Journal of Technology Research (JTR)
Volume 3, Special Issue, (2025), pp 678-683 ISSN 3005-639X
The 7th Conference of Engineering Sciences and Technology
(CEST-2025)

Design and Topology Optimization of L-bracket Using SolidWorks Static Simulation

Emad Elddein I. Elshwain[1], Ali A. Ben Rabaa[2]

[1]Industrial and Manufacturing Department, Faculty of Engineering, Misurata University, Misurata, Libya.

[2]Industrial Department, The college of industrial technology, Misurata, Libya.

*Corresponding author email: elshwion@eng.misuratau.edu.ly

Received: 21-09-2025 | Accepted: 25-11-2025 | Available online: 11-12-2025 | DOI:10.26629/jtr.2025.63

ABSTRACT

The continuous pursuit of efficiency in mechanical design demands components that combine lightweight characteristics with structural strength.This study presents an integrated Computer-Aided Design (CAD) and Finite Element Analysis (FEA) workflow for the topology optimization of a mechanical component, where an L-shaped bracket was selected as a case study.The process began with the initial design and static structural analysis using SolidWorks to establish a baseline performance under typical operating loads.The model was then subjected to a topology optimization study in SolidWorks Simulation, aiming to reduce mass while maintaining structural integrity within predefined constraints. After interpreting the optimization results, the optimized geometry was reconstructed in SolidWorks, and a final validation FEA was performed on the new design.The results demonstrated a 54.74% reduction in mass compared to the original model, while the maximum von Mises stress increased by only 15%, remaining well below the yield strength of the alloy steel. This confirms the effectiveness of topology optimization in achieving substantial weight reduction without compromising key performance criteria, emphasizing its essential role in modern mechanical design and analysis.

Keywords: Topology Optimization, Finite Element Analysis (FEA), CAD, SolidWorks, Lightweight Design.

تصميم وتحسين الطوبولوجيا لحامل على شكل حرف (L) باستخدام المحاكاة الاستاتيكية في برنامج (SolidWorks)

عماد الدين عيسى الشوين[1]، علي عبدالله بن رابعة[2]

[1]الهندسة الصناعية والتصنيع، كلية الهندسة ، جامعة مصراتة ، مصراتة، ليبيا.

[2]الهندسة الصناعية ، كلية التقنية الصناعية ، مصراتة، ليبيا.

ملخـــــــص البحـــــــث

إن السعي المستمر نحو تحقيق الكفاءة في التصميم الميكانيكي يتطلب مكونات تجمع بين خفة الوزن والمتانة الإنشائية. تقدم هذه الدراسة أسلوب عمل متكامل يجمع بين التصميم بمساعدة الحاسوب (CAD) والتحليل بالعناصر المحدودة (FEA) من أجل التحسين الطوبولوجي لمكوّن ميكانيكي، حيث تم اختيار حامل على شكل حرف (L) كدراسة حالة. بدأت العملية بالتصميم الأولي والتحليل الإنشائي الاستاتيكي باستخدام برنامج SolidWorks لتحديد الأداء الأساسي تحت الأحمال التشغيلية الاعتيادية. ثم خضع النموذج لدراسة تحسين طوبولوجي في برنامج SolidWorks Simulation بهدف تقليل الكتلة مع الحفاظ على السلامة الإنشائية ضمن القيود المحددة مسبقًا. بعد تحليل نتائج التحسين، أُعيد بناء الشكل الهندسي الأمثل داخل البرنامج نفسه، وتم إجراء تحليل تحقق نهائي باستخدام

FEA على التصميم الجديد. أظهرت النتائج انخفاض الكتلة بنسبة 54.74% مقارنة بالنموذج الأصلي، في حين ارتفع إجهاد فون ميسز الأقصى بنسبة 15% فقط، وبقي أقل بكثير من حد خضوع مادة الفولاذ. تؤكد هذه النتائج فعالية التحسين الطوبولوجي في تحقيق تخفيض كبير في الوزن دون المساس بمعايير الأداء الأساسية، مما يبرز دوره الحيوي في التصميم والتحليل الميكانيكي الحديث.

الكلمات الدالة: التحسين الطوبولوجي، التحليل بالعناصر المحدودة (FEA)، التصميم بمساعدة الحاسوب (CAD)، SolidWorks، التصميم خفيف الوزن.

1. Introduction

In industries ranging from aerospace to automotive, the drive to reduce weight is primarily motivated by the need for improved fuel efficiency, enhanced performance, and lower material costs.

Traditional design methods often rely on iterative, experience-based modifications, which can be time-consuming and may not yield a truly optimal design.

Topology optimization, as a modern computational design approach, algorithmically determines the optimal material distribution within a defined design space based on applied loads, boundary conditions, and performance requirements.
Unlike shape or size optimization, it can generate non-intuitive geometries, providing a powerful tool for conceptual and lightweight structural design.

Finite Element Analysis (FEA) serves as the computational backbone of this process, accurately predicting how a component responds to real-world conditions such as loads, stresses, and deformations.

The synergy between CAD, FEA, and optimization algorithms enables a fully integrated digital design-to-validation workflow, significantly improving the design efficiency and accuracy of engineering components.

This paper presents a practical application of this integrated approach, focusing on the design and topology optimization of an L-shaped bracket using SolidWorks 2025.

The objectives of this study include:

1. Modeling the initial geometry and conducting a static structural analysis to establish baseline performance.

2. Performing a topology optimization study aimed at minimizing mass while respecting displacement and stress constraints.
3. Reconstructing the optimized geometry and validating its performance through final FEA.

The results highlight how integrated CAD-FEA workflows can lead to lightweight yet structurally efficient components, reinforcing the growing role of topology optimization in modern product development

2. Previous Studies

Gao et al. (2024), This study discussed topology optimization of a bracket structure used in acquisition, pointing, and tracking systems, considering displacement and stress constraints at key points. The results showed that the optimized model reduced mass by approximately 30–35%, while the maximum displacement and stress remained within the allowable limits (below 80% of the yield strength). This confirmed the effectiveness of incorporating multiple operational constraints in the optimization process [1].

Lee, Yoo, and Lee (2021) proposed a Smoothed-Strain approach for topology optimization instead of traditional density-based methods. The results indicated that this method improved computational stability and produced smoother material distributions, reducing numerically invalid elements by more than 40% compared to conventional methods. The optimized structures also achieved about 10–15% higher stiffness at the same mass level [2].

Kambampati, Martins, and Hicken (2020), This study focused on a Level Set approach under combined mechanical and thermal loads. The results demonstrated that the optimized designs achieved better stress and heat distribution, reducing thermal distortions by nearly 20% compared to non-optimized structures while maintaining a mechanical safety factor above 2.5 [3].

Sun, Zhang, and Liu (2024, proposed a Fail-safe Optimization method using a damage scenario penalty. The results showed that optimized structures could retain more than 85% of their stiffness even after removing nearly 10% of the material in selected regions to simulate damage. This demonstrated higher fault tolerance compared to conventional designs [4].

Zhu et al. (2021),provided a comprehensive review of topology optimization applications in additive manufacturing. The review concluded that many applied studies achieved mass reduction rates ranging between 30% and 70%, depending on load conditions and constraints. However, it was emphasized that 3D printing restrictions (such as 45° overhang angles and interlayer bonding limits) reduce the feasibility of some computationally ideal designs [5].

Tang et al. (2024) This review examined topology optimization methods in linear and nonlinear elasticity environments. It was found that nonlinear analysis in large deformation scenarios improved stress prediction accuracy by 15–20% compared to linear methods, but nearly doubled the computational time (~100% increase in solution time). This highlights the trade-off between accuracy and efficiency in industrial applications [6].

4. Methodology

3.1 Initial CAD Model and Material Definition

The initial model of the L-bracket was created in SolidWorks 2025. The design space was defined as a volume corresponding to the typical geometry of an L-shaped structural bracket (Figure 1). The material assigned was Alloy Steel, with its mechanical and physical properties listed in Table 1. The initial total mass of the model was 0.506 kg, which served as the baseline for optimization.

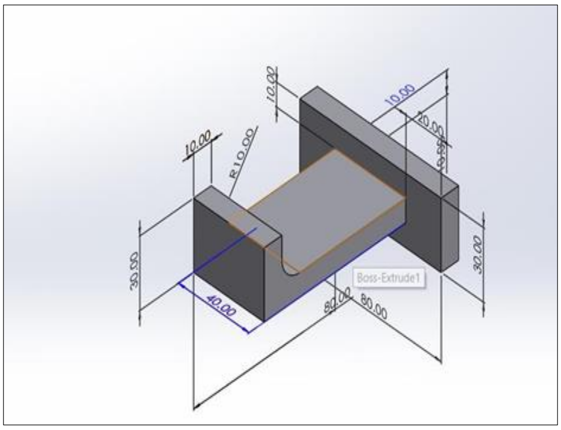

Fig 1. Initial CAD model of the L-bracket.

Table 1. Mechanical and Physical Properties of the L-Bracket.

Properties	Value	Unit
Elastic Modulus	210000	N/mm^2
Poisson's Ratio	0.28	
Shear Modulus	79000	N/mm^2
Mass Density	7800	kg/m^3
Tensile Strength	410	N/mm2
Yield Strength	275	N/mm2
Thermal Expansion Coefficient	1.1x 10-05	1/K
Thermal Conductivity	14	W/(m·K)
Specific Heat	440	J/(kg·K)

3.2 Static Structural Analysis

The SolidWorks Simulation module was used to perform a static structural analysis to evaluate the baseline performance of the initial design.

Boundary Conditions:

- Fixtures**:** The right face of the bracket was constrained using a fixed support, simulating the bolted connection to the assembly.
- Loads: A uniform pressure load of 100 N/cm² was applied to the upper face.
- Meshing: Global Mesh Settings was conducted to verify solution accuracy.

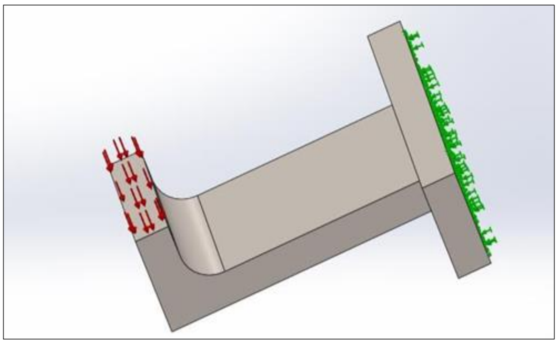

Fig 2. fixture and load constrains for L-bracket**.**

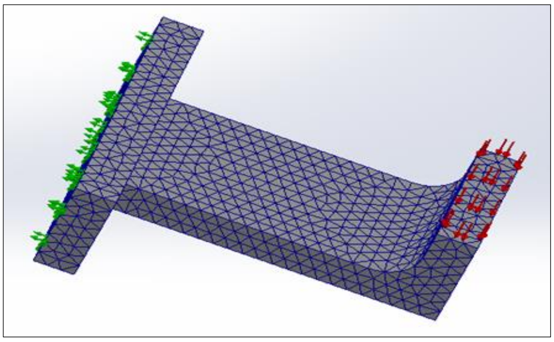

Fig 3. Discrete mesh diagram.

- Mesh Density: Fine
- Mesh Type: Curvature-based
- Max Element Size: 4 mm
- Min Element Size: 0.5 mm
- Growth Rate: 1.5

3.3 Topology Optimization Setup

A topology optimization study was created in SolidWorks Simulation using the static structural analysis as the reference case.

Objective: Minimize mass while maintaining the structural integrity.

Constraints:

- Maximum displacement ≤ 0.1 mm
- Maximum allowable stress ≤ 100 MPa

The solver iteratively removed low-stress regions, identifying the most efficient load-bearing paths and achieving a lightweight optimized structure suitable for re-design and validation.

Mesh Configuration for Topology Optimization

Global Mesh Settings:

- Mesh Density: Very Fine
- Mesh Type: Blended Curvature-based
- Max Element Size: 3 mm
- Min Element Size: 0.3 mm

5. Results and Discussion

4.1 Baseline FEA Results

The baseline finite element analysis (FEA) established the initial performance parameters of the L-bracket before optimization. Figure 4 shows the von Mises stress distribution, where the maximum stress value reached 54.24 MPa, concentrated near the inner corner of the bracket.

This corresponds to a safety factor of approximately 5.07 (275 MPa / 54.24 MPa).

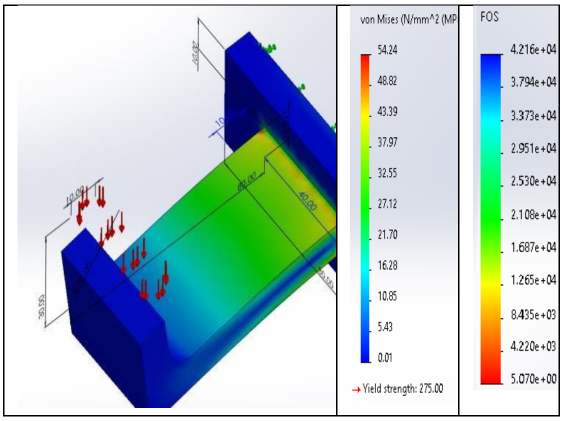

Fig 4. shows the von Mises stress distribution.

Figure 5 presents the total deformation, which reached 0.38 mm at the bracket's free end.

These values served as benchmarks for evaluating the performance of the optimized model.

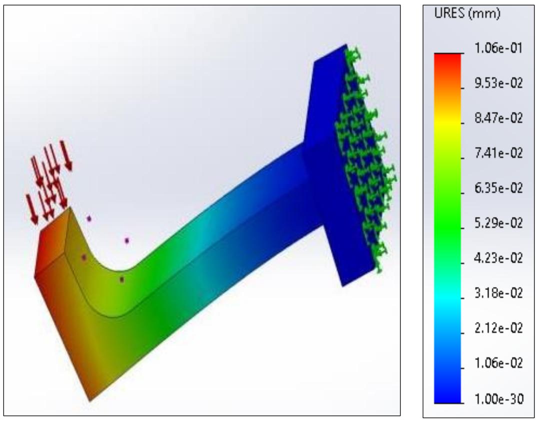

Fig5. Baseline FEA results for the initial design of total deformation (mm).

4.2 Topology Optimization Results

The topology optimization solver generated the material distribution plot shown in Figure 6. Blue areas represent regions of material removal, while red and white zones indicate load-bearing areas required to maintain strength and stiffness. The optimization produced an organic, truss-like structure that efficiently transferred the applied load to the support region. The resulting material reduction was approximately 54.74%, achieving a significant weight saving while maintaining structural efficiency

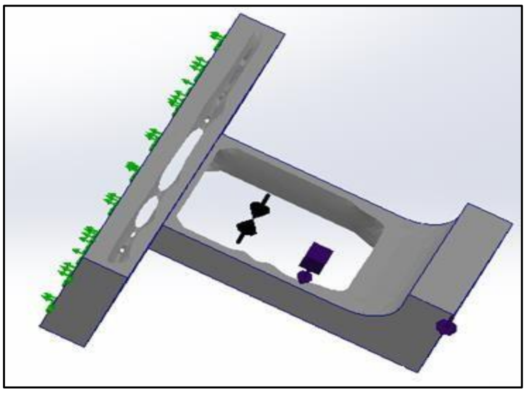

Fig 6. Topology optimization result.

4.3 Redesigned Geometry and Validation

After interpreting the topology optimization results, the geometry was reconstructed in SolidWorks, and a validation FEA was performed under identical boundary conditions, showing:

- Maximum von Mises stress: 97 MPa,
- Maximum deformation: 0.1616 mm,
- Final mass: 0.229 kg.

Table 2. Comparison of Performance Metrics.

Parameter	Initial Design	Optimized Design	% Change
Mass (kg)	0.506	0.229	- 54.74
Deformation (mm)	0.38	0.1616	- 57.4
Max Stress(MPa)	54.24	97	+ 78.9
Safety Factor	5.07	2.834	- 44.2

4.4 Discussion

The topology optimization revealed that nearly half of the original material was structurally redundant, confirming the high efficiency of the optimized design.

Although the maximum stress increased, it remained well below the material's yield strength (275 MPa), ensuring safety. The new geometry displayed an improved stiffness-to-weight ratio and more efficient stress flow through the bracket arms. These results demonstrate that topology optimization in SolidWorks Simulation can effectively balance weight reduction and structural performance, validating its applicability for mechanical design improvement in industrial settings.

5. Conclusion

This study successfully demonstrated an integrated CAD–FEA workflow for the topology optimization of an L-bracket using SolidWorks Simulation 2025. The approach resulted in a 54.74% reduction in mass, while keeping the maximum von Mises stress (97 MPa) well below the material's yield strength of 275 MPa, ensuring structural safety. The main conclusions drawn from this work are as follows:

1. Topology optimization is a highly effective method for achieving significant weight reduction in mechanical components without compromising essential performance characteristics.
2. The integration between CAD and CAE tools creates a closed-loop process — the initial FEA establishes the baseline, the optimization defines the ideal structure, and the final validation confirms the design's efficiency.
3. Although optimized geometries may appear complex, their manufacturability through additive manufacturing or advanced casting must be considered as a design constraint.
4. The study reinforces the potential of SolidWorks Simulation as a comprehensive environment for design, analysis, and optimization within modern engineering workflows. Future work may focus on incorporating manufacturing constraints such as draw direction and overhang limitations for 3D printing and validating results through experimental prototyping.

6. References

[1] Gao, B., et al. (2024). Topology optimization of the bracket structure in the acquisition, pointing, and tracking system considering displacement and key point stress constraints. Aerospace, 11(11), 939. https://doi.org/10.3390/aerospace11110939

[2] Lee, C., Yoo, J., & Lee, I. (2021). Smoothed-strain approach to topology optimization. Journal of Computational Design and Engineering, 8(5), 1267–1282. https://doi.org/10.1093/jcde/qwab060

[3] Kambampati, S., Martins, J. R. R. A., & Hicken, J. E. (2020). Level set topology optimization of structures under stress and thermal loads. Retrieved from https://openmdao.org/pubs/kambampati_topology_2020.pdf

[4] Sun, W., Zhang, S., & Liu, Z. (2024). Fail-safe topology optimization using damage scenario penalty. Applied Sciences, 14(2), 878. https://doi.org/10.3390/app14020878

[5] Zhu, J.-H., Zhou, H., Wang, C., Zhou, L., Yuan, S.-Q., & Zhang, W.-H. (2021). A review of topology optimization for additive manufacturing: Status and challenges. Chinese Journal of Aeronautics, 34(1), 91–110. https://doi.org/10.1016/j.cja.2020.07.011

[6] Tang, T., et al. (2024). Topology optimization: A review for structural designs based on linear and nonlinear elasticity. Retrieved from https://pmc.ncbi.nlm.nih.gov/articles/PMC11643539/