loading conditions. That is, the initial mesh generation was refined such that more detailed and precise results were given after generating a finite element model. Firstly, the number of the meshes were increased as close to 1 000, the maximum number of meshes allowed in the student version, as possible. Secondly, the mesh sizes were varied such that a higher density of meshes was imposed near the holes, that is the areas generating relatively high stress concentrations, and a lower density of meshes farther away from the holes, that is the area generating relatively lower stress concentrations.

This progressive refinement of the mesh is known as a mesh convergence study, an essential aspect that ensured the accuracy and reliability of the simulation results. To put it simply, as the mesh density increased, the simulation predictions of stiffness and stress distribution converged to stable values, confirming the validity of the chosen model. Quantitatively, the global stiffness prediction varied from 66.70 N/mm with 900 nodes to 66.82 N/mm with roughly 1 000 nodes, a change of only 0.18%, demonstrating that the results had effectively converged. It must be noted that an FEA simulation was run every time the mesh was varied. Once the results of the FEA simulation remained near constant, the mesh was said to converge. Figure 7 below shows the refined mesh. Additionally, Figure 8 visually shows the mesh convergence as the number of meshes are increased.
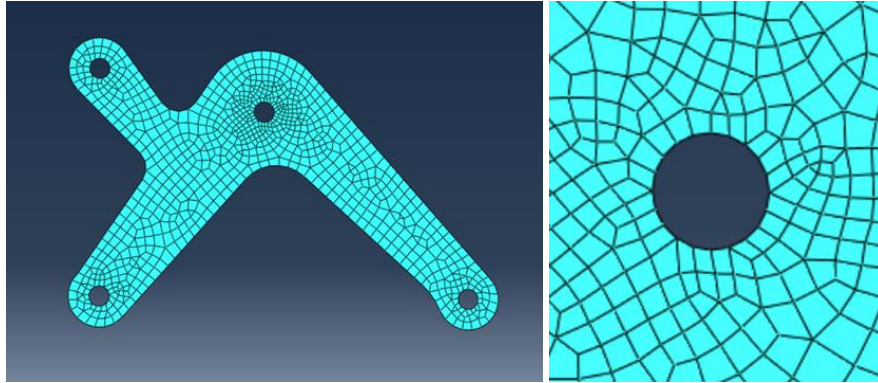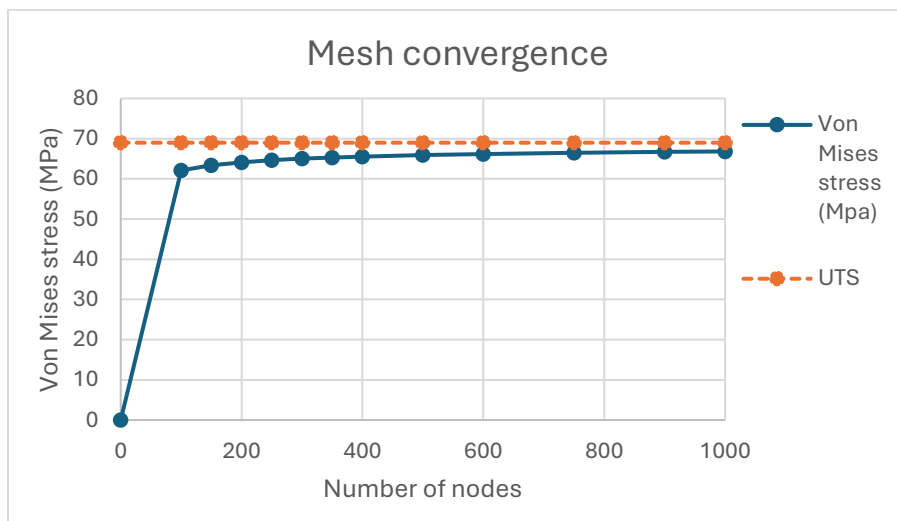
Figure 7. Refined Mesh

Figure 8. Mesh Convergence

With regards to the variable mesh size, Figure 9 below provides more details in relation to the approximate global size, and minimum size control for the converged mesh.
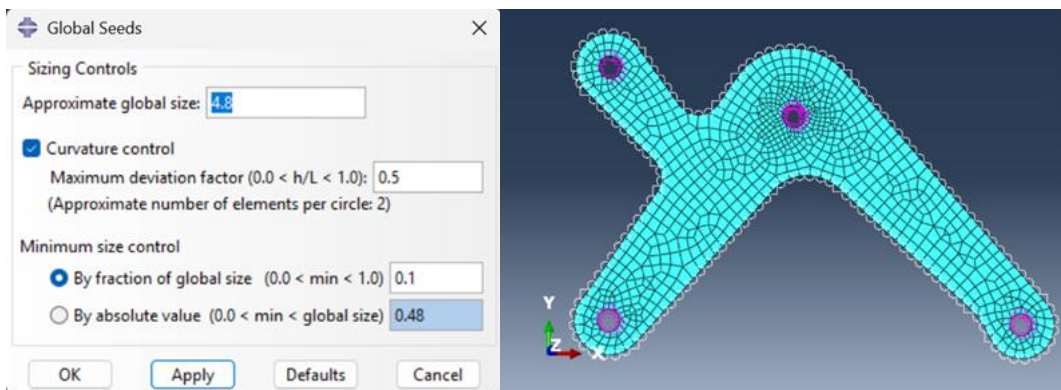

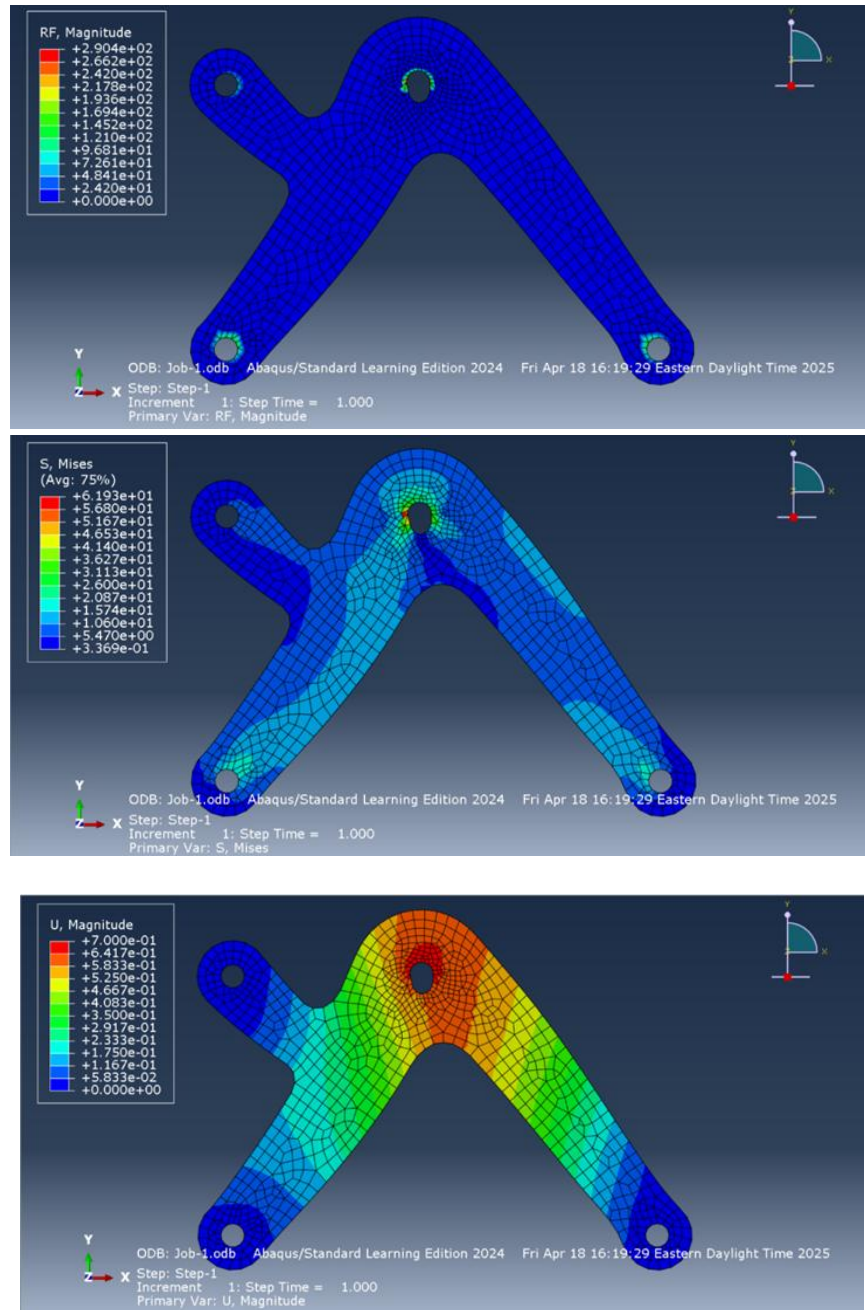**Figure 9.** Refined Mesh Specifications

It must be noted that this section will be revisited once the experimental results are discussed later in the report due to the experimental and theoretical discrepancies.

Manufacturing Consideration

Manufacturing constraints further guide the design process. Given that the bracket will be produced using a laser cutter, the design must remain strictly two-dimensional, with the laser cutting through the full thickness of the sheet. The design is limited to a maximum of five internal cut-outs, which excludes the four standard 10 mm diameter pin holes. In addition, any cut features must adhere to a minimum cutting radius of 10 mm. Furthermore, a uniform material boundary of at least 10 mm must be maintained around the four pin locations to ensure proper structural attachment and ease of assembly. The final design geometry was submitted as a DXF file that complied with the specific instructions provided, ensuring compatibility with the laser cutter's operational requirements.

Furthermore, it was noted that the brackets were going to be manufactured in bulk. Therefore, manufacturing time was also considered. To minimize the manufacturing time, our design was ensured to be as simple as possible. Our design was able to be manufactured via 1 smooth contour. The omission of any sharp edges from our design had obvious mechanical advantages, but it also decreased the corresponding complexity of the laser-cut machine's G-Code ensuring a minimal manufacturing time. Not to mention, a simple design also minimized the chances of any potential manufacturing errors.

Design Report and Drawing(s)

Figure 10. Drawing of the bracket

Testing, Performance and Prediction

Theoretical Results

The results of the FEA simulation run on the converged mesh are displayed below in Figure 11 below.

Figure 11. Finite Element Analysis Results

A summary of the results is provided in Table 2 below.

Table 2. Summary of Theoretical Results

Parameter	Value
Maximum Reaction Force	290.4 N
Displacement	0.7 mm

Maximum Equivalent Von-Mises Stress	61.93 MPa
Stiffness	414.8571429 N/mm

The specimen tested is, therefore, said to be under-stiff as it produced a reaction force smaller than 300 N. The stiffness of the material was calculated by dividing the reaction force by the displacement.

Presentation and Analysis of Testing Results

Given the test results, a force-displacement curve was plotted, as shown in Figure 12 below.
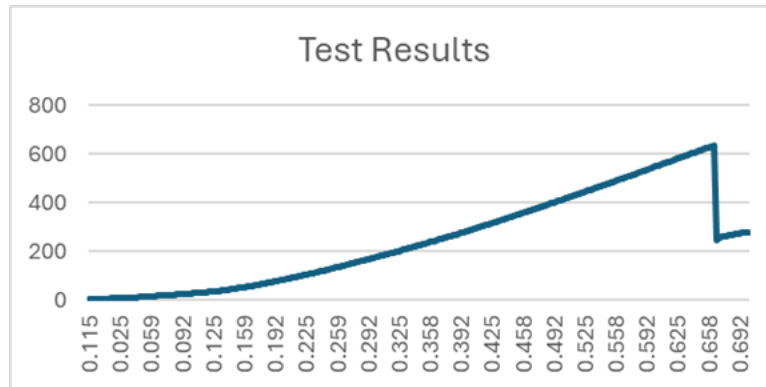
Figure 12. Reaction Force (N) Vs. Displacement (mm) Curve

Furthermore, a stress-strain curve has been plotted below assuming that the area under the load remains constant, thus neglecting the effects of necking. The area was calculated using the geometry of the pin hole.
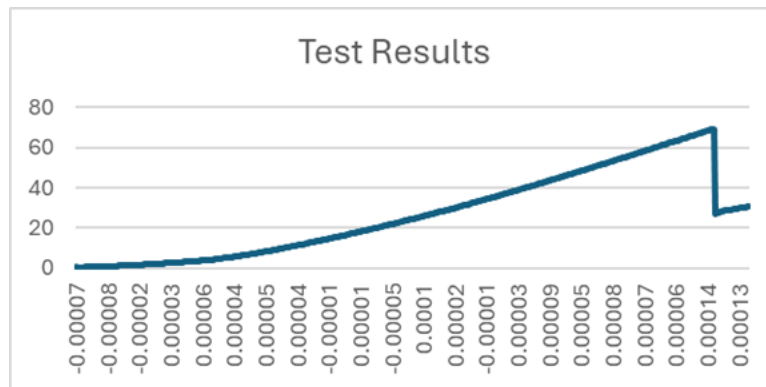

Figure 13. Stress (MPa) Vs. Strain Curve

While testing, the specimen failed at 0.665 mm while producing a reaction force of 633.19 N. An image of the failure is shown below.
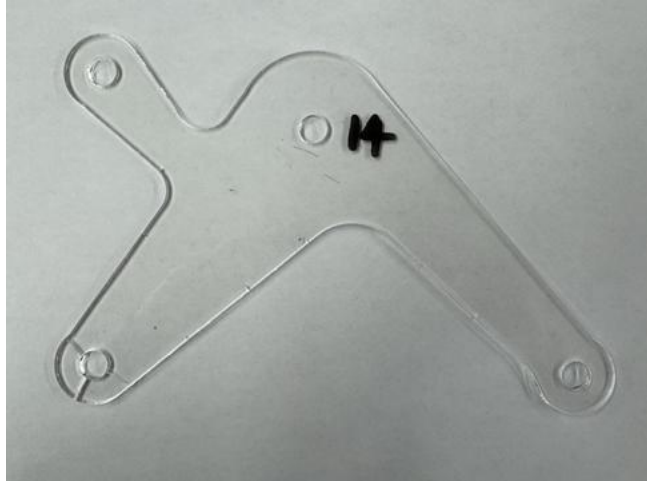
Figure 14. Specimen Failure

Upon completing some research, the discrepancies between the experimental and theoretical results can be attributed to the omission of the differences between the simulation results and the actual behavior of the physical system, that is the sim-to-real ratio. If this project were to be repeated, the sim-to-real ratio should be considered, as it can substantially impact the results obtained.

Performance Against Stiffness Criterion

Table 3. Summary of Experimental Results

Parameter	Value
Maximum Reaction Force	633.19 N
Displacement	0.665 mm
Maximum Equivalent Von-Mises Stress	69 MPa
Stiffness	952.1654135 N/mm

Since our specimen produced a reaction force much larger than 300 N, it is said to be over-stiff. For comparison, the experimental is computed as well, and is found to be more than double the theoretical stiffness.

Predictions of Strength

Prior to the test, it was predicted that one of the four pins would fail upon testing due to the stress concentrations. A shooting method approach was used to determine the load and location at failure. The convergence of the stress to the ultimate tensile strength value and load to the corresponding failure force value can be seen in the figures below.

After several iterations, it was predicted that the failure force would be around 465 N, occurring at Pin D due to the relatively high stress concentrations.
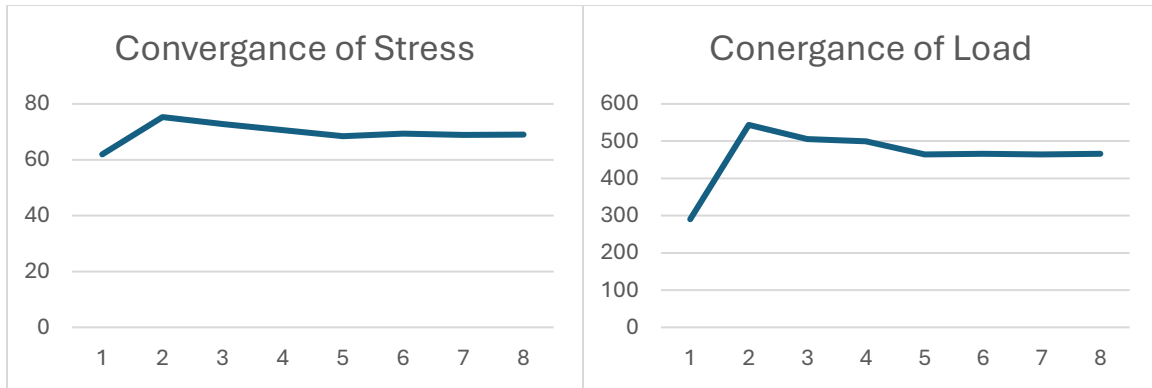
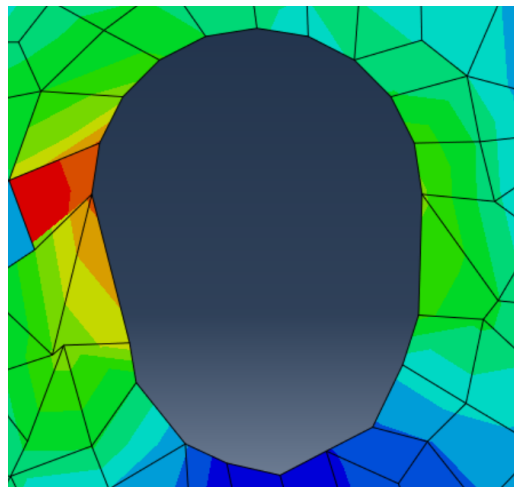**Figure 15.** Load & Stress Convergence to Ultimate Tensile Strength

Figure 16. Stress Concentration at Pin D

Once again, the differences between the theoretical prediction and the experimental prediction can be explained by the sim-to-real ratio.

As for the mode of failure, the material suffered a brittle, tensile failure as seen from its 45-degree crack.

Author Contributions and Peer Review

Author Contributions

Pierre Arbaji and Michael Salameh contributed equally to all aspects of this project. Both authors were involved in concept development, analytical estimations, Abaqus FEA modeling and mech convergence studies, design-for-manufacturing reviews, DXF file preparation, mechanical testing, data analysis, and report writing. All figures, tables, and discussions were jointly developed and refined through regular team meetings and collaborative editing sessions.

Peer Review

Peer Evaluation by Pierre Arbaji of Michael Salameh

Michael demonstrated exceptional technical skill and dedication throughout the project. He led the mesh convergence study in Abaqus precisely, ensuring that our FEA results were reliable and reproducible. His attention to detail in preparing the DXF file and coordinating with the TA for laser-cutting logistics was invaluable. He also provided clear, insightful feedback during data analysis, helping to interpret the load-displacement results and identify the unexpected fracture location. Michael communicated effectively, met every deadline, and was always ready to tackle any challenge, making him a highly dependable and collaborative teammate.

Peer Evaluation by Michael Salameh of Pierre Arbaji

Pierre excelled in guiding the overall design process and keeping the team focused on our stiffness and weight targets. His analytical calculations provided a strong foundation for our initial design iterations, and they skillfully managed the mechanical testing phase, ensuring accurate data collection and thorough post-test observations. Pierre's drafting of the report sections was clear, concise, and well-organized, seamlessly integrating FEA insights with experimental findings. His proactive approach to troubleshooting unexpected test results and his commitment to continuous improvement made him an outstanding partner on this project.

References

[1] Department of Mechanical Engineering, MECH 321 Project Winter 2025: Structural Bracket Design with FEA, Manufacture, and Testing, McGill University, 2025.

[2] W. D. Callister, Materials Science and Engineering: An Introduction, 10th ed. Hoboken, NJ: Wiley, 2018.

[3] Dassault Systèmes, Abaqus Analysis User's Guide, Version 2023, Dassault Systèmes, 2023.

[4] O. C. Zienkiewicz and R. L. Taylor, The Finite Element Method for Solid and Structural Mechanics, 7th ed. Oxford, UK: Butterworth-Heinemann, 2013.

[5] P. Stafford, "Laser Cutting of Plastics: Process Guidelines and Material Considerations," J. Manuf. Processes, vol. 15, pp. 45–52, 2013.

[6] M. P. Bendsøe and O. Sigmund, Topology Optimization: Theory, Methods, and Applications. Berlin, Germany: Springer, 2003.

[7] A. P. Boresi and R. J. Schmidt, Advanced Mechanics of Materials, 6th ed. Hoboken, NJ: John Wiley & Sons, 2003.