DESIGN OPTIMIZATION USING FEM FOR A CARGO FLOOR STRUT

Andrei BENCZE, George Răzvan BUICAN

Transilvania University of Braşov, Romania (andrei.bencze@unitbv.ro, buican.george@unitbv.ro)

DOI: 10.19062/2247-3173.2025.26.13

Abstract: *This paper presents an applied study for topological optimization, a modern engineering design technique that aims to optimally distribute the material in a given area, in order to obtain a structure that is as light and mechanically efficient. The first part of the paper analyzes the general concepts of topological optimization, such as the basic principles, common constraints and objective criteria used in engineering applications. The case study focuses on an aircraft cargo floor strut, a structural element frequently encountered in aeronautical applications. The optimization was performed using the OptiStruct solver developed by Altair Engineering, which offers advanced structural analysis and optimization capabilities. The pre-processing and boundary condition definition steps were performed in HyperMesh, while the analysis of the results and visualization of the optimal material distribution were performed with HyperView. The obtained results demonstrate a reduction in the mass of the component, while maintaining structural performance within the specified limits, thus validating the efficiency of the method used in the optimized design process.*

Keywords: *topology, strut, optimization, finite element method, stress*

1. INTRODUCTION

In recent years, the aerospace industry has witnessed a transformation in the design and manufacturing methods of structural components. The main goals related to weight reduction, efficiency and improved structural performance have led to the adoption of advanced design optimization techniques, especially in the conceptual phase of product development. Among these methods, topology optimization has emerged as an essential tool in modern engineering [1].

The topological optimization is a computational method that aims to determine the optimal distribution of material in a given area, in order to meet objectives such as maximum stiffness, minimum mass or a certain safety factor [2]. Unlike traditional shape optimization methods, which operate on a predetermined geometry, topology optimization does not assume a fixed initial shape, but allows for the achievement of innovative and efficient configurations, often impossible to imagine by classical means [3]. This approach is increasingly used in the design of aerospace components due to its ability to reduce the mass of structures without compromising mechanical performance.

This paper analyzes a practical example of the application of topological optimization in an aerospace context: the design of a cargo floor strut (Fig. 1), a structural component in supporting the floor of an aircraft cargo compartment. This type of element is subjected to significant mechanical loading and has an important role in transferring loads from the floor to the fuselage [4].

Therefore, optimizing its weight can contribute to reducing the total mass of the aircraft and, implicitly, increasing flight efficiency.

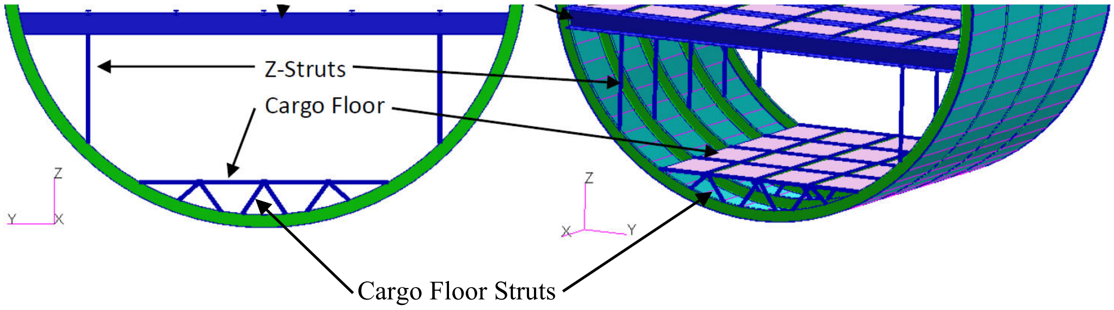


FIG. 1 Cargo Floor Strut fixing the Cargo Floor Crossbeam on the Aircraft Frames.

The analysis was done in the Altair HyperWorks software package. HyperMesh was used for pre-processing: definition of the geometric model, discretization of the domain with finite elements and application of relevant boundary conditions. Optimization was executed with OptiStruct, a specialized solver for structural analysis and optimization, which allows obtaining the material distribution based on specified criteria [5]. Post-processing was carried out in HyperView, for analyzing the numerical results and visualizing the optimized model, highlighting the areas of interest from a structural point of view.

One of the goals of this paper is to demonstrate the applicability of the topology optimization method in the real design of an aerospace component, as well as the integration of this method into the digital engineering development chain. The results are also interpreted in the context of manufacturing possibilities, taking into account technologies such as 3D metal printing – a technology increasingly present in the aerospace sector due to its compatibility with optimized geometries [6].

The present study exemplifies the benefits through a concrete case, in which simulations are used to obtain a functional design, optimized and compatible with modern manufacturing methods.

2. STRUCTURAL OPTIMIZATION METHODOLOGIES

Computer-aided design (CAD) and Finite element method (FEM) have experienced an accelerated development with the advancement of optimization methods, which allow to obtain mechanically, economically and functionally efficient configurations. In the aerospace engineering, where mass reduction and stiffness increase are essential priorities, structural optimization methods are used at all stages of the product development process.

2.1 Types of structural optimization methods. Structural optimization methods are classified according to the level of freedom offered in modifying the shape or material of the structure. The most common categories are:

- size optimization – adjusts the cross-sectional dimensions (thicknesses, heights, etc.) of existing components to meet certain requirements (e.g. minimum mass, maximum allowable deformation) [7].

- shape optimization – modifies the geometric contour of a structure while maintaining topological connectivity (e.g. rounding corners to reduce stress concentrations) [8].

- topological optimization – allows the complete addition or removal of material within the structural domain, without initial constraints related to geometry. It is the most flexible and innovative method, used in the conceptual phases of design [1].

2.2 General principles of topological optimization. Topological optimization aims to determine the ideal distribution of material in a given space in order to maximize structural performance under the defined constraints. The problem is mathematically formulated as an objective function (e.g. maximum stiffness or minimum mass) under mechanical (displacements, stresses), geometric and technological constraints [9].

A classic approach is the SIMP (Solid Isotropic Material with Penalization) method, which involves relaxing the binarity of the material (0 – empty, 1 – full) by introducing a continuous density that varies between 0 and 1. The penalty has the role of forcing the solution towards binary values, avoiding areas of "ghost" material [10].

This method has been successfully extended to complex engineering problems, including in the aeronautical field, offering new shapes of components, adapted for additive manufacturing. Its advantages include: reduction of the mass of components by up to 30–50% without compromising strength, adaptability to various types of loads (static, dynamic, vibrations), possibility of integration into modern manufacturing flows (e.g. 3D printing) [11].

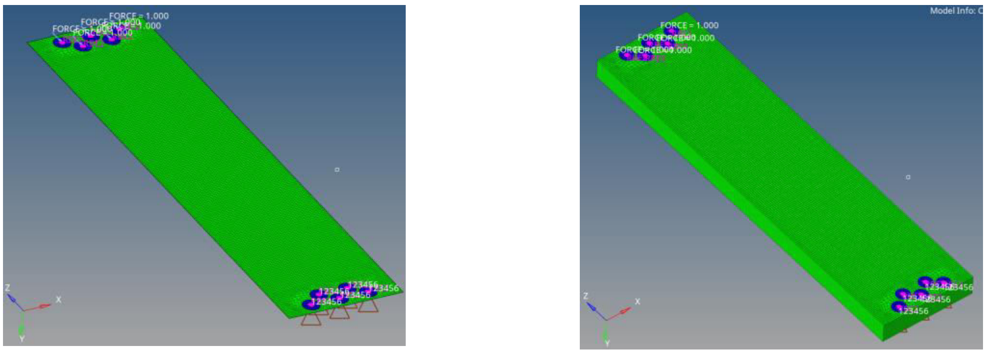
However, topological optimization also presents some challenges, such as the interpretation of numerical solutions, the generation of the post-processed geometry and the technological limitations associated with manufacturing.

2.3 Application of the method in Altair HyperMesh and OptiStruct. In this paper, the topological optimization was performed using the Altair HyperWorks software suite, more specifically the modules HyperMesh (for generating the finite element meshing, defining material properties and applying boundary conditions and loads), OptiStruct (solver for the structural analysis and calculation of the optimal material distribution within the defined volume), HyperView (for visual interpretation of the results and verification of the structural behavior of the optimized solution) [12]. The SIMP method is used to obtain a well-defined solution [13].

The resulting shape can be subsequently used for CAD reconstruction and preparation for manufacturing using light metal alloys [14].

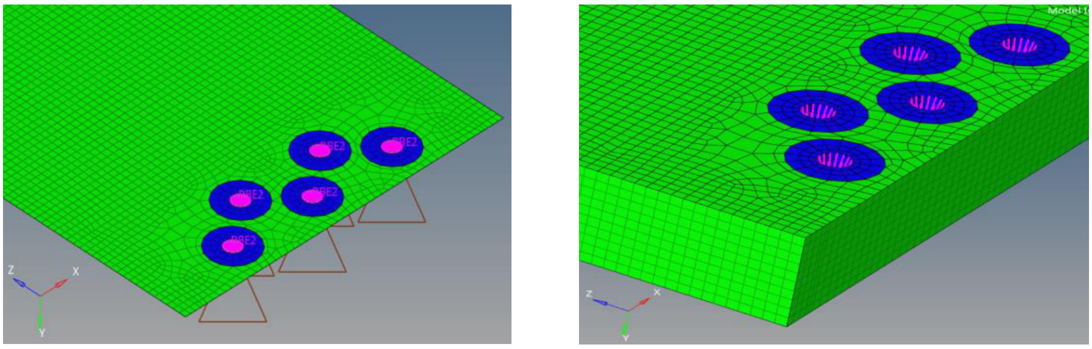
3. STRUCTURAL ANALYSIS, VALIDATION AND COMPARISON OF MODELS

To validate the topology optimization method applied in the design of a cargo floor strut, two separate models were developed and analyzed – one two-dimensional (2D) and one three-dimensional (3D), as presented in Fig. 2 a) and b).



a) 2D model with Shell elements   b) 3D model with tetrahedral elements

FIG. 2 Models used for the topological optimization of the cargo floor strut.



a) 2D model with Shell elements b) 3D model with tetrahedral elements

FIG. 2 (CONT.) Models used for the topological optimization of the cargo floor strut.

These were compared with two additional models (one with 2D shell elements and one with 3D tetrahedral elements), based on the real geometry of a strut used in industry. Figure 3 presents the sectional geometry of the real strut. The height of the extrude U profile is 400 mm.

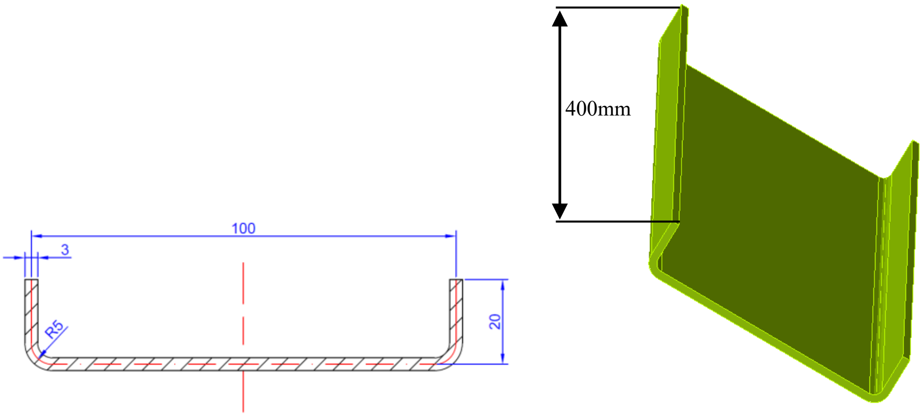


FIG. 3 Sectional geometry (U section) and extruded profile of the real cargo floor strut.

For the additional base models, for the one with 2D elements, the discretization was done on the middle surface, while the 3D model is done with a solid volume (an extruded profile). All models have the same geometry for the holes, located as presented in the 3D model of the real part, as shown in Fig. 4. In all models, around the hole was created a region to account for bearing stresses. This region will remain as is and is not part of the design space that is being optimized.

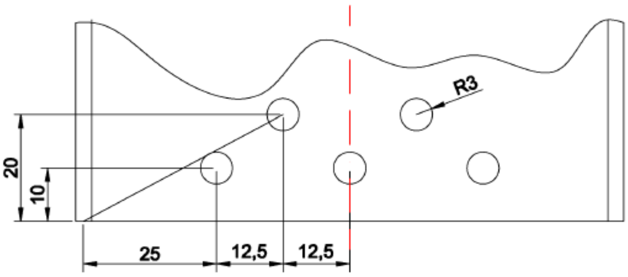
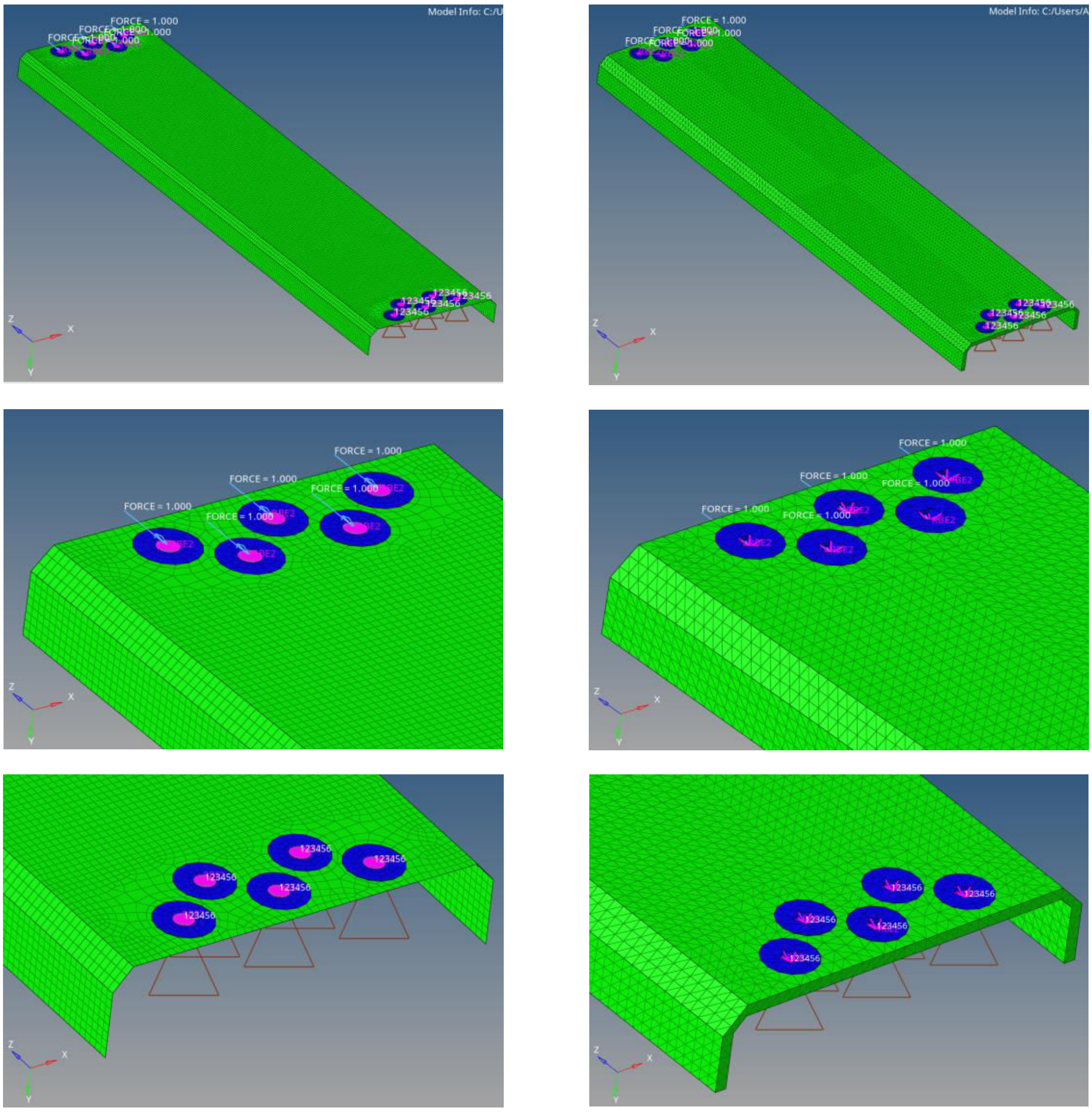


FIG. 4 Fastener holes location and geometry.

The purpose of this multiple analysis is to evaluate the mechanical behavior of the optimized shape and to identify the performance differences between the algorithmically generated solution and the classical component.

Figure 5 presents the two base models (real part model) of the cargo floor strut (one with 2D elements and the other with 3D elements).



a) 2D model with Shell elements b) 3D model with tetrahedral elements

FIG. 5 Base (real part) models of the cargo floor strut.

3.1 Description of the analysis domain and boundary conditions. The design volume, common to both the 2D and 3D models, consists of a rectangular plate with dimensions: length 400 mm, width 100 mm and thickness 50 mm. This volume was considered as the initial structural domain for optimization. Circular holes were defined in the upper and lower areas of the plate that simulate the attachment points with the rest of the structural assembly of the aircraft fuselage: cargo floor crossbeam (at the upper side) and frame (at the lower side) [15].

The fastening system at each extremity involves two rows of bolts, one with 3 and another one with 2 bolts. These were modeled for boundary conditions. The applied load is a 25kN vertical force plus a 2.5kN lateral force distributed between the upper 5 fasteners, simulating the pressure transmitted by the weight of the cargo during the operation of the aircraft. The lower 5 bolts are used to fix the model in all six degrees of freedom.

3.2 Description of the models. The 2D model was created in Altair HyperMesh by discretizing the design volume in a mid-thickness plane, using shell-type planar finite elements. This model allows for a fast preliminary analysis, with a reduced consumption of computational resources. The uniform thickness of 50 mm was considered constant at the beginning of the optimization analysis, and the loads and constraints were applied in the corresponding 2D plane [16].

The topological optimization was applied directly on the two-dimensional domain, resulting in an optimal material distribution, with full preservation of the clamping and loading zones. The optimized model was subsequently subjected to a linear static analysis to determine the maximum deformations and stresses.

For a realistic simulation, a three-dimensional model was also created, using tetrahedral solid elements. This provides a complete representation of the stress distribution in the volume, essential for the evaluation of the parts subjected to applied loads. In this case, the working volume was completely discretized, and the topological optimization was carried out in the same domain, using the SIMP method implemented in the OptiStruct solver [17].

Compared to the 2D model, the three-dimensional simulation involves a considerably longer calculation time, but provides much more detailed results, especially in the stress concentration areas around the clamping holes.

The separate 2D and the 3D models of the cargo floor strut are meshed with an appropriate mesh density to correctly capture stress variations. The support conditions corresponding to the real-world fastenings were applied, and the applied load was defined as a force distributed at the upper side fasteners, simulating the weight of the cargo transmitted through the cargo compartment floor.

To validate the results obtained from the topological optimization, the 2D and the 3D base models are used, representing the geometry of a real strut with a classical U cross-section. These models were analyzed under the same loading and fixing conditions. The comparison was made in terms of the following parameters: mass, stress distribution, deformation [18].

OptiStruct allows the definition of design regions (optimizable areas) and non-design regions (areas that cannot be modified, such as fastening points). A minimization of mass objective was used, with a stress constraint of maximum 300 MPa.

3.3 FEM Results. The optimization results were analyzed in HyperView. The material density distribution showed the formation of an internal reticular structure, indicating a mass reduction without loss of stiffness. The material concentration areas correspond to internal force paths.

In the following figures 6 to 13, the results for all 4 models are presented:

- Base models:
  - the 2D base model
  - the 3D base model
- Optimized models:
  - the 2D optimized model
  - the 3D optimized model.

It is evident that the optimized structures manage to keep the stress flow in a coherent shape, with reduced concentrations and a more efficient use of material.

Note that for the optimized models, the symmetry condition was imposed, since the lateral load can be applied from either side.

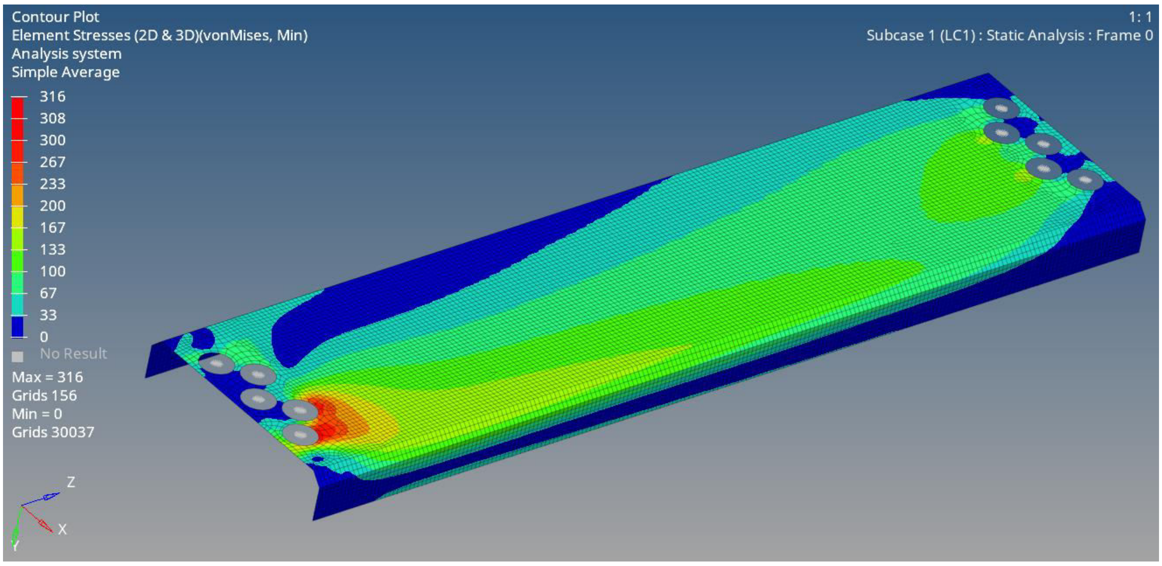


FIG. 6 2D Base (real part) model results – von Mises stress distribution.

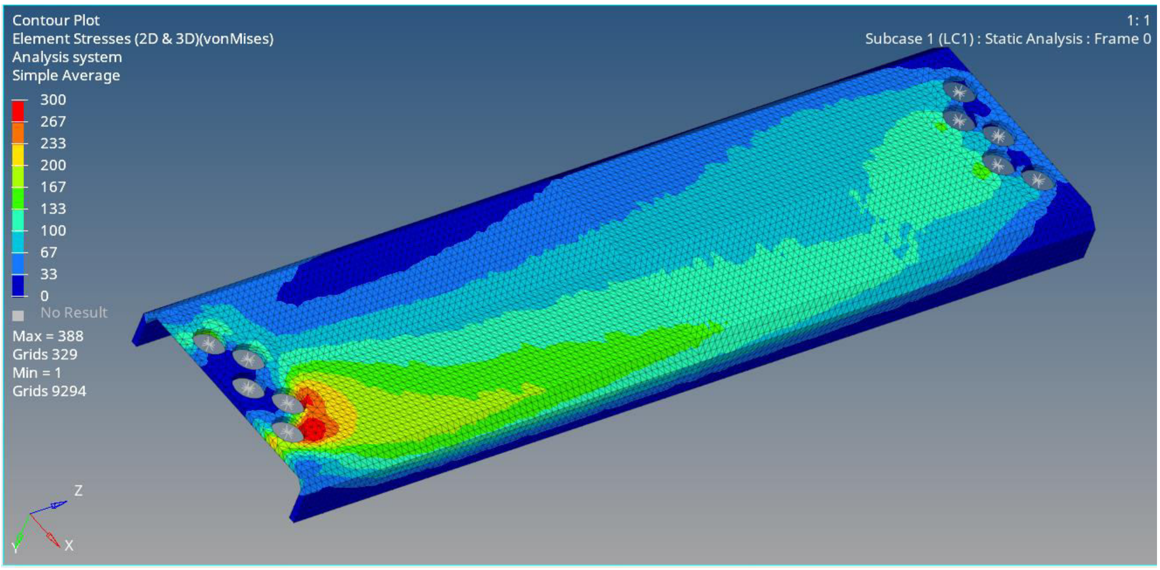


FIG. 7 3D Base (real part) model results – von Mises stress distribution.

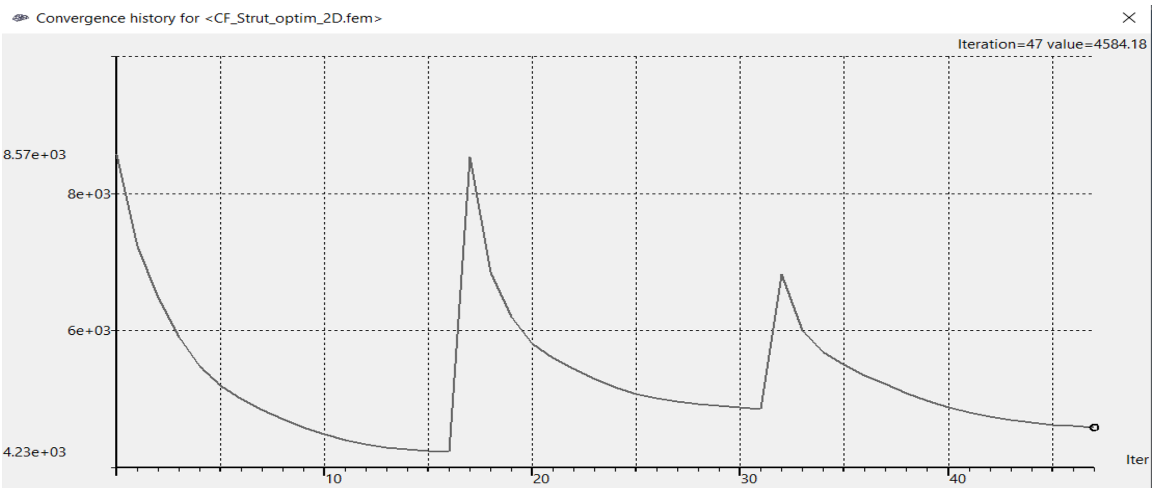


FIG. 8 2D optimization model iteration histogram.

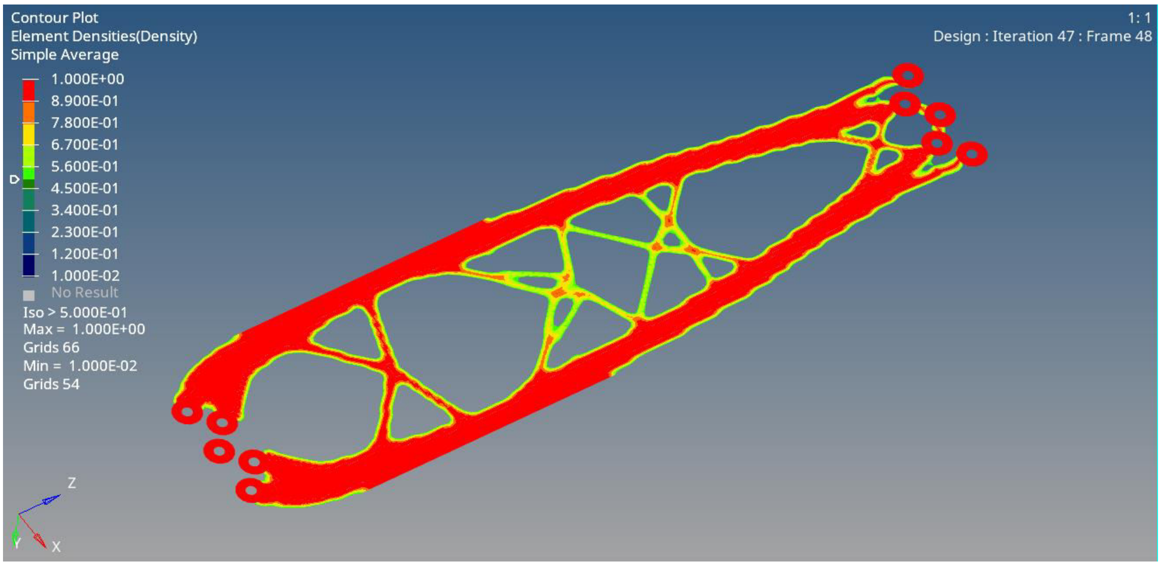


FIG. 9 2D optimization model shape (cut-off at 0.5 element density).

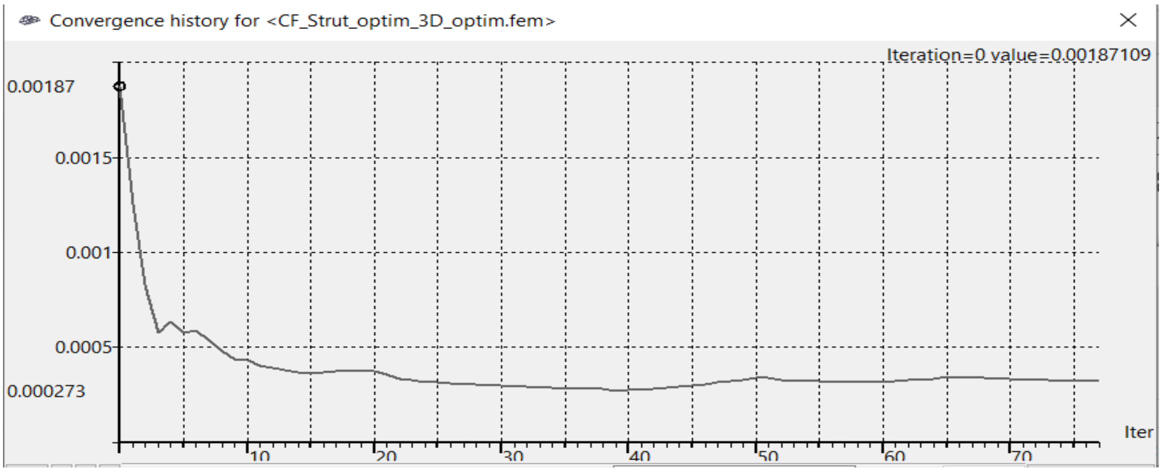


FIG. 10 3D optimization iteration histogram.

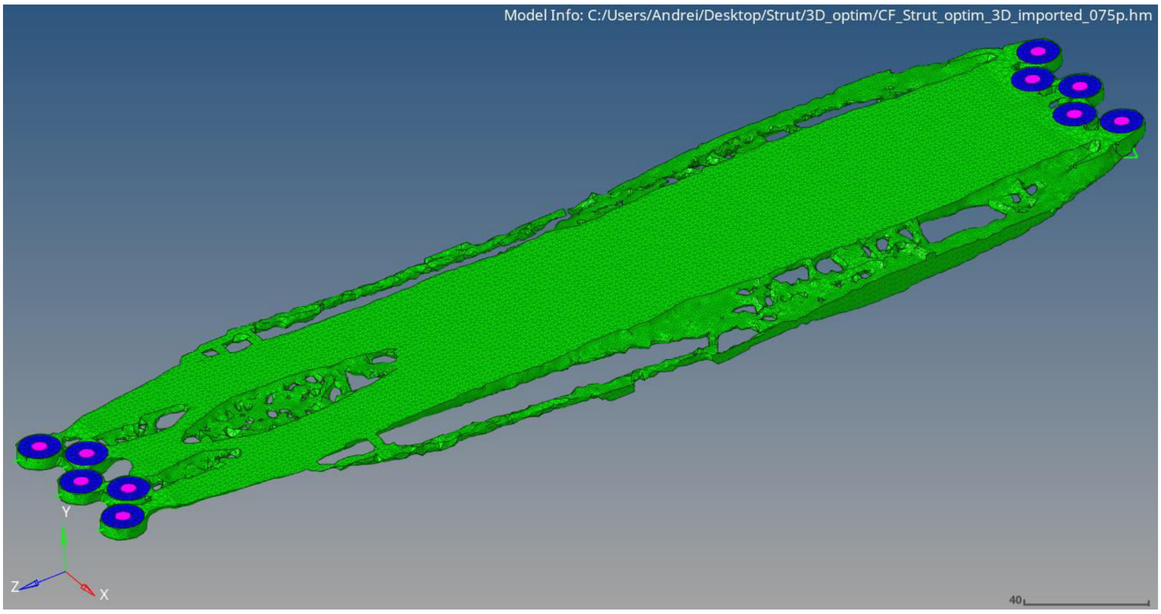


FIG. 11 3D optimization model shape (cut-off at 0.4 element density).

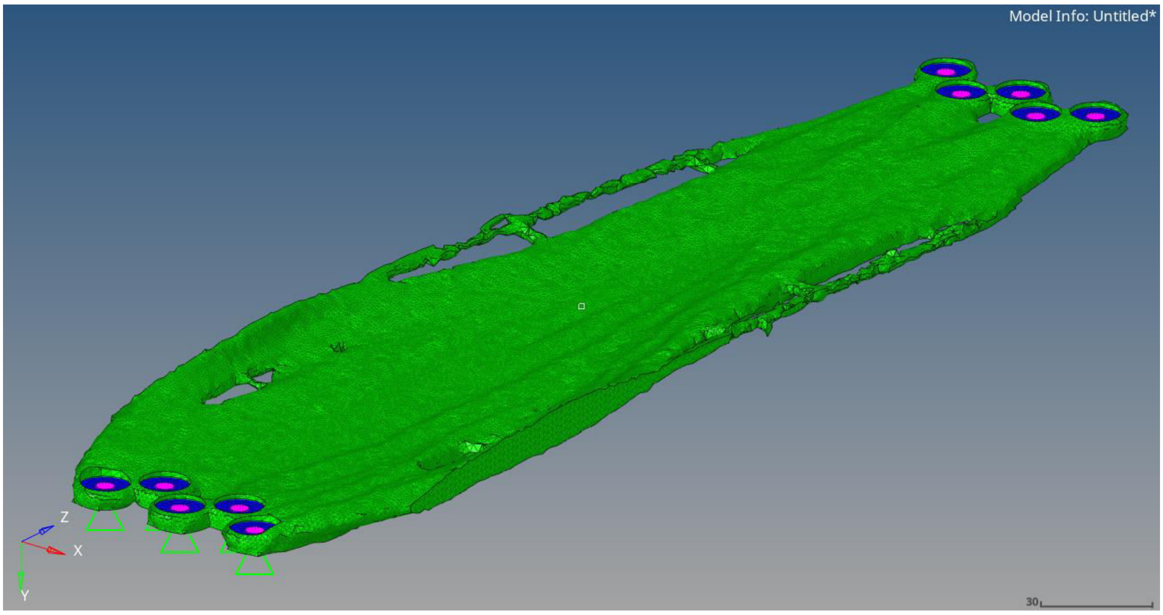


FIG. 12 3D optimization model shape (cut-off at 0.5 element density).

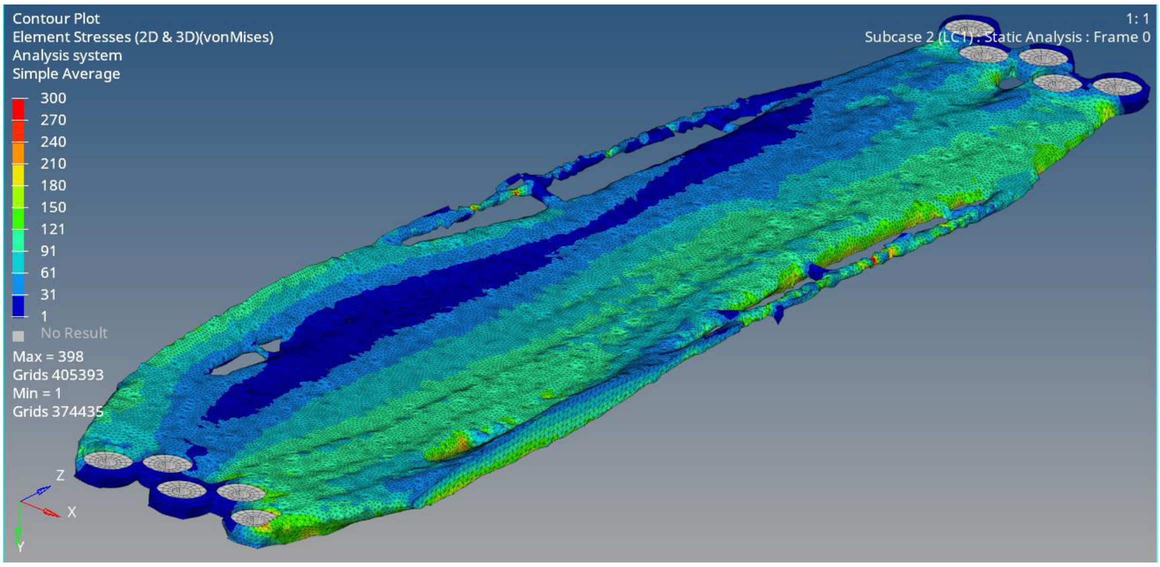


FIG. 13 3D optimization model results – von Mises stress distribution.

The comparative analysis between the two real models (base models) and the two optimized models indicates that the topology optimization method can generate more efficient structural solutions, reducing mass and improving stress distribution. The 3D model provides more accurate and detailed results, but requires more computational resources. The 2D model is suitable for quick assessments and for the early design phases.

4. CONCLUSIONS AND REMARKS

The present study shows the benefit of using topological optimization in the design of lightweight structural components for the aerospace industry, with direct applicability to a cargo floor strut. The chosen method (topology optimization using OptiStruct solver from the Altair suite) permitted the reducing of the mass of components without compromising the mechanical performance.

The 2D and 3D models developed for the simulation demonstrates that by correctly applying constraints, loads and boundary conditions, optimized geometries can be obtained. Comparative results between the optimized models and the real component indicate mass reductions of up to 40-50%, while maintaining the stress distribution and the deformation within acceptable limits. Moreover, the three-dimensional approach has led to more precise and adaptable solutions for subsequent manufacturing.

In the long term, the integration of computational optimization technologies in the initial design phases will lead to significant material, cost and weight savings in complex structural assemblies. Also, the use of these methods allows for an efficient use of the design space, providing new solutions that cannot be achieved by conventional design methods.

In conclusion, the paper demonstrates that topological optimization is a useful solution for improving structural design in aviation.

REFERENCES

[1] M. P. Bendsøe and O. Sigmund, *Topology Optimization: Theory, Methods and Applications*, Berlin, Germany, Springer-Verlag, 2nd ed., 2003;

[2] R. Rozvany, A critical review of established methods of structural topology optimization, *Structural and Multidisciplinary Optimization*, vol. 37, no. 3, pp. 217–237, 2009;

[3] J.D. Deaton and R.V. Grandhi, A survey of structural and multidisciplinary continuum topology optimization: post 2000, *Structural and Multidisciplinary Optimization*, vol. 49, no. 1, pp. 1–38, 2014;

[4] Raghavan, et al., *Structural optimization of aircraft cargo floor beam using topology optimization*, AIAA Scitech Forum, 2018;

[5] *** Altair Engineering Inc., *OptiStruct User's Guide, v2022.1*, 2022. Available at www.altair.com, accessed on April 25, 2025;

[6] A.W. Gebisa and H.G. Lemu, Additive manufacturing for the aerospace industry: a review, *International Journal of Aerospace Engineering*, 2018;

[7] R.T. Haftka and Z. Gürdal, Elements of Structural Optimization, Springer, Berlin, 1992;

[8] N. Olhoff and M.P. Bendsøe, Optimization of structural topology, shape and material, *Structural and Multidisciplinary Optimization*, vol. 21, no. 2, pp. 81–94, 2001;

[9] M. Zhou and G.I.N. Rozvany, The COC algorithm, Part II: Topological, geometrical and generalized shape optimization, *Computer Methods in Applied Mechanics and Engineering*, vol. 89, no. 1-3, pp. 309–336, 1991;

[10] O. Sigmund, A 99 line topology optimization code written in Matlab, *Structural and Multidisciplinary Optimization*, vol. 21, no. 2, pp. 120–127, 2001;

[11] J. Liu and Y. Ma, A survey of manufacturing oriented topology optimization methods, *Advances in Engineering Software*, vol. 100, pp. 161–175, 2016;

[12] *** Altair Engineering Inc., *HyperMesh and OptiStruct Documentation*, 2022. Available at www.altair.com, accessed on April 25, 2025;

[13] W. Zhang and Z. Kang, Topology optimization for additive manufacturing: A review, *Micromachines*, vol. 9, no. 11, p. 540, 2018;

[14] W.E. Frazier, Metal additive manufacturing: A review, *Journal of Materials Engineering and Performance*, vol. 23, no. 6, pp. 1917–1928, 2014;

[15] M.C.Y. Niu, Airframe Structural Design, Conmilit Press, Hong Kong, 1999;

[16] *** Altair Engineering Inc., *HyperMesh User's Guide, v2022.1,* 2022. Available at www.altair.com, accessed on April 25, 2025;

[17] *** Altair Engineering Inc., *OptiStruct Solver Reference Guide*, 2022. Altair. Available at www.altair.com, accessed on April 25, 2025;

[18] R.J. Yang and A.I. Chahande, Topology optimization for crashworthiness, *Structural and Multidisciplinary Optimization*, vol. 20, no. 1, pp. 21–28, 2000.