TOPOLOGICAL OPTIMIZATION AND STRESS ANALYSIS BY FINITE ELEMENT METHOD OF A MECHANICAL STEEL COMPONENT

Assoc. Prof. PhD. Alin STĂNCIOIU, ***Constantin Brâncusi University, Târgu -Jiu, ROMANIA,*** **stancioiualin09@gmail.com**
Asst. Ph.D. student Eng.Elena Loredana STĂNCIOIU, ***Constantin Brâncusi University", Târgu -Jiu, ROMANIA,*** **stancioiuloredana@gmail.com**
Mihai CHIVU, ***National University of Science and Technology POLITEHNICA Bucharest, ROMANIA,*** **mihai.chivu@upb.ro**

Abstract: In the context of modern engineering, reducing the mass of structural components is an absolute priority for cost efficiency and dynamic performance. This paper presents an integrated approach to stress analysis and topology optimization of a generic mechanical component. Using the Autodesk Inventor 2026.2 simulation environment, the study aims to reduce the mass of the part, while maintaining high structural integrity under the action of static pressures. The methodology involved the use of "Shape Generator" algorithms to eliminate redundant material, followed by a detailed "Static Analysis" for validation. The results demonstrate a remarkable mass reduction of 57.51%, without compromising safety, the component registering an overall safety factor of 15. The paper highlights the efficiency of computational methods in the product design cycle.

Key words: CAD, modeling, FEA, shape generator

1.INTRODUCTION

Computer-aided design (CAD) and computer-aided engineering (CAE) have revolutionized the way engineers approach product development. In the past, it was standard practice to oversize parts to ensure safety, which resulted in wasted material and increased inertia of mechanical assemblies. Today, through Finite Element Analysis (FEA) and topology optimization algorithms, material can be distributed exactly where it is needed to take on loads, resulting in organic, lightweight, and extremely strong designs.

The aim of this work is to apply these modern technologies to a steel plate subjected to constant pressure. The main objective is to minimize the mass, rigorously evaluating the structural response (Von Mises stresses, specific strains and displacements) of the new geometry.

2. THEORETICAL FOUNDATIONS

2.1. Finite element method (FEA)

Finite element analysis is a numerical method used to solve complex engineering and mathematical physics problems. It involves dividing a continuous domain (our part) into a set of discrete sub-domains, called "finite elements", connected by "nodes" (a process called discretization or meshing). By applying the laws of physics to each element and assembling them into a global system of equations, the stresses and strains within the material under the action of external forces can be approximated with a high degree of accuracy.

2.2. Topological optimization (Shape generator)

Topological optimization is a mathematical technique that optimizes the distribution of material within a given design space, for a specific set of loads, constraints, and support conditions. The algorithm evaluates the “path” of forces through the part and identifies areas of very low stress, gradually removing material from there. The result is a skeleton-like structure that maximizes stiffness relative to the mass of the part.

2.3. Von Mises yield criterion

For ductile materials, such as the steel used in this study, strength evaluation is most often done using the octahedral maximum tangential stress theory, known as the Von Mises criterion. This criterion combines the three principal stresses into a single "equivalent stress." If this equivalent stress exceeds the yield strength of the material, the part is considered to be in the realm of plastic (permanent) deformation.

3. METHODOLOGY AND MODELING

3.1. Material properties

The analyzed part was initially defined as a generic model, fig.1, with a volume of 169396 mm^3 and a surface area of 42340.7 mm^2. For the FEA simulation, the material was updated to Steel, having the following linear isotropic characteristics:

- **Mass density:** 7.85 g/cm^3.
- **Yield Strength:** 207 MPa.
- **Ultimate Tensile Strength:** 345 MPa.
- **Young's modulus (Elasticity):** 210 GPa.
- **Shear modulus:** 80.7692 GPa.
- **Poisson's ratio:** 0.3.

3.2. Mesh settings

To accurately capture the stress concentrators and complex geometries resulting from the optimization, a curved element mesh was chosen. The settings for the final static analysis were:

- Average element size: 0.1 (fraction of the model diameter).
- Minimum element size: 0.2 (fraction of average size).
- Grading Factor: 1.5, allowing a smooth transition from small to large elements.
- Maximum rotation angle (Turn Angle): 60 degrees.

3.3. Boundary conditions

The simulation reflected a scenario of rigid fixation and uniform loading:

1. **Load:** A pressure (Pressure:1) with a magnitude of 0.981 MPa applied to the selected faces.
2. **Constraints:** The part was spatially locked using two "Fixed Constraint" constraints (1 and 2), prohibiting any degree of freedom (translation and rotation) on the respective mounting surfaces.

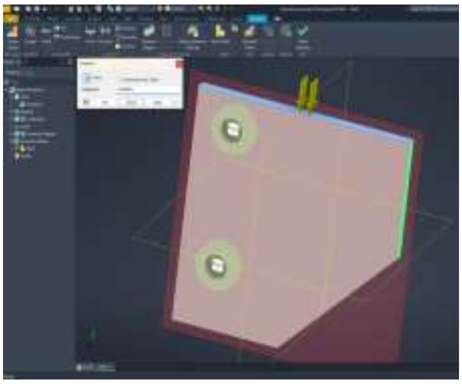

Fig.1 Geometric shape obtained in Inventor

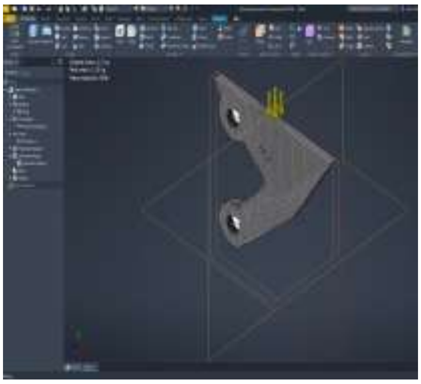

Fig. 2 Geometric shape with Shape Generator

4. RESULTS AND DISCUSSIONS

4.1. Efficiency of topological optimization

The "Shape Generator", fig.2, process targeted a single performance objective (Single Point). Initially, the massive steel part had a mass of 1.32976 kg. After running the algorithm that eliminated structurally unstressed material, the new mass reached only 0.564987 kg. This reduced mass equates to a 57.51% material savings. In a mass production scenario or for aerospace/automotive applications, a reduction of more than half the weight of a component generates huge cost and energy savings.

4.2. Stress analysis (Von Mises and principal)

Validation of the optimized design was performed through a static analysis. The results demonstrate that the part performs flawlessly under a pressure of 0.981 MPa.

The Von Mises equivalent stress varies between a minimum of 0.0625495 MPa and a maximum peak of only 9.68402 Mpa, fig,3.

Comparing this maximum stress (approximately 9.68 MPa) with the yield strength of the steel used (207 MPa), it is observed that the part is loaded to less than 5% of its maximum allowable capacity. This validates the success of the optimization: more than half of the material was removed, and the part still remained extremely stiff and safe.

Directional component analysis shows equally reliable values:

- **Principal Stress 1 (Tensile Stresses):** Maximum 6.61359 MP, fig.4.
- **Principal Stress 3 (Compressive Stresses):** Minimum -6.24372 Mpa, fig.5.
- **Normal Stresses:** Stress XX reaches a maximum of 5.31 MPa, Stress YY reaches 4.27 MPa, and Stress ZZ only 1.50 MPa., fig.6.

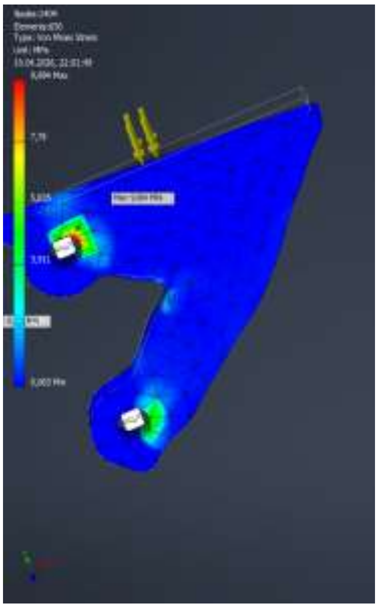

Fig.3. Von Mises stresses

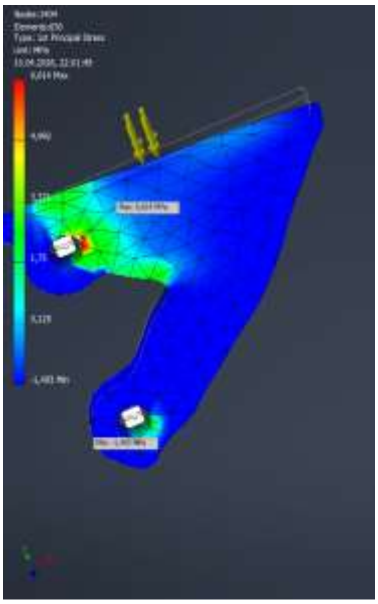

Figure 4: First principal stress

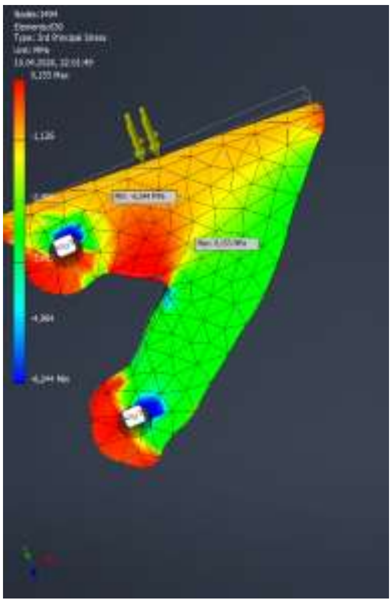

Fig.5: Third principal stress

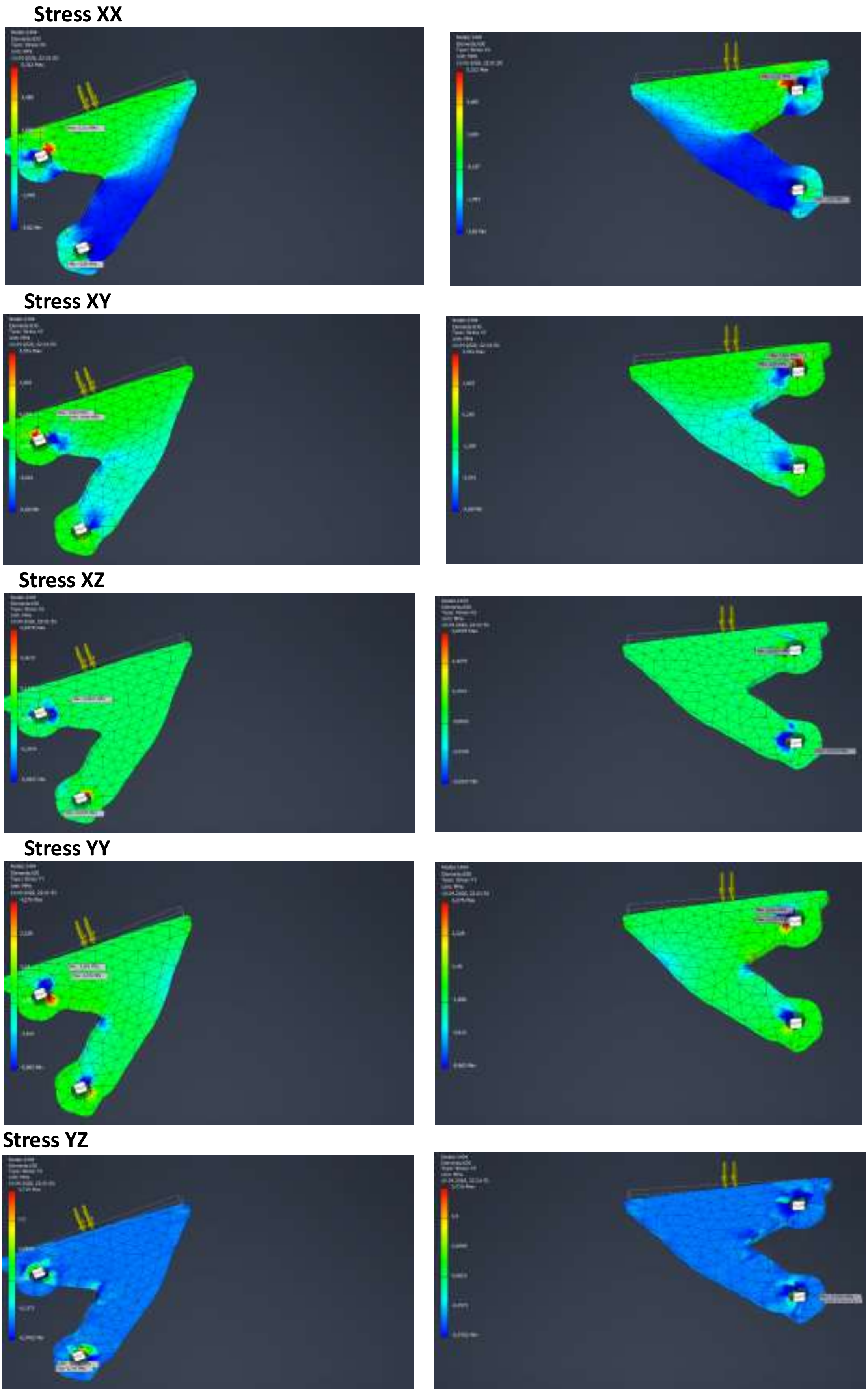

Fig.6 Normal stress XX, YY, ZZ

4.3. Displacement and deformation analysis

The overall rigidity of the structure is excellent. The maximum absolute displacement recorded over the entire part is 0.00492145 mm (less than 5 microns), a negligible value from an engineering point of view. Most of this displacement occurs on the Y axis (approx. -0.00466 mm), fig.7-9.

The equivalent specific strain (Equivalent Strain) reaches a microscopic value of 0.0000399989 ul. All these values confirm that the part functions perfectly in elastic mode, without any risk of residual deformation.

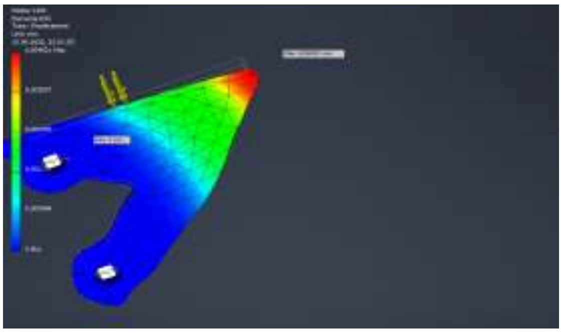

Fig. 7 Displacement

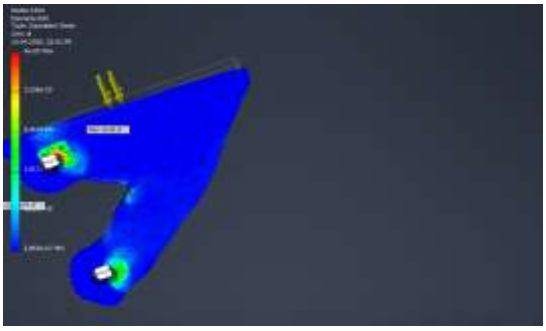

Fig. 8 Foreign equivalent

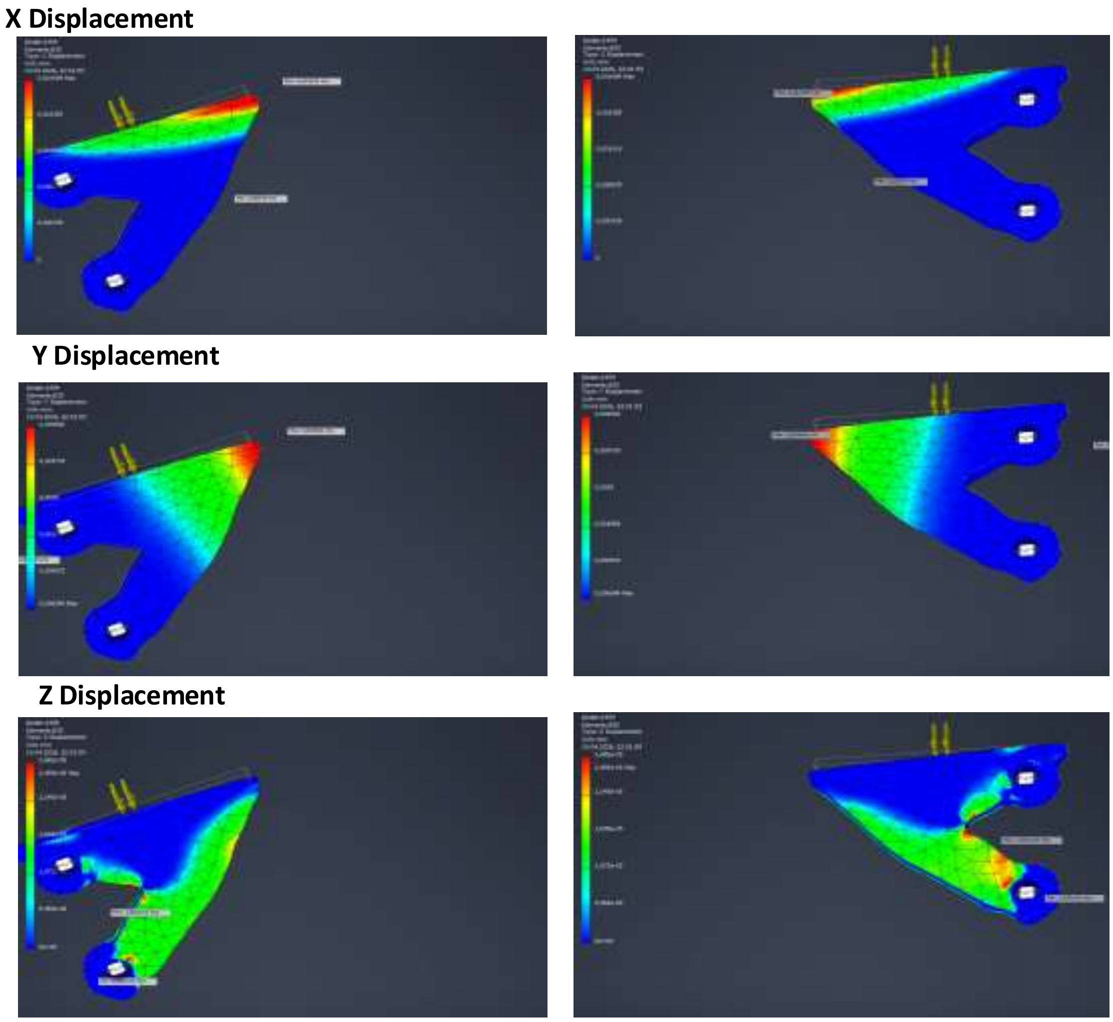

Fig.9 Directional movements X, Y, Z

4.4. Analysis of reactions in supports

The reactions at the fixing points are essential for the sizing of the fastening elements (bolts, welds, etc.). For fixed constraint 1, the reaction force was calculated to be 1377.77 N, accompanied by a reaction moment of 15.3252 N m. For fixed constraint 2, the global reaction force was 1172.39 N, with a reaction moment of 7.64 N m. Knowing these values, the design engineer can accurately select the type of assembly required to fix the part in its assembly.

The software reported a Safety Factor of 15 ul over the entire surface of the
of the part. This value represents the ratio of the yield strength of the material to the calculated maximum stress. A value of 15 indicates a severely over-dimensioned design even *after* topological optimization. In practice, for normal static parts, a factor of 2 to 4 is considered sufficient. The fact that the value is 15 suggests that there is additional room for a new round of optimization, decreasing the initial plate thickness, if the application requires it., fig.10.

4.5. Safety factor assessment

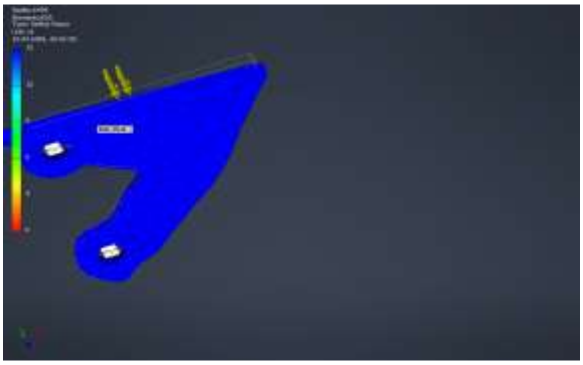

Fig. 10 Safety factor

5. THEORETICAL COMPARATIVE STUDY: STEEL VERSUS ALUMINUM ALLOY

To further highlight possible directions for component optimization, a brief theoretical comparative study was conducted in which the original part material (steel) is substituted with a general purpose aluminum alloy (e.g., Aluminum 6061-T6, commonly used in the aerospace and automotive industries). This analysis highlights the impact that material selection has on mass, stiffness, and structural strength.

5.1. Comparison of ohysical and mechanical properties

The main differences between the steel used in the original simulation and a typical aluminum alloy are as follows:

- **Mass density:** Steel has a density of 7.85 g/cm^3, while aluminum has a density of about 2.70 g/cm^3. This means that aluminum is almost three times lighter.
- **Young's modulus (stiffness):** Steel has a modulus of elasticity of 210 GPa, compared to about 70 GPa for aluminum. This indicates that aluminum is three times more flexible under the same load.

- **Yield Strength:** Molded steel has a yield strength of 207 MPa. An aluminum alloy such as 6061-T6 has a yield strength of approximately 276 MPa, providing superior resistance to plastic deformation despite its much lower mass.

5.2. Impact on optimized part mass

In the Shape Generator study, the mass of the steel part was reduced to 0.564987 kg. Since the optimized volume remains constant when changing the material in the CAD interface, a switch to aluminum would reduce the mass of the part proportional to the difference in density. Thus, the optimized aluminum part would end up weighing approximately **0.194 kg** . Comparing this value with the initial mass of the raw steel part (1.32976 kg), a total mass reduction of over **85% is obtained** .

5.3. Theoretical structural response (Stresses and displacements)

In a standard linear-elastic analysis, the distribution of internal stresses is dictated primarily by the geometry and applied loads, and is independent of the elastic modulus of the isotropic material. Therefore:

- **Maximum Von Mises stress:** It would remain constant, at the value of 9.68402 MPa, under the action of the same pressure of 0.981 MPa.**Safety factor:** Given that the operating stress (9.68 MPa) is well below the yield strength of an aluminum alloy (approx. 276 MPa), the safety factor would remain at an extremely high value (over 15), confirming the absolute safety of the design.
- **Displacements:** Since the modulus of elasticity of aluminum is three times lower than that of steel, the part would become more "flexible". The maximum displacement of 0.00492145 mm calculated for steel would triple, reaching approximately **0.0147 mm**. Even with this increase, the value remains microscopic and does not affect the functionality of the component in most mechanical applications.

5.4. Comparison conclusions

Theoretical analysis demonstrates that switching from steel to an aluminum alloy, on the same topologically optimized geometry, would transform the component into an ultra-lightweight part, while maintaining stresses within the perfect safety zone. However, although aluminum offers superior performance in terms of strength-to-weight ratio, the final material decision in a real project must also take into account economic factors (aluminum being generally more expensive) and fatigue or corrosion resistance in the specific operating environment.

6. CONCLUSIONS

This paper demonstrated the effectiveness of combining shape generation tools (Topology Optimization) with structural validation methods (FEA) in the Autodesk Inventor environment. The main conclusions drawn from the simulation are:

- **Substantial weight reduction:** Optimization allowed a 57.51% reduction in the amount of steel, bringing the weight of the part from 1.32 kg to approximately 0.56 kg. This translates to significant material and energy savings for large-scale production.
- **Maintained structural integrity and rigidity:** The maximum Von Mises stress of 9.68 MPa is extremely far from the yield strength of steel (207 MPa). Furthermore, the structure maintains exceptional rigidity, exhibiting a negligible maximum absolute displacement of only 0.0049 mm (less than 5 microns) , confirming that the component functions perfectly in the

elastic domain without risk of residual deformation.

- **Predictability for assembly sizing:** The explicit calculation of reaction forces in the supports—highlighting maximum forces of 1377.77 N and 1172.39 N — provides critical quantitative data required by design engineers for the exact and safe sizing of fastening elements in the larger mechanical assembly.
- **Material substitution viability:** The theoretical comparative study proved that substituting steel with a lightweight alternative, such as the 6061-T6 aluminum alloy , applied to the same optimized geometry, can achieve a total mass reduction of over 85% relative to the original raw part. Even under these extremely lightweight conditions, the safety factor remains above 15, validating a highly secure design.
- **Additional optimization potential and manufacturability:** A safety factor of 15 ul suggests that the initial component thickness is still severely over-dimensioned , opening the possibility for a new iteration of optimization to further reduce material costs. Finally, the organic geometry resulting from this process validates the "performance-oriented design" concept, being perfectly suited for advanced additive manufacturing (metal 3D printing) or 5-axis CNC machining.

REFERENCES

[1] Bendsøe, MP, & Sigmund, O. (2004). *Topology Optimization: Theory, Methods, and Applications* . Berlin: Springer Science & Business Media.

[2] Zienkiewicz, OC, Taylor, RL, & Zhu, JZ (2013). *The Finite Element Method: Its Basis and Fundamentals* (7th ed.). Oxford: Butterworth-Heinemann.

[3] Hibbeler, RC (2018). *Mechanics of Materials* (10th ed.). New York: Pearson.

[4] Autodesk Inc. (2026). *Autodesk Inventor Professional, Version 2026.2 (Build 302298010, 298A)* [3D CAD design and simulation software].

[5] Reddy, JN (2019). *An Introduction to the Finite Element Method* (4th ed.). New York: McGraw-Hill Education.

[6] Budynas, RG, & Nisbett, JK (2014). *Shigley's Mechanical Engineering Design* (10th ed.). New York: McGraw-Hill Education.

[7] Liu, GR, & Quek, SS (2013). *The Finite Element Method: A Practical Course* (2nd ed.). Oxford: Butterworth-Heinemann.

[8] Kurowski, P. M. (2022). *Engineering Analysis with Autodesk Inventor Standard*. SDC Publications.

[9] Sigmund, O. (2001). A 99 line topology optimization code written in Matlab. *Structural and Multidisciplinary Optimization*, 21(2), 120-127.

[10] Rozvany, G. I. N. (2009). A critical review of established methods of structural topology optimization. *Structural and Multidisciplinary Optimization*, 37(3), 217-237.

[11] Tremblay, T. (2018). *Up and Running with Autodesk Inventor Simulation 2018*. Butterworth-Heinemann.

[12] Bathe, K. J. (2014). *Finite Element Procedures* (2nd ed.). Watertown, MA: Klaus-Jürgen Bathe.