TOPOLOGY OPTIMIZATION METHOD OF AIRCRAFT WING RIB USING ANSYS

Mr. SAMUEL ISSAIAS HAILE[1], Dr.G BHASKARA RAO[2]

1. Lecturer, Department of Mechanical Engineering Mai-Nefhi College of Engineering and Technology,Asmara, Eritrea. E-mail address: samuisayhie1295@gmail.com

2. Lecturer, Department of Mechanical Engineering, Mai-Nefhi College of Engineering and Technology, Asmara, Eritrea. E-mail address: raoganji@gmail.com

Abstract : With the continuous development of science and technology, while ensuring safety and functionality, reducing aircraft life cycle energy consumption and reducing aircraft operating costs are becoming a hot topic in many researches. Starting from the quality of the fuselage, the research performance is up to the standard. The issue of reducing the mass of aircraft parts and ultimately reducing the mass of the entire aircraft is causing widespread concern in the industry. In response to this requirement, this paper selects a certain type of aircraft wing rib as the object and studies the 'topology optimization design' method of the aircraft rib structure. First, the three-dimensional model of the wing rib is established with CATIA software as the basis of finite element analysis and topology optimization design. Then in ANSYS, perform finite element analysis and structural topology optimization of the top full-contact force basic wing rib. The finite element analysis results of stress, strain distribution of the basic wing rib model and the 'topology optimization design' model are obtained. Further, the influence of the wing stringer on the wing rib was considered, and the real model finite element analysis and structural topology optimization of the wing rib under the top part of the area were carried out in ANSYS, A more realistic finite element result of the wing rib model and a topological optimization design model are obtained. In this thesis, by applying topology optimization to the design of aircraft wing ribs, and comparing its performance with the original design model, this paper actively discusses the use of topology optimization to guide the optimization design of aircraft components. After optimization in this paper, this type of wing rib has been effectively reduced in mass while ensuring structural strength and safety, which has a positive significance in reducing the weight of the whole machine and saving operating costs.

Keywords: Topology Optimization (TO), Wing Rib, Finite Element Analysis, ANSYS Optimization

1. Introduction

In this current economically and ecologically challenged environment, TO could perhaps be the key to a new era of engineering, facing these challenges. Aircraft companies are now urging their research teams towards solutions to tackle new environmental issues. The obvious solution is to reduce fuel consumption, and more generally reduce energy consumption. Making systems lighter and easier to manufacture would have this very effect, which is precisely what TO allows, coupled with AM. Lighter aircraft would consume less fuel and therefore use less energy. TO can help engineers face the new environmental and financial issues and demands of the modern aircraft industry.

In the aircraft industry, the optimization and simulation methods are recognized by the Computer-aided engineering for analysis skeleton based on the property of mesh model in finite element (FE) method [4].

Figure 1. Aircraft Wing and Conventional Rib

To obtain new and lightweight component design and demonstrate several systematical stages are used for any specified aircraft wing design. The development of the performance of strength, stiffness, and other condition stability of the wing rib based on weight reduction and increase in strength. There are different methods of optimization tools CAD/CAM, CAE (Computer-Aided Engineering) tools, Assemblies, CNC machining, ANSYS, FEM optimization, etc. The structural optimization of wing rib to make perform the overall aircraft wing which the dominant part of the wing ribs if reduce the weight rib the weight of the wing will reduce up to 50% of the weight reduction as compared to other components of the wing. However, optimization of the wing rib will reduce the manufacturing cost and assembling time consumption overall the wing. In the optimization adding loop optimization of aircraft wing rib, the improvement of the geometric design and structural analysis availability of the modules is the code of analysis performing the skeleton of the design. The objective optimization can model a wing-rib more automatically and rapidly in manufacturing assembly processing [5].

Topology optimization is concerned with minimizing compliance to formulate the structural design maximizing the fundamental usual frequency, buckling loads, or design of compliant mechanism among others to consider the strength a significant issue [6]. In the area of topology optimization, mainly the two elementary approaches have been tracked to include stress and volume constraints are on a finite element discretization of the problem it consists of the minimization of weight the structure subject to local reinforcements and global constraints.

1.1 Research Objectives

- To optimize the mass of the model based on the topology optimization method design analysis
- Develop a topology optimization algorithm based on the current trends
- Deploy a basic topology optimization algorithm
- Perform a topology optimization using the software ANSYS
- Compare the performances of the algorithms
- Conclude on the potential benefits of TO, as well as the feasibility and accessibility of such means

2. Research Methodology

The wing rib used as a sample of the study case, that will be optimized, needed to be clearly defined and modeled to run the initial analysis process. First, the design model of wing rib is done on CATIA V5, which will allow understanding the geometry features of the part that will influence the complication of the calculation later on and the general structural properties and original performances. Before optimizing, the design will analyze the load distribution over the area by applying the load conditions, which exerted the pressure stress on the wing rib at the maximum operation of aircraft. Based on the structural analysis and load distribution will be defined and organized to optimize the size and strength. Then, an algorithm will be implemented to perform TO on the part. The type of the algorithm should be wisely chosen considering the study case and available resources. In addition, a thorough theoretical and basic study of the said algorithm simulation should follow to verify its implementation quality. Next, the reference for topology optimization performances that is chosen for this study is the software ANSYS, which includes an optimization module. The optimization module of ANSYS should be studied deeply before performing any analysis. Because the objective is to compare procedure methods and results from a quality, each should be precisely and deeply mastered so to draw the most accurate conclusions, based on specific sets of parameters.

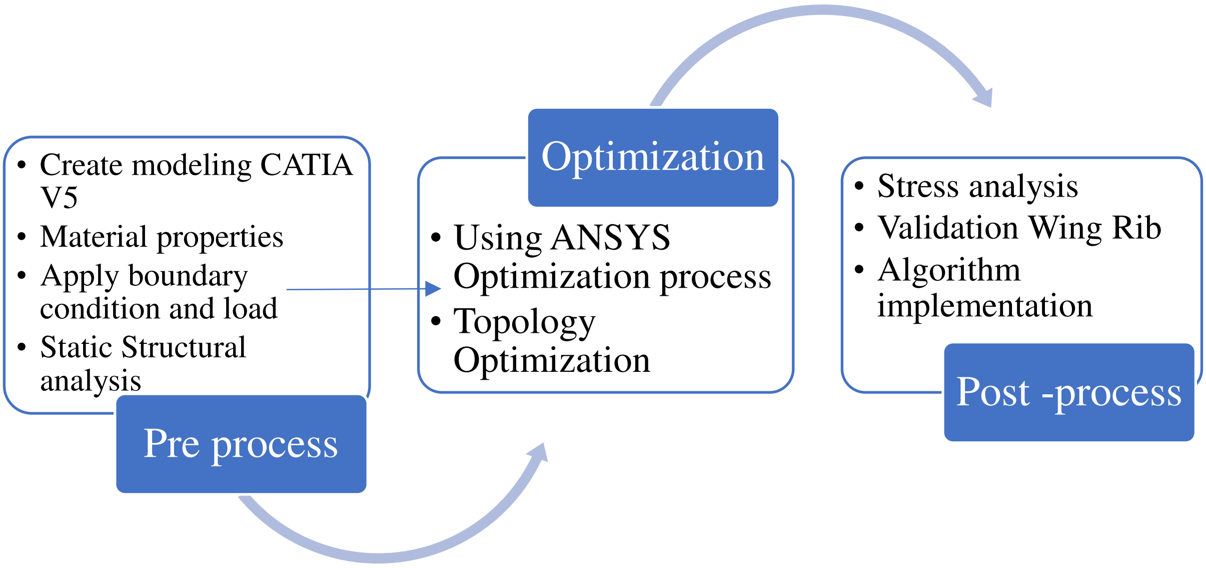


Figure 1. Research Methodology

3. Wing Rib Design

Wing rib plays an important role by providing aerodynamic shape for the wing, load transfer, distribution, and shear forces redistribution and it also limits the span of the stringers. The structural design analysis of aircraft on the advantage of material properties and selected the staid design parameters for the design approach used classical the 3D structural model is design in CATIA V5.

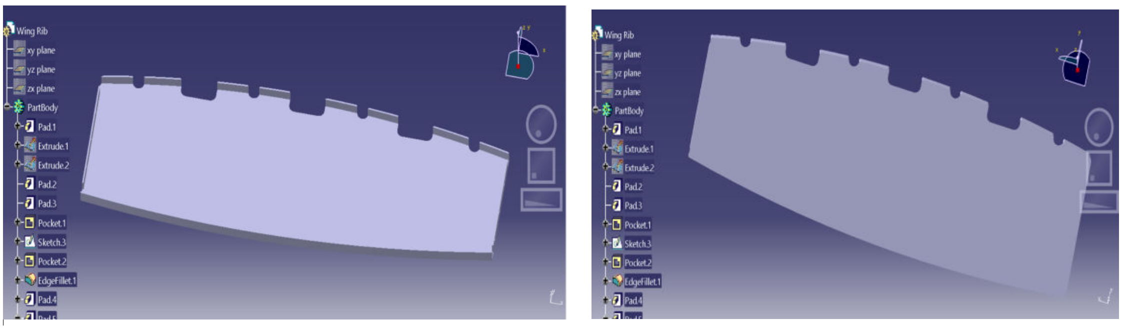


Figure 2 Wing Rib Model

Table 1. Dimensions of the Wing Rib

Property	Dimension (mm)
Rib Thickness	5
Length	1465.7
Folding Width	35
Front edge width	404.6
Rear edge width	345

3.1 Analysis of the Design

Analysis of the structure is completed using ANSYS software. Equivalent stress, strain energy, and displacement for the given structure are determined and characteristic graphs are obtained for various configurations stress. The meshing size used is 50mm regarding the dimensions of the parts are roughly equivalent. The force of 1KN is applied to the structures and the optimization will be 3D done, the structural analysis is conducted in the 3D model also, making the finite element method (FEM) analysis faster. The results get from the analysis are used to govern the Von-Mises stress in the structure and extreme displacements. The maximum Von-Mises stress occurs at the tip edges and displacements occur at the center of the structures. Topology optimization is conducted on the material Aluminum, which is common in aircraft structural design of components with the given physical and mechanical properties. The material used for wing rib is Aluminum Alloy. The key property of Aluminum alloy for the designing structural Analysis is summarized in Table 1. To understand the specification of the material more significant in industrial requirement definition manufacturing process. The material properties influence the outcomes of the optimization based on their input values. The most common imperative factor for an efficient structural analysis is the proper definition of the material properties.

Table 1. Aluminum Material Properties for Structural Analysis

Properties	Values	Properties	Values
Density	2770 Kgm^{-3}	Share Modulus	2.6692e10 Pa
Modulus of Elasticity	71000 MPa	Tensile Yield Strength	2.8e08 Pa
Poisson's Ratio	0.33	Tensile Ultimate Strength	3.1e08 Pa

Table 1. Summarized the values of material property for the FEM analysis basis. The value is the property to differ each material in their strength and stiffness during the optimization.

The continuous topology optimization design in solve problems to attaining an optimum solution for the material distribution without vanishing the elastic stiffness of the material subject to restrictions amount of material available is considered [13]. With the n number of elements consist of problems to attain in the design domain "Ω" for 2D or 3D structures and the subdomain "Ω_1" material filled with 1, the compliance minimization over the design domain satisfying the equation. The design variable in the function of the optimizing problem is defined on the design domain Ω.

The step of the procedure marks the boundary condition of the inclusive border of the interface condition, then identifies the problem through the objective function formulate to design purpose or allocate the sensors at the solution situation. The Optimization algorithm the flow of optimization for several iterations or groups of iteration by defining the internal design structure. The preparation of optimization problems related to the change only in the boundary domain, which afterwards spreads to the entire domain.The topology optimization of the part has formulated a final result for the manufacturing design description. The aircraft wing rib is optimized through the different steps of the design process and finalize for manufacturing. According to the optimization process, the rib will be of more quality and effective as compared to the original design, the model design is developed by increasing the thickness and reducing the surface volume. The FEA the basic condition defined by the optimization for the algorithm flow chart to analyze the product quality [14]. The following framework as shown in figure 1. is more helpful in the design of the model for tracking the path. There are three stages of optimization design procedure to the modular design of wing rib. The first Step considered topology optimization of the selected rib, which means the weight is minimalized subject to the density constraint. The load applied on the wing rib lengths supporting maintaining the design and the stress distribution on the topology shape for optimization of the weight the region where material exists along the periphery of the rib. The second Step involves modified the optimized design of the rib obtained through TO done in stage 1. The objective is to validate minimize the weight design subject to stress and deflection constraints in static structural. The topology-optimized rib has been upgraded by determining stress distribution and strain energy on the rib area. Third Step 3 validate the modified design of the ribs. Analysis of the stress, stiffness, and other required parameters help in the design of the rib to preserving the area and moment of inertia with available standard dimension. The minimum weight is accomplished through topology optimization on the rib to allocate non-dimensional analysis as the chord–length.

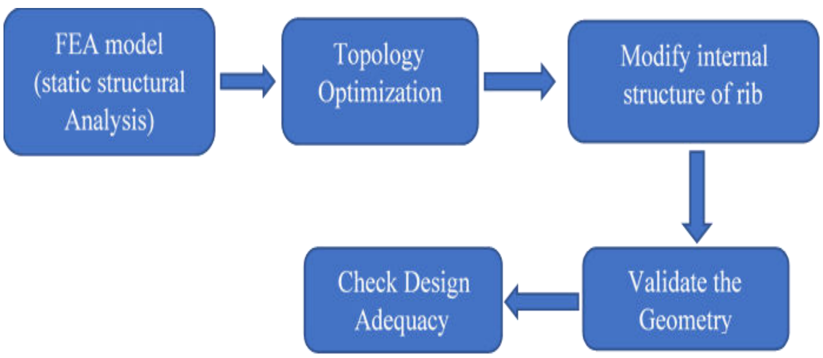
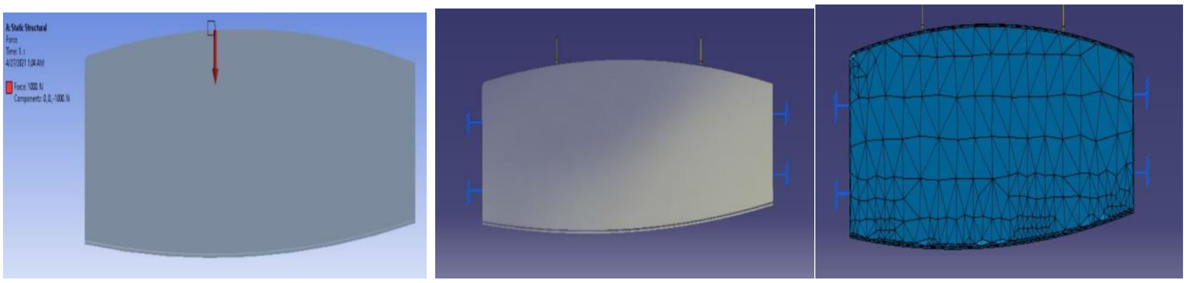


Figure 3.Optimization Design Framework

Topology optimization the method that works for design optimizing and reduce the weight of the overall wing of aircraft, and will be more qualified and high performance. Topology optimization led to proceeding with a 3D wing rib model to improve structural performance and maximizing the stiffness of the wing. The topology optimization has been generated in ANSYS software proceed an aircraft wing rib permitting in the analysis of stress and displacement.

3.2 Topology Optimization Test

The topology optimization is applied to the optimization of an aircraft wing rib to deal with reliability based on topology optimization problems. The optimization leads to a reduction of the weight of the design's simplified wing rib geometry. To verify the efficiency of the topology algorithm and procedure, a simplified wing rib is accessible in this section. The simplified wing rib illustrations of the models under CATIA are given in figure 3. The arrows represent the location of the forces applied on the top edge wing rib, and the top edge full as contact force area.



(a) The wing rib load in ANS (b) The wing rib load in CATIA (c) The Wing Rib with Mesh Design in CATIA

Figure 4. The Load Applied on Top Surface Contact Force Aluminum

In the CATIA-V5 software, several workbenches obtainable can be used for geometry automation and generation. The structural analysis of the stress in CATIA for the wing rib design under the applied load compression load the maximum stress at the tip of the edge.

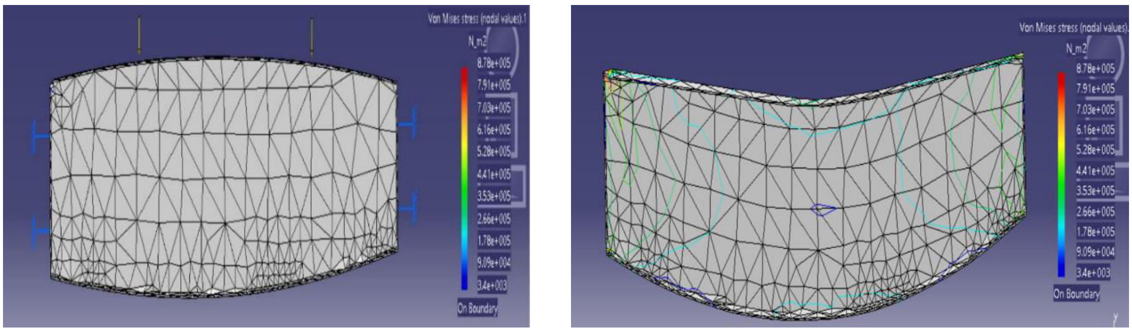
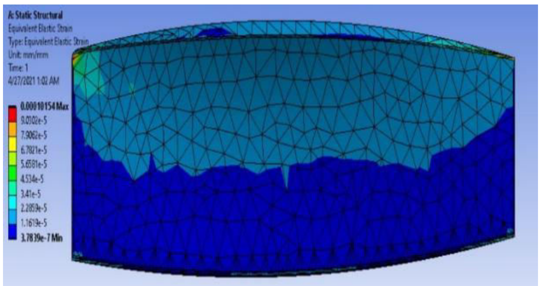


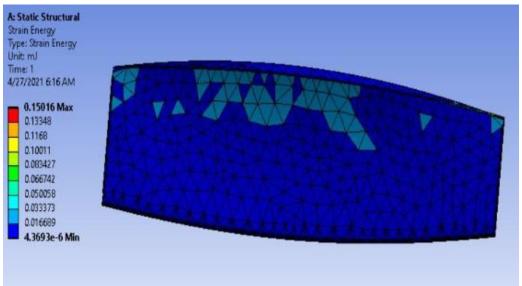
Figure 5. The Stress Result Obtained from Generative Analysis in CATIA

4. Finite Element Analysis of the Structural Static Case

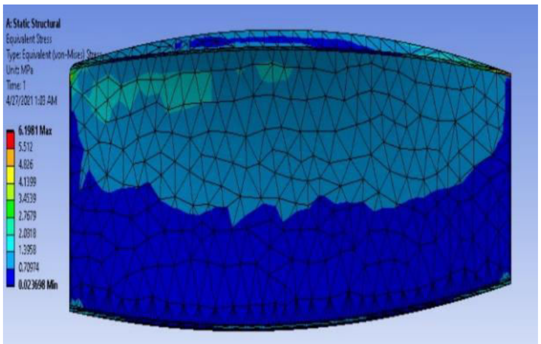
The material used for both parts is Aluminum, as summarized in Table 1, the key properties of Aluminum for static structural analysis. It is also important to understand that material specification which is useful for industrial requirements. If such requirements are not necessary, the material characteristics can be input arbitrarily and their values have little influence on the results.



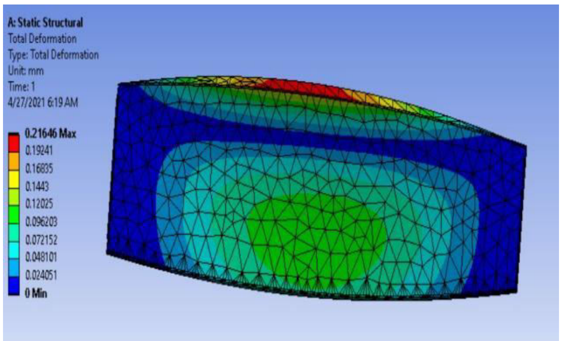
(a) Elastic strain of the wing rib



(b) Strain Energy of the wing rib



(c) Von-Mises Stress of the wing rib



(d) displacement of the wing rib

Figure 6: Finite Element Analysis of Wing Rib Analysis

For the overall procedure, the optimal point is reached when the mass of the part has been met or when the obtained result does not meet the requirements. The key element analyzed is the maximum displacement as it is usually one of the main requirements in engineering problems. FEM analysis is conducted to evaluate both the first objective function and the original maximum displacements. To prepare so, the load cases must first be defined.

The meshing used is defined in this study regarding the dimensions of the wing rib the default mesh is 77mm but for better analysis reduce the mesh size to 50mm used for the structure. The force applying as explained before 1000N is applied to the structures at the top of the edge. The analysis is 3D geometry results of the FEM structural analysis is conducted in ANSYS will be displayed in the contour view in the model. In addition, the results used in the structure are the Von-Mises stress, strain energy, elastic strain, and the maximum displacements. In this case, the maximum displacements occur in the tip of the edge wing rib the structures. Table 4.1 below is summarized the results of the first FEM analysis and figure 4.6 shows the contour view of the FEM analysis distribution over the model. The value considered here is the minimum weight of the wing rib.

4.1 Topology Optimization Results

Generally, the design more or less met towards one close to that of the obvious design. However, in some cases, the parameters of the optimization situation caused the result to be slightly different and therefore result exceptionally. As already mentioned, all the design parameters initially look to meet the obvious design. All of the matter in the area link between the top and bottom is removed at some stage in the optimization stages when using the procedure ANSYS topology optimization directly remove. The results indicate that the objective functions of the method showed the matter in this part, far from the connection, was non-important and could be removed. This is due to the stress distribution in the simplified wing rib under load. The result of the ANSYS topology optimization for the first phase of optimization is shown below in figure 8, with a mass reduction of 50% of the original model mass.

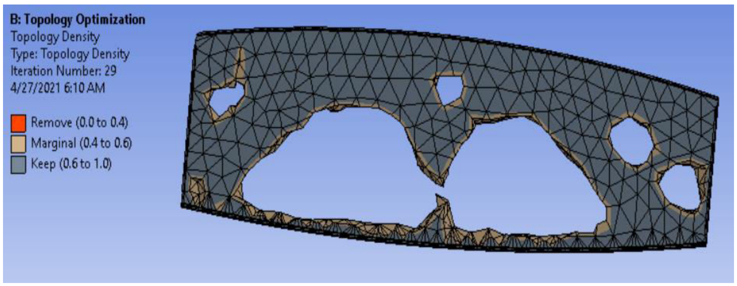


Figure 8: Topology Optimization Results

4.2 Validate the First Topology Optimization Model

Figure 8. shows the evolution of the design shapes after the redesign stages. In this case, required validation of the structure to reach the optimal design point and operation analysis of the FEM.

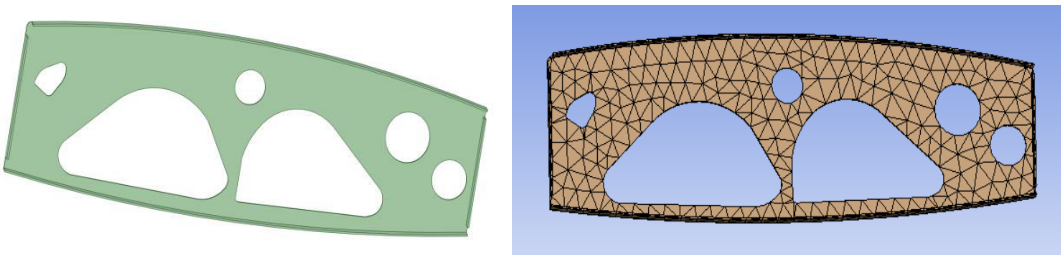
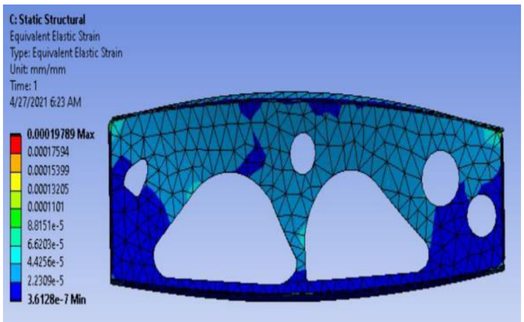
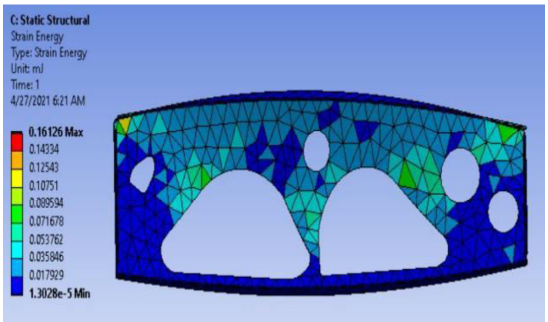


Figure 9: Modified Wing Rib Design

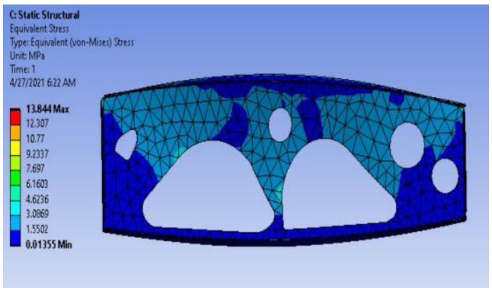
The static structural analysis methods used on the wing rib a procedure developed within this study for the optimization module included in the software ANSYS. The results obtained in the optimized model are shown in figure 9. below. The comparison of the model and optimized model the maximum of result under load and the evolution of the mass of the wing rib is summarized in table 2.



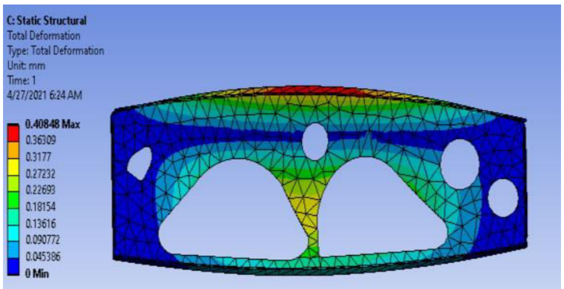
(a) Elastic strain of optimized wing rib



(b) Strain Energy of optimized wing rib



(c) Von-Mises Stress of optimized wing rib



(d) displacement of an optimized wing rib

Figure 10: Finite Element Structural Analysis Result of an Optimized Wing Rib

The elastic state of the structures is confirmed by looking at the maximum value of the stresses in the structure, which is always under the yield strength value of aluminum. The wing rib model is elastic property. In addition, from this first analysis, the parts can be run through the procedure. The boundary conditions were considered as follows: the load area should remain unmodified, and the fixed area could be modified as long as fixation is still possible and effective.

Table 2. Comparison of the Model Wing Rib and Optimized Case

Contents	Model	Optimized model
Weight (Kg)	10.807	7.3872
Max. Displacement (mm)	0.21646	0.40848
Max. Stress (MPa)	6.1981	13.844
Max. Elastic Strain (mm/mm)	1.0154e-4	1.9789e-4
Max. Strain Energy(mJ)	0.15016	0.16126

Subsequently comparing the models result mutually the value of analysis and conclude the optimized rib have less weight and acceptable in strength, wing rib strength is more important in the design of aircraft.

5. Conclusion of Topology Optimization Results

In this case, the results obtained with the top area of the wing rib as contact force illustrates the potential of TO in the simplified wing rib optimization. This procedure has achieved returning the design that gets the required design within the mass reduction of 35% of the original mass. This emphasis is important to achieve the best solution of the required design to the proposed application. To obtain the result of this emphasize analysis is using TO ANSYS software. The use of optimized parts in manufacturing has a major impact on the industry. TO parts, have smaller volumes ,which means the use of raw material can be decreased quite easily, making cost savings. As shown in figure 4.7 a basic TO, far from calculation power demanding, can easily return light and efficient parts. Furthermore, the optimized part also keeps its structural integrity, which means the mechanical requirements are still satisfied after the redesign. In the end, the optimization allows saving costs in terms of raw material use, and the parts can be used in the same way as their original design, without any consequence on the whole system. More precisely, the achieved mass reduction, all methods considered, is around 35%. That means that almost one-third of the material originally used can be saved. This represents a significant cost reduction in the manufacturing of the wing rib.

References

[1] G. K. Ananthasuresh, “Topology and Size Optimization of Modular Ribs in Aircraft Wings [J],” no. June, pp. 1–6, 2015.

[2] A. K. K. Soundarya S , Sathiyavani S, “Topology Optimization of Wing Rib in Cessna Citation [J],” *Int. J. Res. Aeronaut. Mech. Eng.*, vol. 6, no. 3, pp. 7–17, 2018.

[3] D. Walker, D. Liu, and A. Jennings, “Topology optimization of an aircraft wing [C],” *56th AIAA/ASCE/AHS/ASC Struct. Struct. Dyn. Mater. Conf.*, no. January, pp. 1–8, 2015.

[4] J. Tang, P. Xi, B. Zhang, and B. Hu, “A finite element parametric modeling technique of aircraft wing structures [J],” *Chinese J. Aeronaut.*, vol. 26, no. 5, pp. 1202–1210, 2013.

[5] J. Muthuraman, M. Shankar, and M. Vivekanandhan, “Design and Analysis of Wing Rib Using Finite Element Method [J],” *IOSR J. Eng.*, pp. 55–62, 2019.

[6] M. Bruggi and P. Duysinx, “Topology optimization for minimum weight with compliance and stress constraints [P],” pp. 369–384, 2012.

[7] D. Jankovics, H. Gohari, M. Tayefeh, and A. Barari, "Developing Topology Optimization with Additive Manufacturing Constraints in ANSYS® [C]," *IFAC-PapersOnLine*, vol. 51, no. 11, pp. 1359–1364, 2018.

[8] G. Kavya and M. T. C. A. D. Cam, "Design and Finite Element Analysis of Aircraft Wing Using Ribs and Spars," pp. 1443–1455.

[9] J. Muthuraman, M. Shankar, and M. Vivekanandhan, "Design and Analysis of Wing Rib Using Finite Element Method [J]," *Int. Conf. Emerg. Trends Eng. Technol. Res.*, pp. 55–62, 2019.

[10] M. A. M. B. M. Zakuan, A. Aabid, and S. A. Khan, "Modelling and structural analysis of three-dimensional wing [J]," *Int. J. Eng. Adv. Technol.*, vol. 9, no. 1, pp. 6820–6828, 2019.

[11] O. Sigmund, "A 99 line topology optimization code written in matlab,[J]," *Struct. Multidiscip. Optim.*, vol. 21, no. 2, pp. 120–127, 2001.

[12] K. Suresh, "A 199-line Matlab code for Pareto-optimal tracing in topology optimization [C]," *Struct. Multidiscip. Optim.*, vol. 42, no. 5, pp. 665–679, 2010.

[13] M. Stolpe and K. Svanberg, "An alternative interpolation scheme for minimum compliance topology optimization [J]," *Struct. Multidiscip. Optim.*, vol. 22, no. 2, pp. 116–124, 2001.

[14] V. Pugazhenthi, S. Gopalakannan, and R. Rajappan, "Finite Element Analysis of Composite Shell Structure of Aircraft Wing Using Composite Structure [J]," *Int. J. Eng. Sci.*, pp. 74–80, 2018.

[15] S. Bairavi, "Design And Stress Analysis of Aircraft Wing Rib With Various Cut Outs [J]," vol. 358, no. 4, pp. 511–514, 2016.

[16] A. Bharath, E. Keshav Kumar, V. M. Sreehari, and B. Ravikumar, "Design and analysis of an aircraft wing structure [J]," *Test Eng. Manag.*, vol. 82, no. June, pp. 6952–6958, 2020.